STUDY OF EFFECT OF CAVITATION IN KAPLAN TURBINE USING COMPUTATIONAL FLUID DYNAMICS

DISSERTATION

Submitted in Partial Fulfillment of the

Requirement for Award of the Degree

Of

MASTER OF TECHNOLOGY

In

MECHANICAL ENGINEERING

By

MUNENDRA KUMAR

11004050

Under the Guidance of

Mr. ALOK NIKHADE

DEPARTMENT OF MECHANICAL ENGINEERING

LOVELY PROFESSIONAL UNIVERSITY

PHAGWARA, PUNJAB (INDIA) -144402

2015

Lovely Professional University Jalandhar, Punjab

CERTIFICATE

I hereby certify that the work which is being presented in the Dissertation entitled **"Study of effect of cavitation in Kaplan turbine using Computational Fluid Dynamics"** in partial fulfillment of the requirement for the award of degree of **Master of Technology** and submitted in Department of Mechanical Engineering, Lovely Professional University, Punjab is an authentic record of my own work carried out during period of Dissertation under the supervision of **Mr. Alok Nikhade**, **Assistant Professor**, Department of Mechanical Engineering, Lovely Professional University, Punjab.

The matter presented in this dissertation has not been submitted by me anywhere for the award of any other degree or to any other institute. .

Date: **Munendra Kumar**

This is to certify that the above statement made by the candidate is correct to best of my knowledge.

Date: **Mr. Alok Nikhade**

Supervisor (Dissertation II)

The M-Tech Dissertation examination of Munendra Kumar, has been held on

Signature of Examiner

ACKNOWLEDGEMENT

I would like to express my sincere gratitude to Assistant Professor **Mr. Alok Nikhade,** Department of Mechanical Engineering, Lovely Professional University, Punjab for suggesting the topic for my thesis report and for his always ready guidance throughout the course of my DissertationII report. I am highly indebted to him for his creative ideas and criticism at time to time during the entire course of DissertationII.

I am also extremely thankful to **Dr. Rajiv Kumar Sharma** HOS of Mechanical Engineering Department, Lovely Professional University Jalandhar for valuable suggestions and encouragement.

Munendra Kumar

ABSTRACT

A numerical simulation of three dimensional Kaplan turbine under cavitation condition has been conducted under turbulent flow conditions. The Kaplan turbine has capacity of 7 MW power generation and specific speed of 413, with rated speed of 125 rpm. Influence of wicket gate openings on power generation is investigated. At the best efficiency point of operation condition, cavitation is simulated. Vapour fraction at various places on rotor blade surfaces is investigated. Different parameters like velocity, pressure at both sides of rotor blade and pressure distribution in draft tube have been evaluated for studying the effect on performance and flow characteristics of the turbine. Result shows that k-ω SST turbulence model is more suitable and accurate in predicting the cavitation effect, in comparison to results obtained for k-ε turbulence model.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

CFD	Computational Fluid Dynamics
σ	Thoma's cavitation factor
σ_c	Critical cavitation factor
z	Difference in elevation,
P_a	Atmospheric pressure
P_v	Vapor pressure of the water,
ρ	Density of water ,
H	Net head available,
H_a	Barometric or atmospheric pressure head,
H_v	Vapor pressure head,
H_s	Suction head,
V_e	Draft tube exit average velocity and
H_f	Draft tube friction loss.
τ	Stress tensor,
α	Semi – cone angle of draft tube
τ_{Design}	Design stress for the shaft
η	Efficiency
ζ	Torque generated
ω	Angular velocity
d_i	Casing inlet diameter
Dg	Diameter of the wicket gate
d_s	Diameter of shaft
B_g	Height of the wicket gate
Lg	Length of the wicket gate
K_{uo}	Speed ratio
K_v	Velocity coefficient
kW	Kilowatt
MW	Megawatt

N	Rpm of turbine runner
N_s	Specific speed of turbine
P_i	Pressure at inlet
P_o	Pressure at outlet
Q_0	Quantity of water flowing per second
R_θ	Radius of spiral casing at given θ
R_i	Radius at casing exit
T	Torque on the shaft
V_i	Velocity at the inlet to casing
Q	Rated discharge
P	Power
D	Runner diameter
WGO	Wicket gate opening

CHAPTER 1

INTRODUCTION

1.1 GENERAL

With the rise in population and a positive population growth the demand of power is also increasing and to generate electricity the resources are limited. The expected time for which resource will be available is reducing. First is conventional and second is non- conventional. In conventional energy resources, fossil fuels which produces large amount of greenhouse gases and pollution. At present in India almost 17%of its total electricity is being produced by hydropower plants.

Table 1 Contribution of energy sources in Indian power sector [1]

MODES	THERMAL	NUCLEAR	HYDRO	RES
INSTALLED CAPACITY(MW)	180361.89	5780	40867.43	31692.14

1.2 COMPONENTS OF HYDROPOWER PLANT

Components of hydropowerplant can be categorized as following below:

1.2.1 Civil Works Components

Civil works components of hydropower are listed below

- Weir
- Intake works
- Sand trap
- Forebay
- Power channel
- Penstock

1.2.2 Electro-Mechanical Components

Electro-Mechanical equipment's mainly include following

- Hydro-Turbine.
- Generator
- Governor
- Gates and valves and other auxiliaries.

1.2.1 Turbine

First turbine was made in 18^{th} century in the form of watermill. From then till now various types of turbines have been developed for various applications. Main types of turbines are reaction and impulse turbines.Hydro-Turbine is a machine which converts the water potential energy into mechanical energy of a rotating shaft which drives an electric generator to produce electricity.
Power produce by turbine can be written as following:

$$\boldsymbol{P = \rho g Q h} \tag{1.1}$$

P = power

g= gravitational acceleration

h = head

Q = discharge

Table 2 Classification of Turbines according to their capacities[2]

TYPES	LARGE	MEDIUM	SMALL	MINI	MICRO	PICO
RANGE	>100(MW)	25-100(MW)	1-25(MW)	100(KW)-1MW	5-100(KW)	<5KW

1.3 TYPES OF TURBINE

- Impulse Turbine
- Reaction Turbine

1.3.1 Impulse Turbine

Impulse turbine uses kinetic energy of the water to produce mechanical energy.in this turbine water strikes on the bucket and its kinetic energy gets converted into rotational energy of the runner.

1.3.2 Reaction Turbine

Reaction turbines are turbines which use both pressure and kinetic energy of the water. A schematic diagram is shown in fig 1.

Types of reaction turbines:

- Francis Turbine
- Kaplan Turbine
- Propeller Turbine

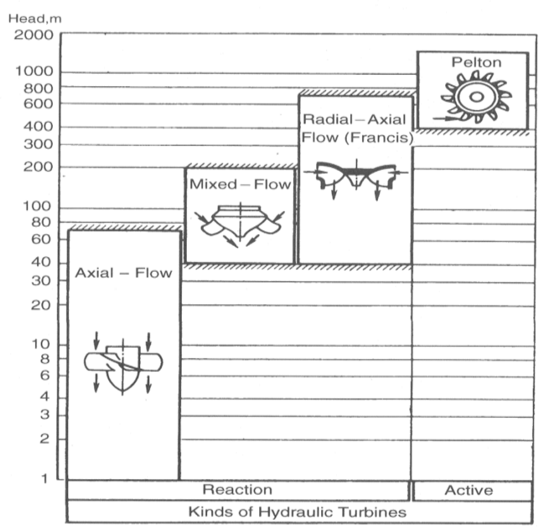


Figure 1 Types of hydraulic turbines [3]

1.3.2.1 Kaplan

Kaplan turbine uses both pressure and kinetic energy of the water to produce mechanical energy. Kaplan is used in those applications where head is low and discharge is high.

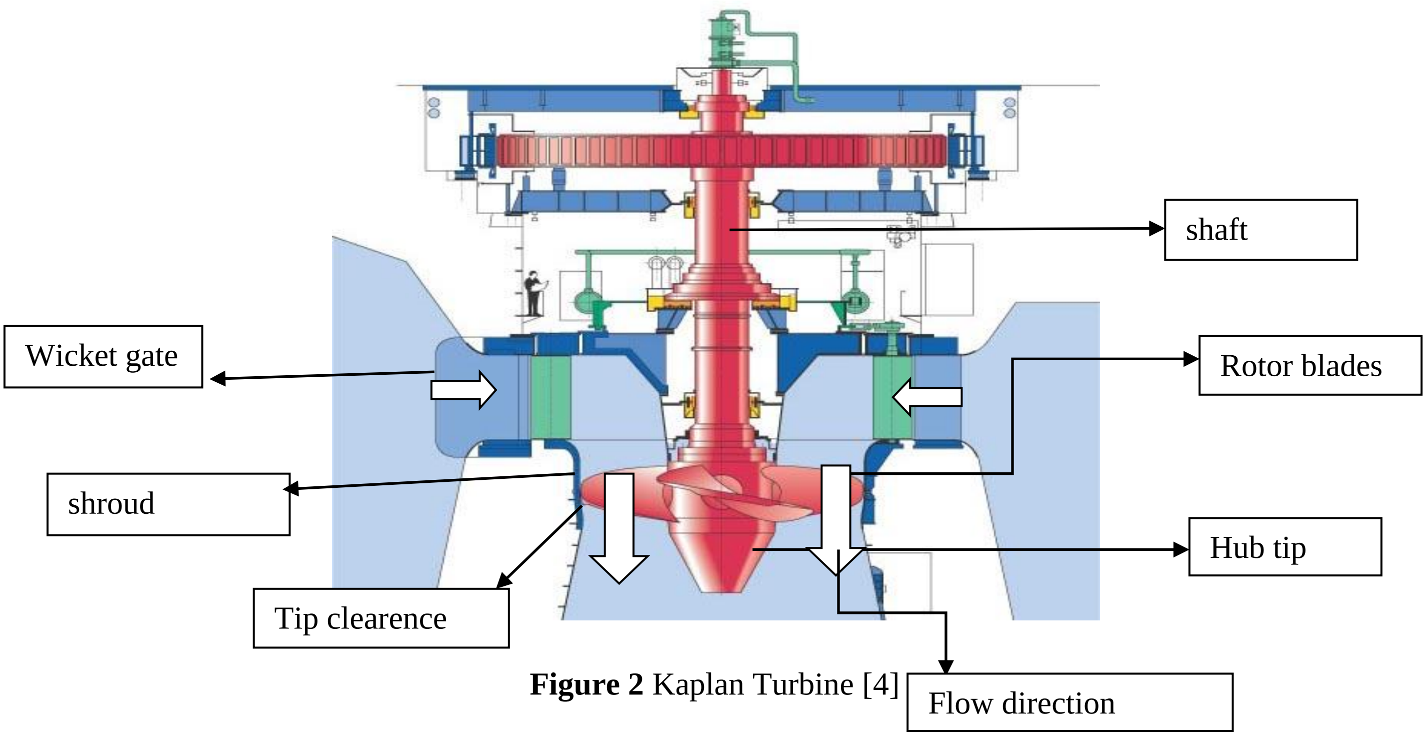


Figure 2 Kaplan Turbine [4]

(i) Moving parts in a Kaplan Turbine

- Wicket Gate
- Rotor Blade

Unique ability is found in Kaplan Turbine as compare to propeller turbine and it facilitates to match the frequency in part load condition.

(ii) Main issues in Kaplan Turbine

- Silt erosion
- cavitation

(iii) Selected turbine specification

In present study 7MW Kaplan turbine with 15m net head and 125rpm is used.

1.4 STRUCTURE OF THE REPORT

Chapter 2: In this chapter review of literature is done on both experimental and numerical studies for design of turbine and cavitation.

Chapter 3: In this chapter, brief introduction is given, about cavitation in hydro turbines.

