CFD BASED PERFORMANCE ANALYSIS OF COATING ON FRANCIS RUNNER ERODED DUE TO CAVITATION

A DISSERTATION

Submitted in partial fulfillment of the

requirement for the award of the degree

Of

MASTER OF TECHNOLOGY

in

ALTERNATE HYDRO ENERGY SYSTEMS

By

AMANDEEP MEHTA

ALTERNATE HYDRO ENERGY CENTRE

INDIAN INSTITUTE OF TECHNOLOGY, ROORKEE

ROORKEE- 247667 (INDIA)

MAY 2016

CANDIDATE'S DECLARATION

I hereby declare that the work which is presented in the dissertation entitled as, "**CFD BASED PERFORMANCE ANALYSIS OF COATING ON FRANCIS TURBINE ERODED DUE TO CAVITATION"**, submitted in partial fulfilment of the requirement for the award of the degree of **Master of Technology** in "**Alternate Hydro Energy Systems**" in **Alternate Hydro Energy Centre, Indian Institute of Technology, Roorkee,** is an authentic record of my own work carried out during the period from May 2015 to May 2016 under the supervision and guidance of **Dr. M.K. Singhal,** Senior scientific officer, and **Dr. R.P. Saini,** Professor, **Alternate Hydro Energy Centre, Indian Institute of Technology Roorkee, Roorkee (India)**.

I also declare that I have not submitted the matter embodied in this dissertation for award of any other degree.

Date : May, 2016

Place: Roorkee **(AMANDEEP MEHTA)**

CERTIFICATE

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

(Dr. M.K. Singhal)

Senior scientific officer,

Alternate Hydro Energy Centre,

Indian Institute of Technology,

Roorkee – 247667

(Dr. R.P. Saini)

Professor,

Alternate Hydro Energy Centre,

Indian Institute of Technology,

Roorkee – 247667

ACKNOWLEDGEMENT

I feel much honoured in presenting this dissertation in such an authenticable form of sheer endurance and continual efforts of inspiring excellence from various coordinating factor of cooperation and sincere efforts drawn from all sources of knowledge. I express my sincere gratitude to **Dr. M.K. Singhal,** Senior scientific officer, and **Dr. R.P. Saini,** Professor, **Alternate Hydro Energy Centre, Indian Institute of Technology, Roorkee** for their valuable guidance and infilling support for the completion of the seminar work.

Last but not the least I am also grateful to **Dr. M.P. Sharma** Professor and Head **Alternate Hydro Energy Centre, Indian Institute of Technology, Roorkee** and all faculty members and staff of **Alternate Hydro Energy Centre, Indian Institute of Technology, Roorkee.**

I extend my thanks to all classmates who have given their full cooperation and valuable suggestions for my dissertation work.

I would like to express my humble respect and special thanks to my parents & others who directly or indirectly helped me during completion of this dissertation work.

Date : May, 2016

Place: Roorkee **(AMANDEEP MEHTA)**

ABSTRACT

In the field of hydropower, the designing of different components which is cost effective is a challenging work. For proper design of these components are need to analyse critically as the flow is turbulent as well as three dimensional in behaviour. As there are many components in hydropower plant, but the most important component is turbine which converts hydro energy into mechanical energy. There are many factors which affect the turbine life as well its efficiency so there is need to analyse these factors to increase the efficiency and the life.

As in earlier times model testing was used to test performance of turbines which was costly and time consuming. Large hydro power plants have high initial cost and model testing will also be costly. Computational fluid dynamics (CFD) is an alternative to model testing, it takes less time and low in cost. Performance predicted by CFD is also accurate and very near to model testing if used properly. Cavitation is a big problem in reaction turbines and this causes pitting on various region of turbines which leads to decrease in efficiency of turbines. This problem can be reduced by checking blade geometry, blade material whether suitable or not against problem of cavitation in CFD.

In the present study, the Francis turbine of rated capacity of 255MW is modelled in a CAD software and attempted to do CFD analysis in ANSYS 15 to investigate the effect of the cavitation on hydro turbine before and after the use of coating on turbine runner. Flow analysis of the Francis turbine is done in ANSYS FLUENT. The flow simulation has been carried out in cavitation flow condition to find out a suitable coating to prevent runner from cavitation and zones of runner affected due to cavitation.

Finally it is concluded that the modelled Francis turbine has maximum efficiency of 82.4% at rated discharge of 112.22 m^3/s at guide vane opening of 21.6^o. The best coating material out of the materials taken is found to be cavitec because of lower volume erosion rate as compared to other materials. It is also concluded that if turbine model is very big then smoothing should be low and mesh should be unstructured and method used should be tetrahedral patch conforming.

CONTENTS

PARTICULARS	**PAGE. NO.**

LIST OF TABLE

LIST OF FIGURES

S.No. Title Name	Page no.

NOMENCLATURE

LIST OF SYMBOLS

SYMBOLS	DESCRIPTION
P	Power generated by turbine
H_n	Nominal head
H_{max}	Maximum head
Q_n	Nominal discharge
Z_r	Number of runner vanes
Z_w	Number of guide vanes
B_o	Height of gate mechanism
N	speed of turbine
H	head on turbine
N_s	Specific speed of turbine
H_b	Barometric pressure in meters of water column
H_v	Vapour pressure in meters
σ	THOMA's cavitation coefficient
H_s	Suction head in meters
K_u	Peripheral velocity coefficient
g	Acceleration due to gravity
D	Diameter of runner
L	Length of draft tube
B	width of draft tube
D_o	Diameter of guide vanes circle
ω	Angular velocity of runner
α_o	Guide vane maximum angle at full load

CHAPTER 1

INTRODUCTION AND LITERATURE REVIEW

1.1 GENERAL

India has placed fourth in energy consumption all over the world. Due to increase in population and economic development energy consumption in India is increasing at faster rate. Due to increase in development and life style the demand is increasing day by day. In order to fulfill their demand today industries rely on the diesel fueled backup systems. India's energy planning, which is focusing on the two major objectives of high economic growth and providing electricity to all, is not getting it up to mark.

In 2012 domestic power demand in India was 918.2 billion units. It is expected that power demand will reach to 1640 billion by 2020 as the growth rate is 9.8%. At this speed, In the next eight years requirement of India will be 390 GW [1] which is almost 1.5 times its installed capacity now which is of 258.70 (GW) [2].

The difference of energy sources is increasing between rural and urban as well as in developed and developing nations. But the in India population without the access of electricity is 304 million and the national electrification rate is 75 % [3]. So to tackle with this energy problem we have to look at other energy resources. The difference between urban and rural energy availability can be reduced by using alternative source of energy. Currently a large part of population and industries in our country rely on coal and fossil fuels. Now we have to think about that how we can save our resources for the future generation. Load to increase energy availability and to reduce bad environmental impact India is making policies related to more use of renewable energy. There is large amount of potential in renewable energy sector which can solve our problem of supply and demand of energy.

Energy industry has an important role in our life. It is key source for the economic development of the countries like India. Each sector like manufacturing, agriculture, food industry, textile all needs energy. So to make development in these sector the energy sector should be encouraged which leads to increase in per capita energy consumption which increases the economic growth of the country according to human development report 2012-13 [4] is shown in Table 1.1 and in Fig. 1.1:

Table 1.1: Country wise per capita energy consumption [4]

s.no.	Country	Per capita energy consumption in kWh/year
1.	CANADA	17179
2.	U.S.A.	13338
3.	JAPAN	8076
4.	U.K.	6206
5.	CHINA	2476
6.	INDIA	917