Chapter 4: This chapter describes CFD modelling and various turbulence models used in present work.

Chapter 5: This chapter contains design of Kaplan turbine of 7MW used in present study.

Chapter 6: It contains results observed from present thesis and analysis.

CHAPTER 2

TERMINOLOGY

Turbine

First turbine was made in 18^{th} century in the form of watermill. From then till now various types of turbines have been developed for various applications.Main types of turbines are reaction and impulse turbines. Hydro-Turbine is a machine which converts the water potential energy into mechanical energy of a rotating shaft which drives an electric generator to produce electricity.

Kaplan Turbine

Kaplan turbine uses both pressure and kinetic energy of the water to produce mechanical energy. Kaplan is used for low head applications where discharge is high.

Cavitation

Cavitation occurs in turbines only only if local pressure of the turbine goes below the vapor pressure at operating temperature. If fluid flow is steady then pressure will decrease if velocity is increasing.

Computational fluid dynamics

Science of predicting fluid flow, heat and mass transfer, and other related phenomenon by solving mathematical equations which govern these processes using numerical methods is known as Computational Fluid Dynamics

CHAPTER 3

SCOPE OF STUDY

Various numerical studies have been done on Hydro turbines. Turbines show different flow characteristics on different turbulence models and boundary conditions. Their Flow behavior also varies with different geometric parameters. So it becomes necessary to investigate the suitable combination of turbulence models and boundary conditions for turbine according to their requirements. It is suitable to investigate this using Computational Fluid Dynamics. CFD is a cost effective approach and gives reliable results in very less time. CFD is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions and related phenomenon by solving mathematical equations which govern these processes using numerical methods.

Present research deals with cavitation in Kaplan turbine with combination of different boundary conditions and 2-eq turbulence models. Modeling of turbine is done in ANSYS Design Modeler and MESH is generated in ANSYS 14.0 mesh module. Further analysis is done in FLUENT solver. Model is validated with previous studies and theoretically both.

CHAPTER 4

LITERATURE REVIEW

Turbines are regarded as the most significant component of any hydro power plant. They cover about 15 – 35 percent of the total project cost. Feasibility of the project depends on the cost of its components. Many laboratory experiments were carried using physical model to know the hydrodynamic behavior of the machine. Physical modelling has been one of the methods in knowing the behavior of the turbine but has been a costly and time consuming technique Since past two decades computational fluid dynamics has become a powerful tool in analyzing the flow field of complex turbo machines and has been used extensively during the study for design of the turbine in order to optimize the design as well as to save time. In this chapter literature review for design optimization of Kaplan turbine and Cavitation in turbine is described.

Categories of topics

- Experimental investigation
- Numerical studies

4.2 EXPERIMENTAL INVESTIGATION

Punit Singh and Franz Nestmann [5] detailed experiment investigated on the effects of exit blade geometry on the part-load performance of low-head axial flow propeller turbines. The relationship between exit tip angle, discharge, shaft power, and efficiency has presented.

Punit Singh and Franz Nestmann [6] investigated the influence of design parameters in low head axial flow turbines like height of blade, blade profiles and blade number.by experimental result developed physical relationship between the two design parameters (blade height and blade number) and the performance parameters.

4.3 NUMERICAL STUDIES

Harsh vats and R.P. Saini [7] investigated the Combined Effect of Cavitation & Silt Erosion on Francis turbine and concluded that combined effect of sand erosion and cavitation is more pronounced than their individual effects

Jain *et al*., 2010 *et al* [8] studied Two types of methodology are generally used for the simulation of the turbine depending upon the required objectives. Steady state simulation has been used to determine the efficiency of the turbine . The nature of flow in turbine is unsteady in nature due to rotating of the runner. Therefore unsteady state simulations are conducted to know the transient behavior of the flow and for the investigation of the pressure fluctuation in different component of the turbine and also the interaction between rotor and stator.

.Taun singh tanwar *et al* [9] studied. CFD based performance analysis and stress analysis done on runner blade and guide vane respectively of a Francis turbine. Flow behavior and fatigue analysis is done respectively and found the life, damage, factor of safety for that particular model.

Nilsson and Davidson *et al* [10] studied dissimilar turbulent modelling were found being used in simulating flow in hydro turbine. In this low Reynolds's number turbulent models are being used to resolve the viscous sub layer it also resolve the transition phenomena occurring in turbine and very useful if there is hub and tip clearance is present in that turbine.

Jain *et al*[11] studied among the turbulence model in a Francis turbine, k-w shear stress transport model has been found to give better convergence and turbine performance compared with standard k-e, Renormalization group (RNG) k-ϵ. SST model has been successful in giving accurate results for cases where there are adverse pressure gradient flows and where separation of flow occurs.

Hsing-nan Wu *et al* [12] investigated the design parameters that affect the performance of horizontal-axis water turbines by using computational fluid dynamics and Simulation result validated with experimental result. The result found that the power is directly proportional to square of radius of runner blade and cubic of velocity.

Dr. Vishnu Prasad [13] CFD analysis done on axial flow turbine at different operating condition and validated with existing experimental setup result. The variation of computed parameters justifies with the characteristics of axial flow turbine. The computed efficiencies at some regimes of operation are critically compared with experimentally tested model results and found to bear close comparison.

Kiran Patel *et al* [14] investigated CFD analysis of Francis turbine and validated of CFD result with experimental result. CFD analysis of complete system is conducted at best efficiency point as well as at part load and performance along with losses of various components predicted. Predicted CFD analysis results are compared with experimental results and they show good agreement. This case becomes guidelines for our future new development project.

Alok Mishra and R.P Saini [15] studied, CFD-based performance analysis of Kaplan turbine for micro hydro range. The operating parameters of the Kaplan turbine have been optimized using commercial CFD package ANSYS-14. The turbine having rated capacity of 100 kW at rated head and discharge of 1.5 m and 7.03m3/s respectively. Based on CFD analysis it has been found the total pressure variation is high in the runner and near the wall of the casing. In the casing maximum value of total pressure has been observed while minimum value of The efficiency of the Kaplan tubular turbine has been found to be maximum as 91% for design condition i.e. at rated discharge 7.03m3/s and at rated head 1.5 m at 85% wicket gate opening.

Jacek Swiderski *et al* [16] investigated practical applications of Computational Fluid Dynamics (CFD) based on CFX-TASC flow software. Tried to solve reverse problem should there be a strict mathematical to the N-S equations, the solution to a problem of finding a shape of flow channel to achieve certain effect would be possible.

Dinesh Kumar *et al* [17] carried out flow analysis of Kaplan turbine using commercial CFD package ANSYS 14 at different wicket gate openings. This analysis can be useful to determine the basic flow physics in various domains. A turbine having rated capacity of 7 MW at rated head and discharge of 15 m and 47.54 m3/s respectively has been considered for this analysis. Based on CFD results obtained, it is observed that best efficiency point of turbine is at 80% WGO. The analysis ascertained the trend of losses and flow pattern in various domains. The numerical simulation results are found in order to be consistent with the real situation. It shows that ANSYS-14 is able to generate good computational results in an efficient way.

PENG YU-cheng *et al* [18] studied three Gorges hydropower units found Large-area erosions such as rust and obvious cavitation on the surface of the guide vane. The flow passages from the inlet of the spiral case to the outlet of the draft tube are included in the computational domain. The results found that the static pressure on the guide vane surface is much higher than the critical pressure of cavitation. Cavitation phenomena studied at different depth by using large eddy simulation technique

Bernd Nennemann and Thi c.vu [19] predicted the Kaplan turbine blade and discharge ring cavitation, cavitation found at a number of different locations, notably at the blade leading edge, on the blade suction side, in both tip and hub gaps and on the discharge ring. Cavitation can result in frosting or pitting in the above-mentioned locations.

BraneSirok *et al* [20] reviewed new method of the cavitation monitoring was tested on the model Kaplan turbine, where beside the computer-aided visualization various integral parameters were simultaneously observed. The procedure tested and performed in different integral operational regimes. Results of the study indicated that visualization method is the most appropriate for the cavitation monitoring.

C Deschenes *et al* [21] studied. CFD based analysis is done on Kaplan turbine numerical model and prototype and validate their results with experimental data. Flow was reproduced at best efficiency point after so many results came from flow analysis at different combination of boundary condition and turbulence models.

Huixuan Shi *et al* [22] investigated on cavitation in Kaplan turbines are to achieved Characteristics of sound waves emitted by cavitation, The laws of cavitation intensity varying with operating states donated by water head and power output, The relationship of cavitation intensity to cavitation erosion level.

Pardeep Kumar and R.P. Saini [23] reviewed cavitation on hydro turbine. Based on literature survey various aspects related to cavitation in hydro turbines, different causes for the dropped performance and efficiency of the hydro turbines and appropriate remedial measures suggested by various investigators have been discussed.

A Rivetti*et al* [24] studied. CFD based analysis is done on Kaplan turbine numerical model and prototype measurements were used to validate the transient simulations. In this study RNG k-ε model is used in ANSYS CFX and in pressure pulsation signals at low frequency, good argument found both in shape and amplitude.

Santiago *et al* [25] employed steady and unsteady simulation for the numerical analysis of the Francis turbine. Boundary conditions were mass flow inlet and opening with pressure outlet with solid surfaces as walls for both types of simulation. He stated that this type of boundary condition represents real flow in the turbine. He developed the hill chart of the Francis turbine to determine its efficiency by 25 simulations with five different openings of guide vanes

4.4 GAPS IDENTIFIED

As reviewed from the literature it was found that

a. For the studies that have been referred in the preceding literature survey, flow analysis has been done only through steady state simulation of flow with different boundary conditions, though an unsteady or transient simulation can reveal the flow characteristics upon varying wicket gate openings. With the limited computational capability, the same can be achieved by steady state simulation of different wicket gate opening for the same rotational speed of the runner.

b. Study of cavitation effect on Francis turbine of small hydro power has been investigated by authors as noted in the literature survey, though such investigations remain at hand to be carried out for Kaplan turbine in the small hydro range.

c. Study of cavitation on large scale hydropower turbines have revealed that k-ω SST viscous turbulence model has proven, with great accuracy, helpful in predicting the generation of vapor and the appropriate vapor fraction values could be estimated in comparison to the other two equation turbulence models.Investigation of cavitation effect has been successful for Kaplan turbines but with k-ϵ model, however more accurate results can be obtained with k-ω SST model, thus such investigation must be carried out.

CHAPTER 5

OBJECTIVE OF THE STUDY

The number of hydro plant face sever cavitation problem in turbine which over a period of time drastically reduce the overall efficiency of power generation system. Investigation of Kaplan turbine efficiency under effect of cavitation at different boundary condition (i) pressure inlet and pressure outlet. (ii) mass flow inlet and pressure outlet has not been carried out extensively and not available in literature. Further, CFD based analysis of hydro turbine performance is considered as cost effective method and time saving approach.

Keeping this in view, the present study is proposed with the following objectives.

i. To study and work out the design of a Kaplan turbine.

ii. To identify the system and operating parameters.

iii. To carry out the CFD analysis for the performance of Kaplan turbine under different flow condition.

a. Performance of Kaplan turbine at different wicket gateopening

b. Performance of turbine under cavitation at different boundary conditions.

c. Performance of turbine under cavitation at different turbulence models.

CHAPTER 6

CAVITATION

6.1 GENERAL

Cavitation occurs in turbines only when the local pressure of the turbine falls below the vapor pressure at operating temperature. If fluid flow is steady then pressure will decrease if velocity is increasing.

6.2 THOMA'S COEFFICIENT

It is usually necessary to specify the operating condition of the hydraulic turbines to minimize or avoid the cavitation. It is achieved by specifying the geometric parameters of the turbine to meet the acceptable range bound by a coefficient accepted and named 'thoma coefficient'.

$$\sigma = \frac{H_{SV}}{H} \tag{3.1}$$

$$\sigma = \frac{\frac{P_a - P_v}{g\rho} - Z}{H} \tag{3.2}$$

$$H_{sv} = H_a - H_v - H_s + \frac{V_e^2}{2g} + H_f \tag{3.3}$$

Where,

H_{sv} is the net positive suction head (NPSH) which is total absolute head less vapor pressure head,

Z is the difference in elevation,

P_a is the atmospheric pressure,

P_v is the vapor pressure of the water,

ρ is the density of water ,

H is the net head available,

H_a is barometric or atmospheric pressure head,

H_v is the vapor pressure head,

H_s is the suction head,

V_e is the draft tube exit average velocity and

H_f is the draft tube friction loss.

If the draft tube friction loss and exit velocity head are ignored, then

$$\sigma = \frac{H_a - H_v - H_s}{H} \tag{3.4}$$

Thoma cavitation factor (σ) for a particular type of turbine is calculated from the Eqn. (3.4) and is compared with critical cavitation factor (σ_c) for different types of turbine. Critical cavitation factor is defined as the minimum value ofσ at which cavitation occurs [26]. If the value of σ is greater than σ_c cavitation will not occur in that particular turbine.

The following empirical relationships in terms of specific speed are used for obtaining the value of σ_c for different turbines [26]:

For Francis turbine, σ_c=0.625(Ns /380.78)^2 (3.5)

For Propeller turbine, σ_c=0.28+ {(1/7.5)*(Ns /380.78)}^{3} (3.6)

For Kaplan turbine, value of σ_c can be obtained by increasing the σ_c of Propeller turbine by 10%.

Where,

N_s is the specific speed of turbine.

Limit of Thoma cavitation factor of hydraulic turbines for safe operation can be detected from the Fig3. The inclined lines in this figure shows for each specific speed, the minimum Thoma cavitation factor at which a turbine can be expected to perform satisfactorily; that is, the cavitation will be absent or so limited as not to cause efficiency loss, output loss, undesirable vibration, excessive pitting etc.

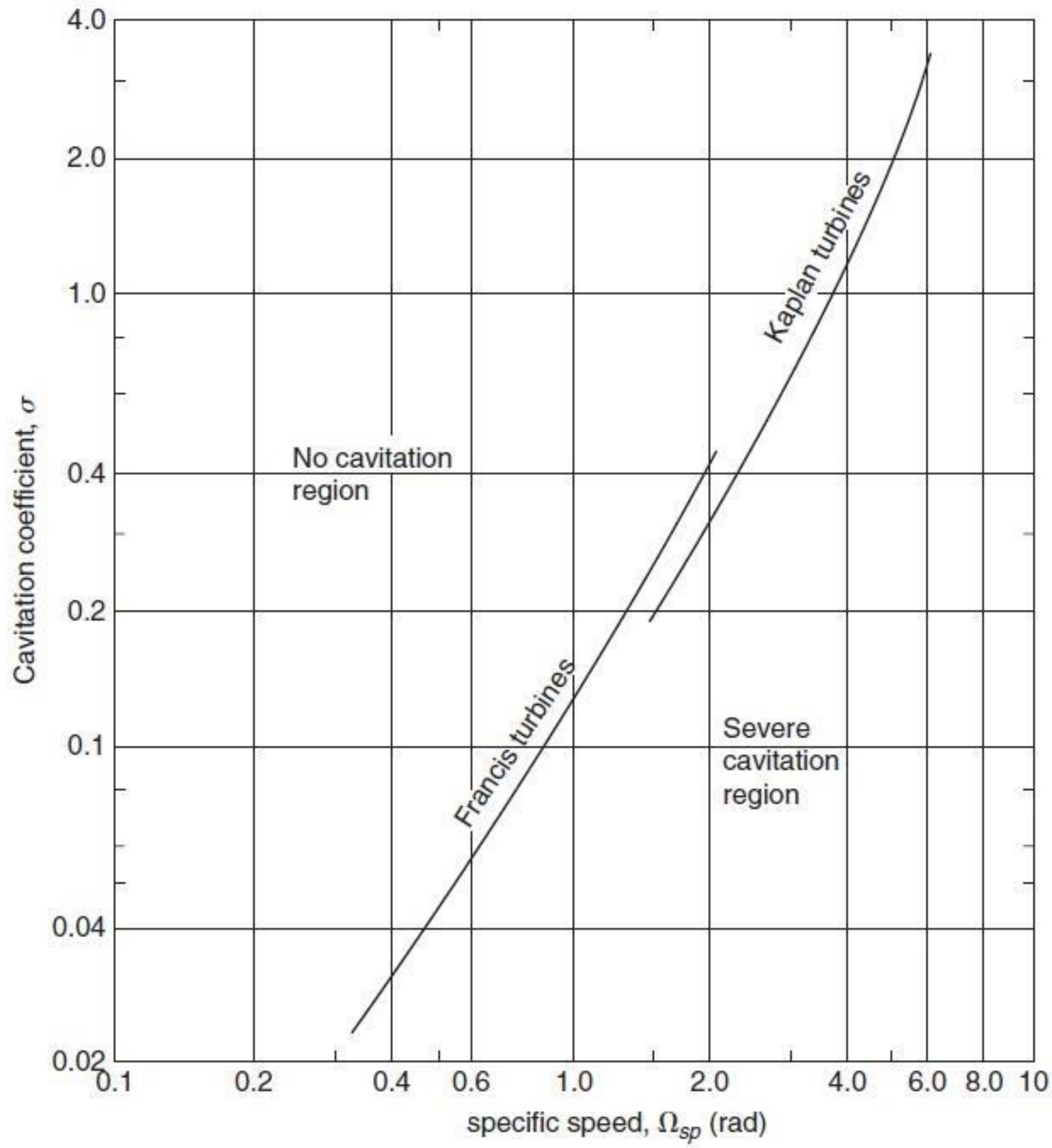


Figure 3 Experience limits of Thoma cavitation factor versus specific speed [27]

6.3 ELEMENTS INFLUENCING CAVITATION IN HYDROTURBINES

a. Specific speed of the turbine (N_s)

b. Absolute exit velocity

c. Pressure of water (vapor)

d. Total Pressure (absolute)

e. Suction head

f. Fluid’s flow pattern

6.4 EFFECT OF CAVITATION IN HYDROTURBINES

a. Changes in flow pattern

b. Increase in energy damage

c. Huge sound and vibration

d. Corrosion

e. Increase in inefficiency

Different locations in Kaplan turbine at which cavitation occurs

Tip Clearance Cavitation Chamber

Tip Vortex Cavitation

Leading Edge Cavitation

Hub Cavitation

6.5 METHODS TO AVOID CAVITATIONS

Cavitation cannot be removed completely but for reduce its occurrence the design of the turbine should be like this which would avoid the development of low pressure. Parameters like several point of flow, static pressure should not fall below the vapor pressure.

- Material should be cavitation resistant. (Stainless steel, carbon steel).
- Temperature of the fluid should be low.
- Pressure (static) should be high.

CHAPTER 7

CFD MODEL

7.1 INTRODUCTION

Computational fluid dynamics is widely being used to analyze the flow field since last 20 years. It's capability to solve complex flow phenomena with enhanced software have made it popular in several fields. X-Y-Z momentum of Navier Stokes equation and continuity equation is solved numerically. These are differential equations of complex fluid dynamics which requires discretization of the flow in order to solve them. There are several methods of discretization that have been used. Finite volume, finite element and finite difference are the three methods that are currently being used as discretization method for CFD analysis. Among these, finite volume method is used in CFX in which volume is developed by generating mesh to solve the partial differential equations of mass, momentum and energy [28].

7.2 HYDRODYNAMICS

The fundamental conservation laws governing the fluid flow in the turbine are: Conservation of mass: For any period of time mass of a system will remain conserved. The net mass flow out of the system must be equal to rate of decrease of mass inside the system. Following equation is the partial differential equation form of the continuity equation.

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho V) = 0 \tag{4.1}$$

Conservation of momentum: Net force applied on the moving fluid is equal to the mass of an element times its acceleration. Forces acting on the fluid element are body forces which includes gravitational, electric and magnetic forces and surface forces which are due to the pressure distribution acting on the surface and the shear stress and normal stress distribution due to outside fluid. Following is the equation for the conservation of momentum.

$$\rho\left[\frac{\partial V}{\partial t} + V.\nabla V\right] = F_b - \nabla_p + \mu\nabla^2 V + \frac{\mu}{3}\nabla(\nabla.V) \tag{4.2}$$

These equations are solved for the finite volume element as developed with the meshing tool and analytical solutions are obtained as required for the study. With proper boundary condition and quality mesh, CFD is able to solve both laminar and turbulent flows. Laminar flows are easier to solve compared with the turbulent flows as turbulent flows are unsteady in nature which are developed at high Reynolds's number. Small scale and large scale eddies are developed which are three dimensional and random in space and time. There are flow models developed to solve turbulent flows. Depending upon the requirement of the results, flow models are selected and computational power required for the simulations depends on these selected models. Turbulence modelling is solved by Reynolds's Averaged Navier Stokes (RANS) equation, large eddy simulation (LES) model and direct numerical modelling (DNS).

Direct numerical simulation gives very accurate result by solving all time and space scales. However, the grids needs to be very small to resolve spatial and temporal scales and consumes large computational time. Reynolds's Averaged Navier Stokes model solves by time and space averaging. This model provides pretty good result but fails to give accurate results for transient simulation where variables vary with time.

7.3 TURBULENCE MODELLING

Large scale and small scale eddies are generated due to turbulence in the flow. Various turbulence models have been developed to capture the effect of these unsteady turbulent eddies. These models are based on Reynolds's Averaged Navier Stokes equation which is the sum of statistically averaged component and fluctuation component.

7.4 SHEAR STRESS TRANSPORT MODEL

Shear stress transport model is the combination of k-e and k- ω model. These two models are two equation turbulence models whose transport equations are solved along with mass and momentum equations. The k denotes turbulent kinetic energy, e denotes rate of turbulent dissipation and ω denotes specific dissipation. The limitations of these two models are incorporated in SST model. k-e model over predicts the shear stress in adverse pressure gradient flows and it requires near wall modification as well but still it cannot capture proper flow separation in the turbulent flow. For proper flow prediction in the near wall layers k- ω model provides better accuracy then k-e model. However it also fails to predict for separation induced by high pressure. Shear stress transport model was developed to overcome the limitations of these two models. With the introduction of blending factors, k- ω and k-e zones are selected automatically without user interaction at near wall region and away from the surface respectively. This model gives accurate results for the flows with adverse pressure gradient like in airfoils and flows with separation. Hence, Shear stress transport model is reliable to be used for turbines [29]

7.5 APPLICATIONS OF CFD

Computational fluid dynamics is a very effective tool and is being used for research in industries. Applications of CFD are in heat transfer analysis and fluid flow analysis in hydrology and aerospace.

7.6 NUMERICAL SIMULATION PROCEDURE

To numerical simulation of a physical problem involves approximation of the problem geometry, choice of appropriate mathematical model and numerical solution techniques, computer implementation of the numerical algorithm and analysis of the data generated by the simulation. Thus, this process involves the following steps [30]

I. geometry modeling of the domain associated with the problem.

II. Selection of he best fit mathematical model for physical problem.

III. Selection on appropriate method of discreitization.

IV. grid generation based on the geometry and the discretization method.

V. Selection of technique to solve these equations..

VI. Set appropriate convergence criteria for iterative solution methods.

VII. Prepare the numerical solution for further analysis

7.6.1 Geometry modelling

The numerical simulation needs a computer representation of the problem domain. For most of the engineering problems, it may not be possible or even desirable to include all the geometric details of the system in its geometric model. The analyst has to make a careful choice regarding the level of intricate details to be chosen.

7.6.2 Mathematical modelling

Keeping in view the physics of the flow problem and simulation's objective an appropriate mathematical model has to be selected. Available computing resources and level of accuracy desired also have a important role in selecting a mathematical modeling.

7.6.3 Classification of navier-stokes equations

Navier-Stokes equations are coupled nonlinear partial differential equations in four variables. For mathematical classification, we can look at their linearized form. The formal classification for incompressible flows is as follows:

I. Steady viscous flows: elliptic.

II. Unsteady viscous flows: parabolic.

Energy equation has the same behavior, i.e. it is elliptic for steady flows, and parabolic, for Time dependent problems. Unsteady Navier-Stokes and energy equations are in fact parabolic in time and elliptic in space. Hence, solution of these equations requires (a) one set of initial conditions, and (b) boundary conditions at all the boundary points for all values to time $t > 0$. Compressible Navier-Stokes equations may be considered as mixed hyperbolic, parabolic and elliptic (or incompletely parabolic) equations. The elliptic equations are more difficult to solve than parabolic equations, which lend themselves to marching type solution procedure. Thus, in practice, steady viscous flows are usually converted to unsteady problems, and solved using a time marching scheme. The solution obtained for large values of time *t* provides the desired solution of the steady viscous flow problem.

Navier-stoke's equation

$$\frac{\partial y}{\partial x} = \nabla.(\rho VV) = \rho b - \nabla p + 2\nabla[\mu(s - \frac{1}{3}(\nabla.V)I] \qquad (4.3)$$

7.6.4 Classification of Euler equation

Classification of inviscid flow equations governed by Euler equation is different from that of Navier-Stokes or energy equations due to complete absence of the second order terms. The classification of these equations depends on the extent of compressibility. Further, for compressible flows, flow speed (Mach number, Ma) has a significant role in determining the behavior of the problem. The formal classification of the inviscid flows is as follow:

I. Inviscid incompressible flows

a. Steady flows: elliptic.

b. Unsteady flows: parabolic.

II. Inviscid compressible flows

a. Steady subsonic flows ($Ma < 1$): elliptic.

b. Steady supersonic flows ($Ma > 1$): hyperbolic.

c. Unsteady flows: hyperbolic.

Dependence of the flow behavior on local Mach number makes the solution of steady inviscid compressible flows pretty complicated. For example, let us consider high speed flow around a bluff body. Even if the upstream Mach number $Ma > 1$, in the vicinity of the solid surface, there exists a subsonic zone as the flow velocity goes to zero at the stagnation point. Therefore, separate algorithms would be required for numerical simulation of the problem in subsonic and supersonic zones (whose extent and boundaries are themselves unknown). This situation has baffled aerodynamicist for a while in early 1960s. A way out was provided by the time dependent approach to solve a steady problem, since the unsteady Euler equation for compressible flows is hyperbolic everywhere (irrespective of the local Mach number). In fact this time dependent approach to steady state is also widely used now for solution of steady state viscous flows.

Euler's equation

$$\frac{\partial(\rho V)}{\partial t} + \nabla.(\rho VV) = \rho b - \nabla p \tag{4.4}$$

7.6.5 Discretization method

There are many dicretization approaches to convert continuum mathematical model into a discrete system of algebraic equation. The most popular are the following [31].

I. Finite difference method (FDM)

II. Finite volume method (FVM)

III. Finite element method (FEM)

7.6.6 Grid generation

The problem domain is discretized into a mesh/grid appropriate to the chosen discretization method. The type of the grid also depends on the geometry of the problem domain. Structured grid is required for the finite difference method, whereas FEM and FVM can work with either structured or unstructured grids. In case of unstructured grids, care must be taken to ensure proper grading and quality of the mesh.

Types of grids

There are three types of grids.

I. Structured grids

II. Block-structure grids.

III. Un-structured grids

Grid generation process

The process of grid generation for complicated geometries normally involves following steps:

I. Decompose the problem domain into a set of sub-domains (blocks)

II. In each block, generate the grid. Typical sequence of operations would be

a. Generate edge-grid (i.e. divide the edges of a surface in desired number of one dimensional elements).

b. Using the edge grids, generate the grid on the block-surfaces.

c. Use surface grids as input to generate volume mesh.

III. Check mesh quality, and modify the mesh as required.

7.6.7 Numerical solution

The discretization method applied to the mathematical model of the problem leads to a system of discrete equations: (a) a system of ordinary differential equations in time for unsteady problems, and (b) a system of algebraic equations for steady state model. For unsteady problems, time integration methods for initial value problems are employed, some of which transform the differential system to a system of algebraic equations at each time step. algebraic equations are solved by iterative method.choice of methods being dependent on the type of the grid and size of the system..

Solution of discrete algebraic system

Application of FDM, FVM, OR FEM lead to a system of algebraic equation which may be linear or non-linear depending on the problem

I. Iterative method for linear system:

a. Direct solver (LU decomposition, Gauss elimination etc.)

b. Iterative solver (SOR, Gauss-seidel, PCG etc.)

c. Accelerated iterative method(conjugate gradient, GMRES)

II. Iterative method for nonlinear system:

a. Newton-Raphson method

b. Global method (Picard iteration)

c. Mix of global and Newton-Raphson method

III. Time dependent problem:

a. Two level method for first order initial value problem

b. Multi-point method (Adams-Bashforth method)

c. Multi-level predictor-corrector methods

d. Runge-Kuttamethods(these method are two level multi point method)

7.6.6 Post processing

Numerical simulation provides values of field variables at discrete set of computational nodes. Post processing which involves report generation and understanding flow with different plots and contours. Numerical simulation provides values of field variables at discrete set of computational nodes. For analysis of the problem, the analyst would like to know the variation of different variables in space-time. Further, for design analysis, secondary variables such a stresses and fluxes must be computed. Most of the commercial CFD codes provide their own postprocessor which compute the secondary variables and provide variety of plots (contour as well as line diagrams) based on the nodal data obtained from simulation. These computations involve use of further approximations for interpolation of nodal data required in integration and differentiation to obtain secondary variables or spatial distributions

7.6.7 Validation

In present studyKaplan turbine's 7 MW model is used. It has a rotation speed of 125 and specific speed of 413 with 15 meter of head and mass flow rate of 47.57 m^3/sec. Kaplan Turbine model is validated with previous studies [17] results and also validated theoretically.

7.6.8 Models available in ANSYS fluent

There are two types of flow problem steady and turbulent. Different models for steady flow as well as turbulent are present. Laminar flow are characterized by smoothly varying velocity fields in space and time in which individual laminar move past one another without generating traverse currents. These flows arise when the fluid viscosity is sufficiently large to damp out any perturbations to the flow that may occur due to boundary imperfections or other irregularities. These flows occur at low to reasonable values of the Reynolds's number. The turbulent model flows are characterized by large nearly random fluctuations in velocity and pressure in both space and time. These fluctuations occur from instabilities that grow until nonlinear interactions cause them to break down into finer and finer whirls that eventually are dissipated (into heat) by the action of viscosity. There are different turbulence models available in fluent.

i) Spalart-Allmaras model

ii) k-ε models

- Standard k- ε model

- Renormalization-group (RNG) k- ε model

- Realizable k- ε model

iii) k-ω models

- Standard k-ω model

- Shear-stress transport (SST) k- ω model

iv) v2-f model

v) Reynolds stress model (RSM)

vi) Detached eddy simulation (DES) model

vii) Large eddy simulation (LES) model

Mutiphase model available in ANSYS fluent

i) Mixture

ii) Eulerian

iii) Volume of fluid

As mentioned above, among these models, k-ω SST model is very popular in industry for turbulence modelling. This model gives fairly good results. Here in this work k- ω SST model is used for flow analysis of Kaplan turbine and for cavitation multiphase mixture model is used for the further analysis.

CHAPTER 8

RESEARCH METHODOLOGY

In order to meet the objective of the work, first step will be to size the Kaplan turbine for the available data of site like discharge and head, which includes obtaining the dimensions of the turbine and associated components. The dimensions obtain/calculated during sizing of the turbine to generate the 3-D model of the Kaplan turbine and its assembly components. 3-D model of the turbine was generated in Design Modeler in ANSYS Workbench 14.0.After assembling the Kaplan turbine successfully; it has been exported to 'Mesh' module of ANSYS. The generated mesh was transferred to the 'FLUENT' module where simulation has been carried out with the help of suitable turbulence model and the results were obtained.

A flow chart for the methodology to be adopted for the proposed dissertation work is shown in Fig. 4.

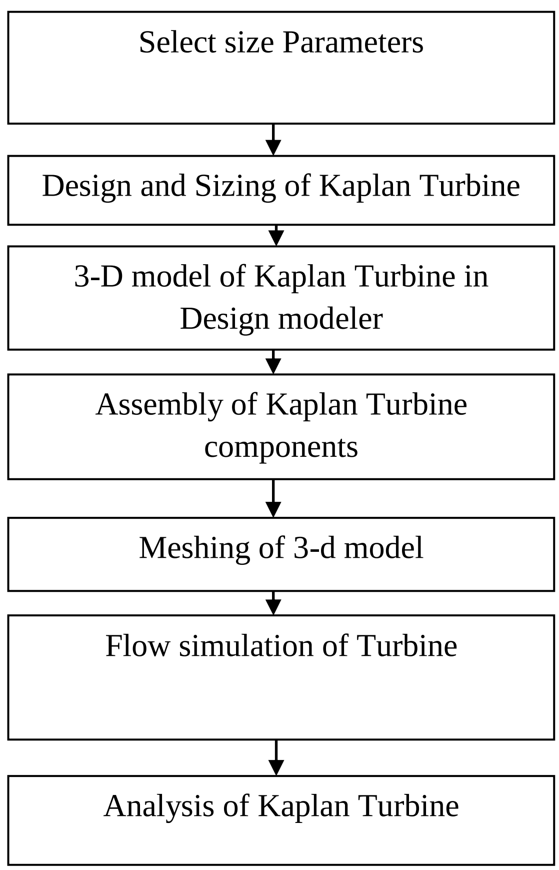


Figure 4: Flow chart showing the methodology for CFD simulation

8.1 DESIGN OF KAPLAN TURBINE

In the present study, the Kaplan turbine with spiral casing for small hydro range has been designed. Hence, Kaplan turbine having rated head of 15 m and discharge of 47.57 m3/s is considered.

Kaplan turbine

A Kaplan turbine is type of propeller turbine which was developed by the Austrian Engineer V. Kaplan. It is the axial flow turbine, which is sustain for relatively low heads, and hence required a large quantity of water to develop large amount of power. It is also a reaction type turbine and hence it operates in an entirely closed conduit from the head race to tail race. The main components of a Kaplan turbine are stay ring, scroll casing, arrangement of guide vanes and draft tube. Between the guide vane and the runner the water in a Kaplan turbine turn through a right-angle into the axial direction and then passes through the runner. The runner of a Kaplan has three, four or six (or eight in some exceptional cases) blades and it closely resembles a ship’s propeller. The blades are attached to hub or boss is so shaped that water flows axially through the runner. Ordinarily the runner of a propeller turbine is fixed, but the Kaplan turbine runner blades are movable, so that their angle of inclination may be adjust while the turbine is in motion. This adjustment of runner blade is usually carried out automatically by means of a servo motor operating inside the hollow coupling of turbine and governor shaft. When both guide vane angle and runner blade angle may thus be varied, a high efficiency range can be maintained over a wide range of operating condition. In other words even at part load, when a lower discharge is flowing through the runner, high efficiency can be attained in the case of a Kaplan turbine. It will be observed that although the corresponding change in the flow through runner does affects the shape of the shape of the velocity triangles, yet as the blade angles are simultaneously adjusted, the water under all the working conditions flows through the runner blades without shock.

Components of Kaplan Turbine

Wicket gate

Wicket gate in Kaplan is same as the guide vanes in Francis turbine. The only difference is that the guide vane in Francis usually used for guiding the water but in the case of Kaplan wicket gate used for controlling the work. The wicket gates or guide vane are fixed between two rings in the form of a wheel known as guide wheel. The wicket gate have a hydrofoil section which allow water to pass over them without forming eddies and with minimum friction losses. Each guide vane can be rotated about its pivot center which is connected to the regulating ring by means of a link and a lever and hence the required quantity of water can be supplied to the runner by opening or closing the guide vanes. The guide vanes are generally made of cast steel.

Stay Vanes

In case of big units, the stay vanes are provided at the inside circumference of the casing. The stay vanes provide strength to the casing and also direct the water towards guide vanes.

Runner

The flow in the runner of a Kaplan turbine is purely axial. There is axial entry and exit of water. The width of the runner depends on the specific speed. The high specific speed runner is wider than the one which has a low specific speed because the high specific speed runner has to work with a large amount of water. The runners are made of cast iron for small output, cast steel for large output and stainless steel or a non-ferrous metal like bronze, when the water is chemically impure and there is a danger of corrosion.

Shaft and Bearing

The runner is keyed to the shaft which may be vertical or horizontal. The shaft is generally made of steel. It is provided with a collar bearing for transmitting the axial thrust to the bearing. The turbine is generally provided with one bearing. In vertical shaft turbine, bearing carries full runner load and acts as thrust cum supporting bearing. Right selection of bearing is therefore extremely important.

Draft Tube

The water after passing through the runner flows to the tail race through draft tube. A draft tube is a pipe or passage of gradually increasing cross section area which connects the runner exit to tail race. It may be made of cast or plate steel or concrete. It must be alright and under all conditions of operation its lower and must be submerged below the level of water in the tail race.

Casing

Spiral casing is taken for this study to avoid loss of efficiency, the flow of water from the penstock to the runner should be such that it will not form eddies. In order to distribute the water around the guide ring evenly, the scroll casing is designed with cross sectional area reducing uniformly around the circumference, maximum at the entrance and nearly zero at the tip. This gives a spiral shape and hence the casing is named as spiral casing**.**

8.2 SIZING OF KAPLAN TURBINE

Potential of plant [32]

$$P=9.81*Q*H \text{ kW} \tag{5.1}$$

=7000kW

Where Q=rated discharge in (m^3/s)

H=net head in (m)

For the present study, the rated potential of the plan has been taken 7000 kW as the convenience for the design calculation of Kaplan turbine

Specific speed

$$N_s = \frac{N\sqrt{(P*1.358)}}{H_n^{5/4}} \tag{5.2}$$

N=125 r.p.m

$N_s = \frac{125\sqrt{(1000*1.358)}}{15^{5/4}}$

N_s=413

N=speed of turbine

N_s=specific speed of turbine

Runner

Outer Diameter of runner has been calculated by the equation which is given below;

$$D = \frac{84.6\times0.0233\times N_S^{2/3}/\times H_n^{1/2}}{N} \tag{5.3}$$

D = 3.4 m

Hub to tip ratio d/D=0.35

Diameter of hub d=1.2 m

Wicket gate

Diameter (D_g)

Diameter of wicket gates has been calculated by formula which is given below

$$Dg = \frac{60*K_{ug}*\sqrt{2gH}}{\pi N} \tag{5.4}$$

Dg = 3.6 m

Where, $K_{ug}$is speed ratio, varies from 1.3 to 2.25, increase with the specific speed.

For the present study, the value of the speed ratio (K_{ug}) has been taken 1.38.

Height (B_g)

In the case of Kaplan turbine the value of the ratio of Height of the wicket gates and diameter of runner vary 0.10 to 0.3 according to the specific speed. For the present study, the value of the ratio has been taken 0.23.

I.e. $B_g/D = 0.23$

So$B_g = 0.8$ m

- Length (L_g)

Empirical relation between the length and diameter of the wicket gate is given below;

$L_g=0.3*D_g = 1.1$ m

No. of wicket gates

Number of wicket gates varies 8 to 24, as increasing diameter of guide wheel.

Let the no. of wicket gate 15 which is suitable for the runner diameter of Kaplan turbine

Scroll casing

Since the diameter of the casing decrease along its path, it takes the shape of a Spiral, also known as Archimedean Spiral. So, it becomes necessary to know or figure out the dimensions and coordinate at various angles taken from the centre.

Quantity of water flowing per second, $Q_o = Vi*\frac{\pi}{4}*d_i^2$ (5.5)

Where, d_i is casing inlet diameter and, V_i is the velocity at the inlet to casing (m/s)

Also, $V_i= K_v \cdot \sqrt{2gH}$ (5.6)

Where, K_v is velocity coefficient.

The maximum value of V_i can be 10 m/s.

Starting with inlet diameter d_i of the spiral casing, it should be less than or equal to the penstock diameter. Now V_i is known d_i, Q_o is calculated.

At any angle θ with respect to Centre line of casing,

$$R_\theta = R_i = \frac{\theta}{2\pi} d_i \quad (5.7)$$

Where, R_θ is radius at given θ, R_i is radius at casing exit

- Quantity of water

$$Q = (\theta / 2\pi) \cdot Qo \quad (5.8)$$

- Diameter of Shaft

$$d_s = [\frac{16T}{\pi\tau_{Design}}]^{1/3} \quad (5.9)$$

Where, T is torque on the shaft (N-m) and $\tau_{Design}$is design stress for the shaft (N/m^2)

Table 3 Radius of scroll casing at different angle

ANGLE IN DEGREE	CRANK RADIUS (m)
0	2.9
30	3.0
60	3.1
90	3.2
120	3.3
150	3.4
180	3.5
210	3.6
240	3.7
270	3.8
300	3.9
330	4.0
360	4.1

Draft tube

Dimensions of straight divergent type draft tube were carried out using following.

i. Diameter of draft tube at inlet

$D_i = D_2$

Where, D_2 is diameter of turbine at outlet

Suction head $H_s=H_a-H_v-\sigma H_{net}$

Critical thoma coefficient $\sigma_c = 0.28 + \left(\frac{1}{7.5}\left(\frac{N_S}{380.78}\right)^3\right)$

$\sigma_c=0.45$

For Kaplan turbine $\sigma_c=1.1*0.45=0.495$

$H_s=3.2$ m

ii. Semi cone angle

α = 4-8 degree

Semi angle will be taken 5 degree

Table 4 lists the all parameters of the Kaplan turbine which have been computed by using design in this chapter

Table 4 Parameters of the Kaplan turbine

Rated Head	15m
Rated Dischage	15m^3/sec
Rated Turbine output	7MW
Rated Speed	125rpm
No. of Runner Blade	3.4m
Runner diameter	1.2m
Height of Draft tube	3.2m

8.3 FLOW ANALYSIS OF KAPLAN TURBINE

Flow analysis of the hydro turbine is a way to predict the flow behavior inside the turbine under known conditions. It is done to know the inside flow conditions i.e. pressure and velocity variations to acquire the flow physics. So the flow analysis of the considered turbine is necessary to validate the designed model with pure water, so that this model can be used for further research in design, if solution is not converged and other possibility like in interest of improving performance of turbine, substantial improvement in variables i.e. design optimization is done. A 7 MW Kaplan turbine with spiral casing under rated head of 15 m and 47.57 m3/sec discharge is considered in this study .The all parameters of the Kaplan turbine have been computed which are given in table 4. In order to carry out the flow analysis, 3-D modelling is done of each component of the turbine by using all parameters which are given in table 4. 3-D model of all components are developed separately. Flow analysis of Kaplan turbine with pure water and with mixture is discussed below in detail.

8.4 3-D MODELLING OF TURBINE COMPONENTS

3D modelling of turbine components in DESIGN MODELER cad software is the first step of Flow analysis. DESIGN MODELER is most user friendly software though which complicated geometry easily can be created.

Runner blade:

By using ANSYS module BLADEGEN blades of turbine has generated, runner is shown in Fig 5

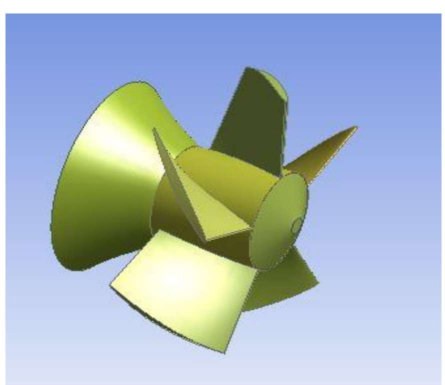


Figure 5 Runner of Kaplan turbine

Wicket gate

The 3-D geometry of wicket gate has created in DESIGN MODELER software by using PATTERN command. Wicket gate are shown in Fig 6.

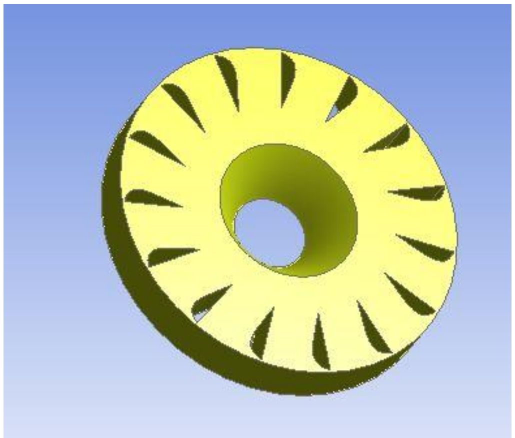


Figure 6 Wicket gate of Kaplan turbine

Scroll casing

The 3-D geometry of scroll casing has been done by sweep SWEEPBLEND.casing is shown in Fig 7.

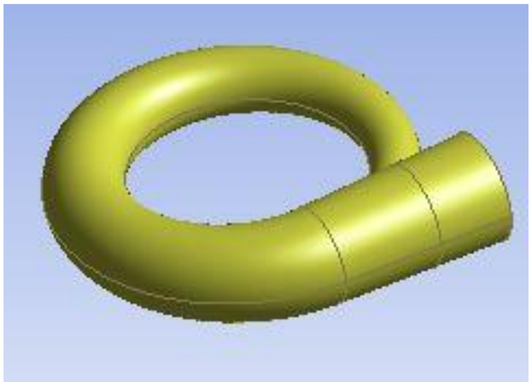


Figure 7 Scroll casing of Kaplan turbine

Draft tube

The 3-D geometry of draft tube has created in DESIGN MODELER software by using REVOLVE command. draft tube is shown in Fig 8.

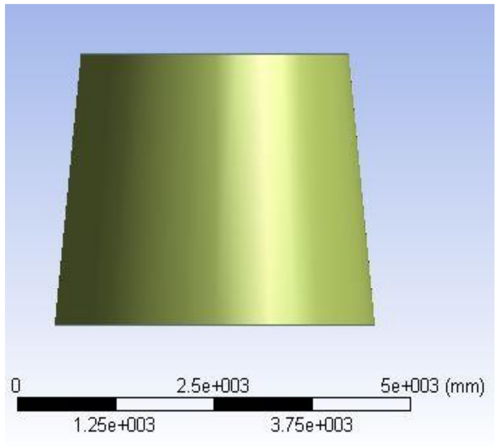


Figure 8 Simple draft tube of Kaplan turbine

Assembly of the Kaplan turbine

Assembly of the turbine has performed in DESIGN MODELER using Body operations. The assembly of the turbine is shown in Fig 9.

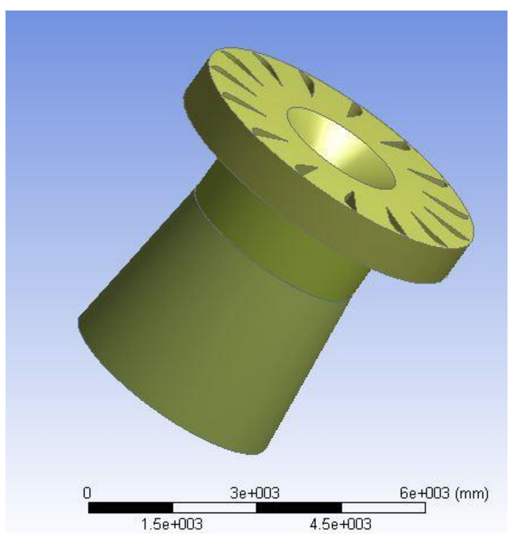


Figure 9 3-D model of Kaplan turbine

8.5 MESHING OF THE KAPLAN TURBINE:

Meshing has generated in ANSYS module MESH, Meshed models of each part are shown below in Fig 10. And the number of nodes and elements for each part are given in.

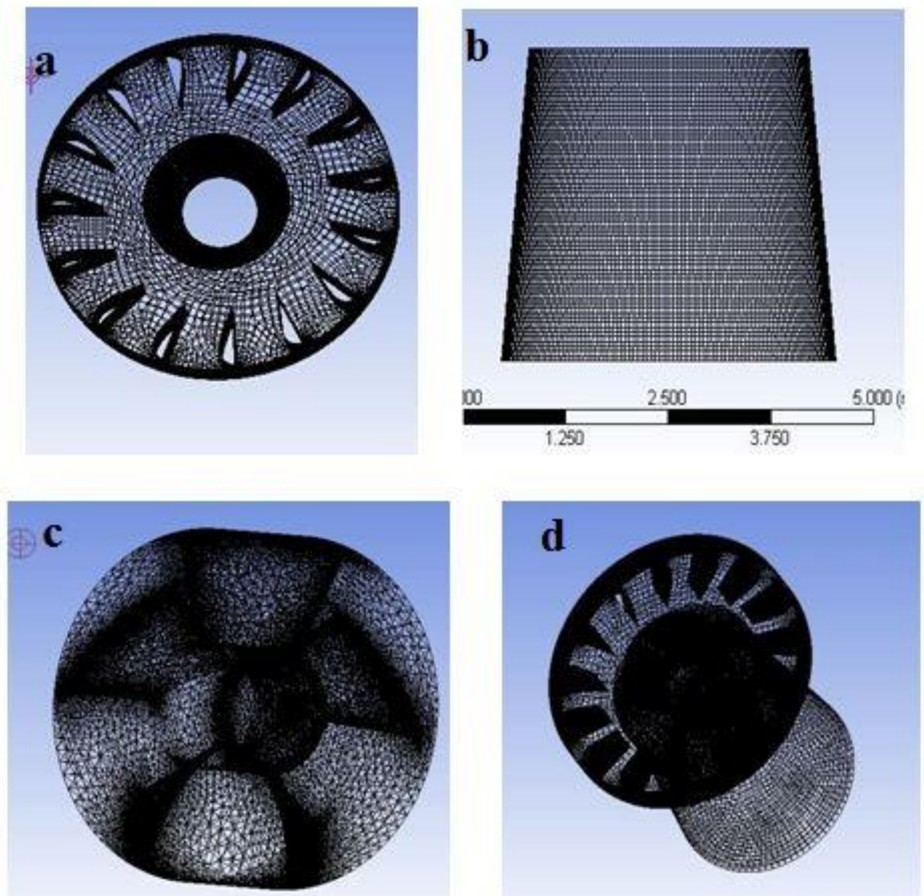


Figure 10 Meshed components of Kaplan turbine (a) Wicket Gate (b) Draft Tube (c) Runner (d) Complete assembly

Table 5 Details of MESH

NODES	ELEMENTS	ORTHOGONAL QUALITY	ASPECT RATIO	SKEWNESS
4459203	2825164	.86	2.03	0.234

8.6 SIMULATION SOLVER SETUP

CFD simulation cannot be achieved without going through this step thoroughly as here boundary conditions, operating condition of the turbine or the hydraulic machine under consideration. Following are the main steps in the solver execution**.**

(i) Importing and scaling the mesh file.

(ii) Selecting the physical models.

(iii) Defining the material properties.

(iv) Defining initial operating conditions and cell zone condition.

(v) Defining boundary conditions.

(vi) Defining Solution method and control.

(vii) Initialization solution.

(viii) Run calculation.

8.6.1 for Pure Water

Step 1: Importing Mesh: Transfer the mesh to the fluent module.

Step 2: Opening of Fluent: Open the Fluent module by double clicking the setup under the fluent dialogue box. Unable double precision other thing Accept the default settings in fluent launcher.

Step 3: General Setting: After Step 2, the main Fluent window will appear, click on General in the Problem set up column. Then, check, scale, display and report quality the grid by clicking the respective tab.

Step 4: Model Setup: Select the suitable model for the simulation, for the present study k-ω SST turbulence model is used.

Step 5: Material Setup: Defines the various properties of solid/liquid materials here. Properties like density, viscosity can be set up using material sub-window. Here material has been taken water liquid with density 998.2 kg/m3, and viscosity 0.001003 kg/m-s.

Step 6: Cell Zone & Operating Conditions Set Up: cell zone condition is defined to set up the moving frame and stationary frame. In the present study, runner is the only moving frame and rotating speed is 125 r.p.m, rest is stationary. Operating conditions defines the conditions in which machinery will be operating like the pressure and direction of Gravitational force. Here operating condition has been taken as operating pressure 101325 Pascal, gravitational acceleration 9.81 m/s2 along negative z-direction and Operating temperature 300 K.

Step 7: Boundary conditions: For the present study only two boundary conditions are specified i.e., at inlet mass flow rate is defined and at outlet i.e., pressure is defined and also provide moving condition for runner. Mass flow rate is 47570 kg/s and static pressure is 19620 pa. Intensity has taken 5 % and hydraulic diameter has taken at inlet 3.2 m and outlet 4.2 m.

Step 8: Solution Method: For the present study, SIMPLE method is used with PRESTO and Second upwind method has been used for solving the turbulence as well as moment.

Step 9: Solution Initialization: standard initialization is used with compute from inlet and absolute reference has taken.

8.6.2 For Cavitation model

Setting for cavitation model includes following changes in pure water simulation. Changes are mentioned below:

Step 1: Model: Select k-ω SST turbulence model and mixture multiphase model. Also check the slip velocity.

Step 2: Phase: Define water-liquid as primary and water-vapor as secondary phase.

Step 3: Define interaction between the phases. Select cavitation mechanism and vapor pressure taken 3540 Pa and other value take by default.

CHAPTER 9
RESULTS & DISCUSSION

9.1 GENERAL

In order to estimate or predict the amount of loss that occurs due to the effect of cavitation, using CFD performance of Kaplan turbine have been done. In the present analysis, mass flow rate is taken as 47570 kg/s at full gate opening and wicket gate opening is modeled at 60%, 80%, 100% and 110%. At a 80 % W.G.O analysis is made for single phase pure water flow.2-phase water with vapor flow is used for cavitation calculation. The static pressure 19620 pa is specified at outlet boundary condition. The reference pressure is taken as 1 atmospheric. The rotational speed of runner is specified as 125 rpm. The wicket gate and draft tube domains are taken as stationary. The SST K-ω turbulence model has been used and the wall of all domains is assumed to be smooth with no slip. Coupled arithmetic is applied for coupling between velocity and pressure.

9.2 ANALYSIS OF RESULT AT DIFFERENT WICKETGATE OPENING

The rotational speed of turbine is taken as the rated speed i.e. 125 rpm, specific speed.The wicket gate and draft tube domains are taken as stationary. The SST K-ω turbulence model has been used and the wall of all domains is assumed to be smooth with no slip.

9.2.1 Stream line flow pattern for different component

The CFD analysis is carried out by considering steady state single phase at best efficiency point as well as at part load conditions. The numerical flow analysis is accomplished at four Wicket gate openings corresponding to four different loading on turbine from best efficiency point to overload.

(i)For pure Water analysis

The 3-D streamlines and pressure contours at 60% WGO in draft tube are shown in Fig 10 and 11.

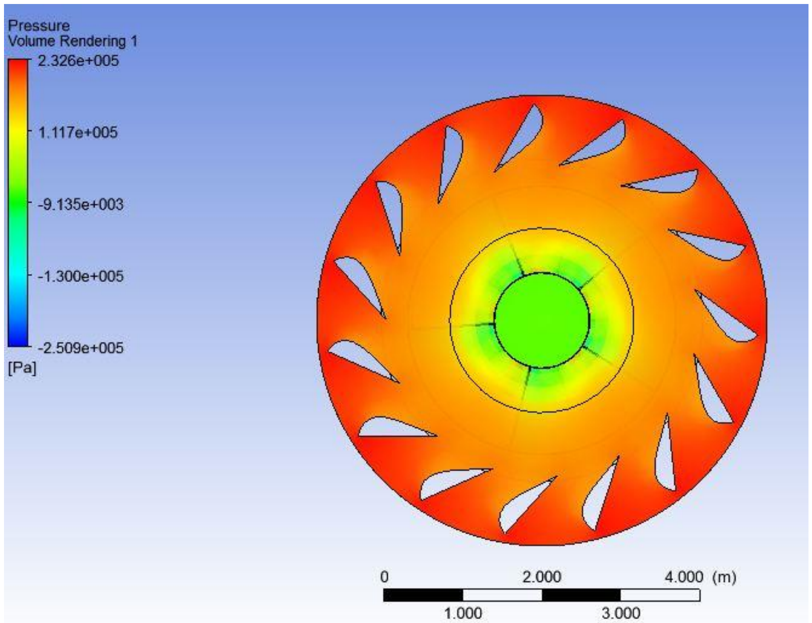


Figure 11 Pressure distribution at 60%

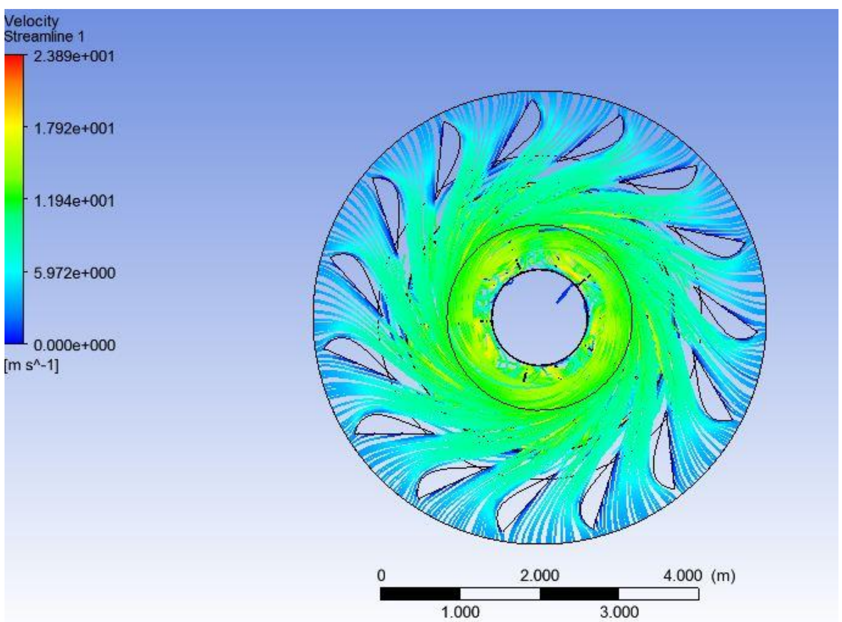


Figure 12 Velocity distribution at 60%

The 3-D streamlines at 80% WGO in Wicket Gate are shown in Fig 13.

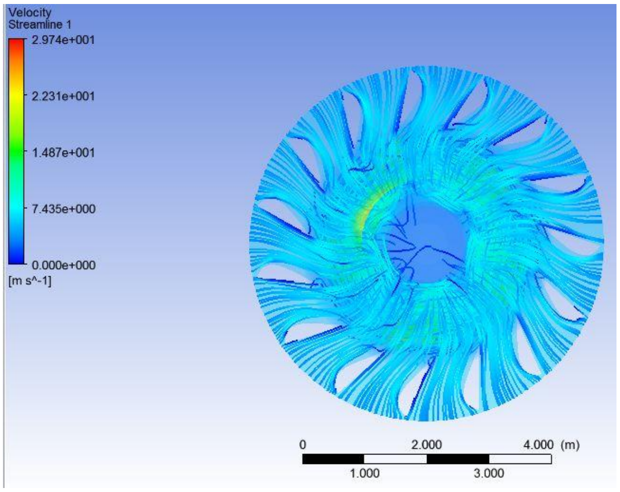


Figure 13 Velocity distribution at 80%

The 3-D streamlines and pressure contours at 110% WGO in draft tube are shown in Fig 14.It shows gradual reduction in pressure from casing inlet to exit of runner. It is seen that static pressure variation in hub region is more at larger wicket gate opening. Velocity profile from Fig14 inside the turbine assembly indicates that wicket gate and runner domain has smooth velocity profile whereas as soon as water enters draft domain velocity starts decreasing and profile becomes non-uniform.

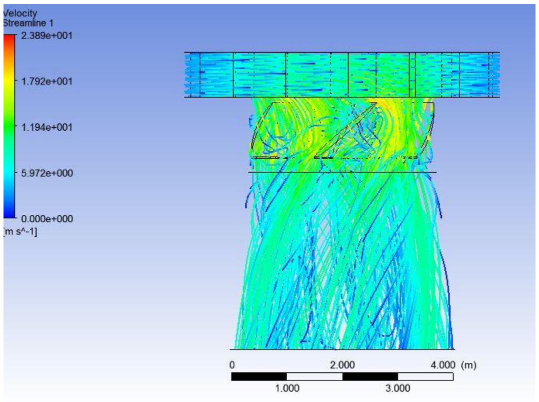
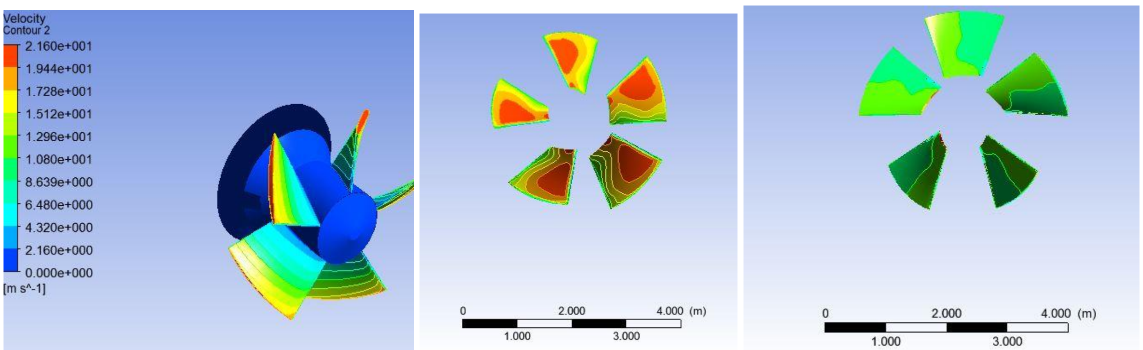


Figure 14 Velocity distribution at 80%

The velocity is more and pressure is less on suction side and velocity is less and pressure is more on pressure side of runner blades..indicate that low velocity zone is formed at diverging side of passage. The velocity decreases and pressure increases from the runner domain to exit of draft tube.

Velocity and Pressure distribution on rotor blade shown in Fig 15



(a) (b) (c)

Figure 15 Velocity and Pressure distribution on runner blade at 80% wicket gate opening

(a)Velocity distribution (b) Pressure contour at Pressure side (c) Pressure contour at suction side

pressure contour pattern within draft tube shown in Fig 15

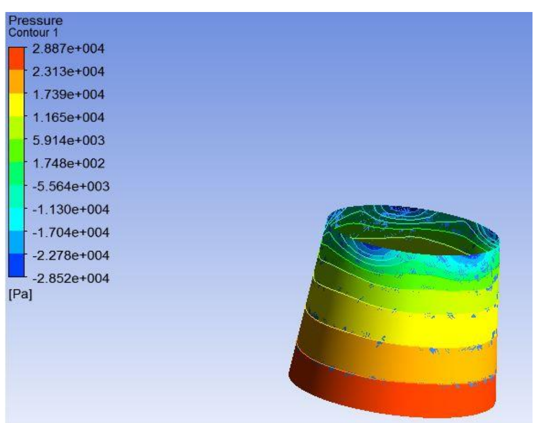


Figure 16 Pressure distribution on drafttube at 100% wicket gate opening

For Cavitation flow analysis

Vapor volume fraction distribution on the runner blade shown in Fig 16

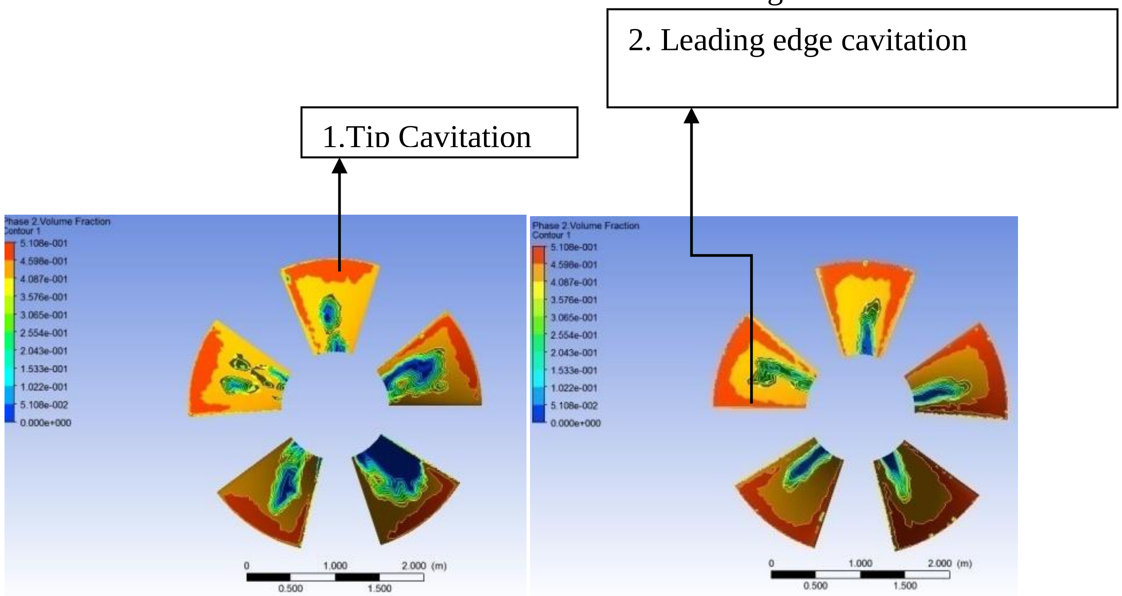


Figure 17 Cavitation flow at 80% Wicket gate opening (a) Pressure side (b) Suction side

Velocity distribution on the runner blade shown in Fig 17

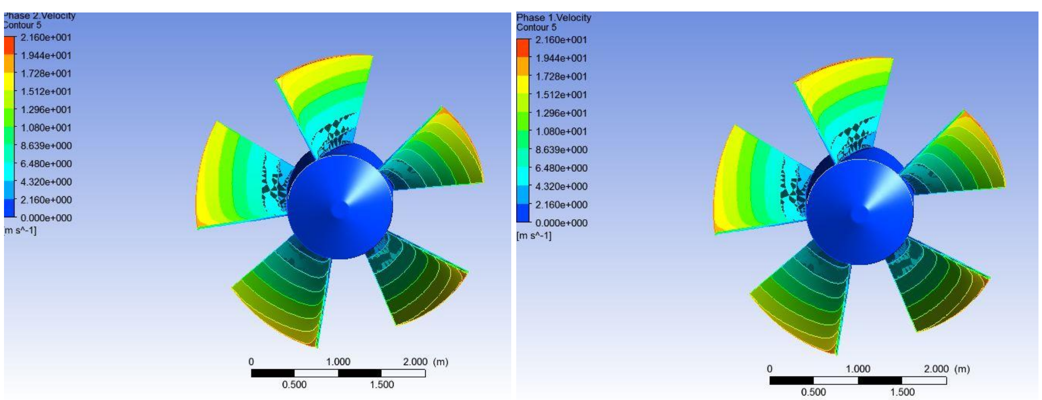


Figure 18 Cavitation flow at 80% wicket gate opening (a) Phase 1 Velocity distribution at rotor blade(b) Phase2 Velocity distribution at rotor blade

From the results it has been seen that velocity is increasing towards the tip and because of that vapor bubble generates at that point.

9.4 COMPUTATION OF LOSSES AND EFFICIENCY

Hydraulic parameters are calculated for performance assessment of Kaplan turbine under different wicket gate openings by using following formulas are given below.

Specific speed $N_s = \frac{N\sqrt{(P)}}{H_n^{5/4}}$ (9.1)

Available power $P = \rho g Q h$ (9.2)

Pressure $P_r = \rho g H$ (9.3)

(Head loss) domain $H_{loss} = \frac{(\text{Pr inlet} - \text{Pr outlet})}{9810}$ (9.4)

$H_{available} = \frac{(\text{inlet pressure casing – outlet pressure draft tube})}{9810}$ (9.5)

Turbine Efficiency (η) $= \frac{(\text{H available} - \text{H loss}) * 100}{\text{H available}}$ (9.6)

Or $\eta = \frac{\tau * \omega}{Q(Pi - Po)}$ (9.7)

Where τ is the net torque acting on the runner (N-m), ω is the angular speed(radian/s), Q is the discharge through the turbine (m^3/s), P_i is the total pressure at theinlet of the W.G (Pa) and Po is the total pressure at the exit of the draft tube (Pa).

9.5 HEAD LOSS AND EFFICIENCY OF TURBINE AT DIFFERENT WICKET GATE OPENING

In case of pure water flow condition at different wicket gate opening, the high pressure region were observed at the inlet side of the runner and the low pressure region was observed near the exit of the runner blade. The computed efficiencies by numerical simulation at four wicket gate openings are given in Table 6, which shows that best efficiency point is at 80% wicket gate opening. The 3-D stream line and pressure contours are observed and it is seen that at 80% WGO loss in runner is minimum. The effect of boundary layer at the blade surface can be seen at 80% WGO. Similarly the pressure and velocity contour variation on suction and pressure side of the runner is smoother at 80% WGO as compared to other operating regimes.

Table 6 Computational Head Losses in Various Domains at Different WGO

WGO	Wicket gate loss (m)	Runner Loss(m)	Draft tube Loss(m)	Total losses (m)	Turbine Efficiency (%)
60%	.86	1.1	.42	2.38	84.09
80%	.83	.93	.32	1.89	87.39
100%	.6	.98	.53	2.11	85.92
110%	.55	1.3	.61	2.46	83.55

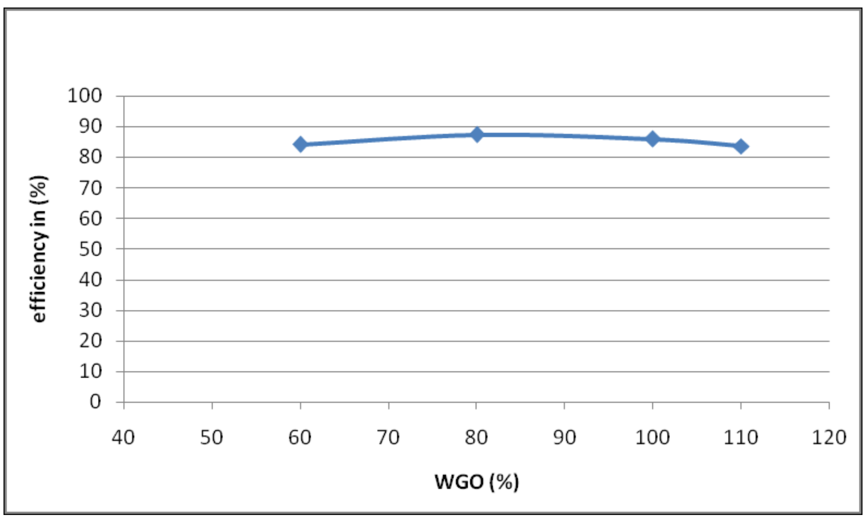


Figure 18 comparison of efficiency at different Wicket gate opening condition

9.6 COMPARISON OF EFFICIENCY AT DIFFERENT FLOW CONDITION

The efficiency of Kaplan turbine has been calculated at different flow condition. Maximum efficiency is 85.90 % has been found at 100 % W.G.O for pure water flow, in two phase cavitation flow efficiency has been reduced by 0.96 %.

Table 7 Output parameter of turbine at different flow condition

Flow condition	Pressure (pa) inlet	Pressure (pa) outlet	Torque(N-m)	Mass flow rate (m3/s)
Pure	164053	19602	451125	47.67
Cavity	164053	19602	446070	47.57

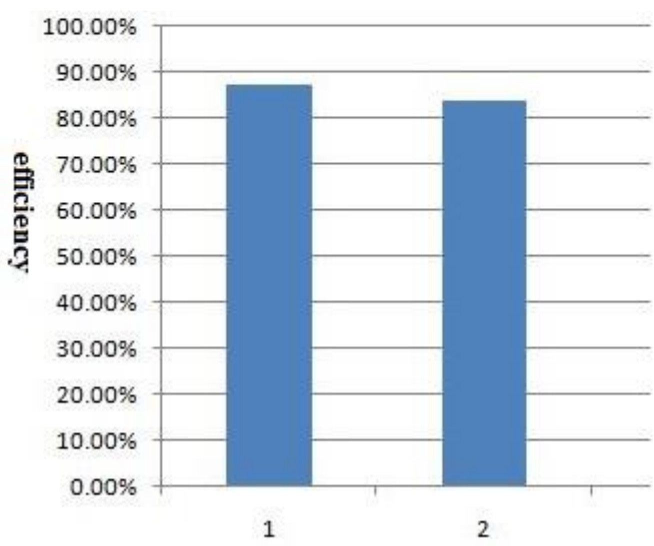


Figure 19 comparison of efficiency at different Wicket gate opening condition

(1) Pure water flow (2) Cavitation flow

CHAPTER 10

CONCLUSIONS AND FUTURE SCOPE

Under the present study of CFD based analysis of small hydro Kaplan turbine of 7 MW capacity, the effect of cavitation has been studied and analyzed. The turbine analyzed hereby works under the reaction principle to generate mechanical energy from the hydro energy with available head of 15 m. The flow analysis was carried out using Computational Fluid Dynamics using the FLUENT v 14 module available in University computational facility.

The following analysis were carried out during this study:

i. Study of single phase water based flow analysis to predict and validate the performance of the computational model.
ii. Study of two phase flow, including liquid water and water vapor, where water vapors are generated due to pressure reducing below the vapor pressure of water at 25^{o} C to simulate the effect of cavitation.
iii. Study of effect of cavitation on the performance of the Kaplan turbine, provided other conditions remain same.

The following conclusions drawn from CFD simulation of the Kaplan turbine for four flow conditions:

i. Based on CFD results obtained for single phase flow of water under the boundary conditions mentioned in the preceding chapters, it is observed that best efficiency point (BEP) of turbine is at 80% WGO and head losses are also found minimum at this gate opening.
ii. For the single phase water flow at different part load conditions, the different values of efficiency have been obtained as 84.09 %, 87.39%, 85.92% and 83.55% at different WGO, i.e.60%, 80%, 100% and 110% respectively.
iii. Under the two phase flow, vapor with water, the cavitation is found at suction side of the blade, blade rim and at outlet of the blade. The efficiency of turbine has been reduced by 0.96 % due to cavitation erosion.

In Computational Fluid Dynamics, the results of the flow analysis are highly dependent on the geometry of elements composing the flow domain. Although the mesh has been developed to accommodate greater detail to flow behavior at critical passages of water over blades and inlet passage, there still lies vast opportunity in organizing the mesh structure to further improve the accuracy of the results to match with the field study or a prototype analysis.

The mesh, as is employed is tetrahedron based control volumes, provide good results for complicated geometry of rotating fluid machinery. Though, the upcoming hexahedral mesh may be employed in a possible future work to observe the comparison of effectiveness.

The flow analysis involving cavitation has revealed the locations in the flow domain, where the bubbles of vapor are generated and displayed in the results in the form of vapor fraction concentration. These results are valid against the experimental investigation by Deschenes C *et al* [21]. A more elaborated experimental study can be done in a follow up work by studying the effect of this cavitation on the metallic surfaces by using a towing tank for the flow and rheological analysis of one hydrofoil based turbine blade.

REFERENCES

[1] http://www.mnre.gov.in/Ministry of New and Renewable Energy,Govt. Of India. Accessed on 31/1/2015

[2] http://www.mnre.gov.in/Ministry of New and Renewable Energy,Govt. Of India. Accessed on 31/1/2015

[3]http://www.mcnallyinstitute.com. Accesed on 31/1/2015

[4] http://www.voith.com. Accesed on 31/1/2015

[5] Punit Singh, Franz Nestmann, "*Exit blade geometry and part-load performance of small axial flow propeller turbines: An experimental investigation*", Experimental Thermal and FluidScience 34 (2010) 798–811.

[6]Punit Singh, Franz Nestmann, "*Experimental investigation of the influence of blade height and blade number on the performance of low head axial flow turbines*", Renewable Energy 36(2011) 272-281.

[7] Harsh Vats & R.P. Saini. "*Investigation on combined effect of cavitation and silt erosion on Francis turbine*". International Journal of Mechanical and Production Engineering (IJMPE) ISSN 2315-4489, Vol-1, Iss-1, 2012.

[8] Jain, S., R. P. Saini, and A. Kumar. 2010."*Cfd Approach for Prediction of Efficiency of Francis Turbine*".

[9] Tarun Singh Tanwar, DharmendraHariyani and Manish Dadhich, "*Flow simulation & static structural analysis of a radial turbine*", IJMET Volume 3, Issue 3, September - December (2012), pp. 252-269.

[10]Nilsson, H., and L. Davidson. 2003.*"Validations of CFD Against Detailed Velocity andPressure Measurements in Water Turbine Runner Flow*". *International Journal for NumericalMethods in Fluids*: 863–879.

[11] Jain and Kokubu 2011. *"CFD study of different turbulence models in kaplan turbine".*

[12] Hsing-nan Wu, Long-jengChen , Ming-hueiYu , Wen-yiLi , Bang-fuhChen. *“On Design and performance prediction of the horizontal-axis water turbine”*, Ocean Engineering 50 (2012) 23–30.

[13] Dr. Vishnu Prasad, *“Numerical simulation for flow characteristics of axial flow hydraulic turbine runner”,* Energy Procedia 14 (2012) 2060 – 2065.

[14]Kiran Patel, Jaymin Desai, Vishal Chauhan and ShahilCharnia. “*Development of Francis Turbine usingComputational Fluid Dynamics”*. The 11th Asian International Conference onFluid Machinery and The 3rd Fluid Power Technology Exhibition, Paper ID AICFM_TM_016November 21-23, 2011, IIT Madras, Chennai, India.

[15] Alokmishra, R.P. Saini and M.K. Singhal. “*CFD Based Performance Analysis of KaplanTurbine for Micro Hydro Power*”. International Conference on Mechanical and IndustrialEngineering, 2012 .

[16] JacekSwiderski. *“Recent approach to refurbishments of small hydro projects based on numerical flow analysis*”. Small Hydro Workshop. Montreal 2004.

[17] Dinesh kumar,Saurabhsangal and R.P. Saini. *“flow analysis of Kaplan hydraulic turbine by computational fluid dynamcs”*. IJAER, ISSN 0973-4562,Vol-8, Iss-6,pp-61-64, 2013.

[18] Peng Yu-cheng, Chen Xi-yang, CAO Yan, HOU Guo-xiang. *“Numerical study of cavitation on the surface of the guide vane in three gorges hydropower unit”.*journal of hydrodynamic2010,22(5):703 -708 DOI: 10.1016/S1001-6058(09)60106-2.

[19] Bernd Nennemann, Thi C. vu. *“Kaplan turbine blade and discharge ring cavitation prediction using unsteady CFD”.*2nd IAHR International Meeting of the Workgroup onCavitation and Dynamic Problems in Hydraulic Machinery and Systems Timisoara,RomaniaOctober 24 - 26, 2007.

[20] BraneSirok, MakoHoEevar, Igor Kern, Matej Novak. *“Monitoring of the Cavitation in the Kaplan Turbine”,* ISIE‘99 - Bled, Slovenia.

[21] C Deschenes, G D Ciocan, V De Henau, F Flemming, J Huang, M Koller, F A Naime, M Page,RQian and T Vu“*General overview of axial turbine project- a partnership for low head*

turbine development". Members of the Scientific Committee of the Consortium on Hydraulic Machines from:

[22] Huixuan Shi, Xuezheng Chu, Zhaohui Li, Qingfu Sun. *"Experimental Investigations on Cavitation in Large Kaplan Turbines",* 2011 Third International Conference on MeasuringTechnology and Mechatronics Automation.

[23] Pardeep Kumar, R.P. Saini. *"Study of cavitation in hydro turbines—A review",*Renewableand Sustainable Energy Reviews 14 (2010) 374–383.

[24] A Rivetti, CLucino, S Liscia, D Muguerza and F Avallan. *"Pressure pulsation in Kaplan Turbine"*.

[25] Laín, S., M. García, B. Quintero, and S. Orrego. 2010."CFD Numerical Simulations of Francis Turbines". RevistaFacultad De Ingeniería Universidad De Antioquia (51): 24–33.

[26] Dr. R.K. BANSAL "*Fluid Mechanics and Hydraulic Machine*".

[27]Pande V K, "*Effect of hydraulic design of Pelton turbines on damage due to silt erosion and its reduction"*. PowerPoint presentation, BHEL.

[28]Chung, T. J. "*Computational Fluid Dynamics. 2nd Ed., Cambridge University Press, Cambridge, UK"*.

[29] Date, A. W. "*Introduction to Computational Fluid Dynamics. Cambridge University Press, Cambridge"*.

[30] http://www.cfd-online.com/Accessed on 31/1/15.

[31] Versteeg, H. K. and Malalasekera, W. M. G. (2007). "*Introduction to Computational Fluid Dynamics":* The Finite Volume Method. Second Edition (Indian Reprint) Pearson Education.

[32] NHPC guidelines for selection of turbines, preliminary dimensioning and layout of surface hydroelectric power houses.(1993-07)

LIST OF PUBLICATION

Munendra Kumar and Alok Nikhade. *"Cavitation in Kaplan Turbines – A Review"* Journal of Material Science and Mechanical Engineering, ISSN: 2393-9095 Vol. 2, Number 5, April-june2015, pp 425-429.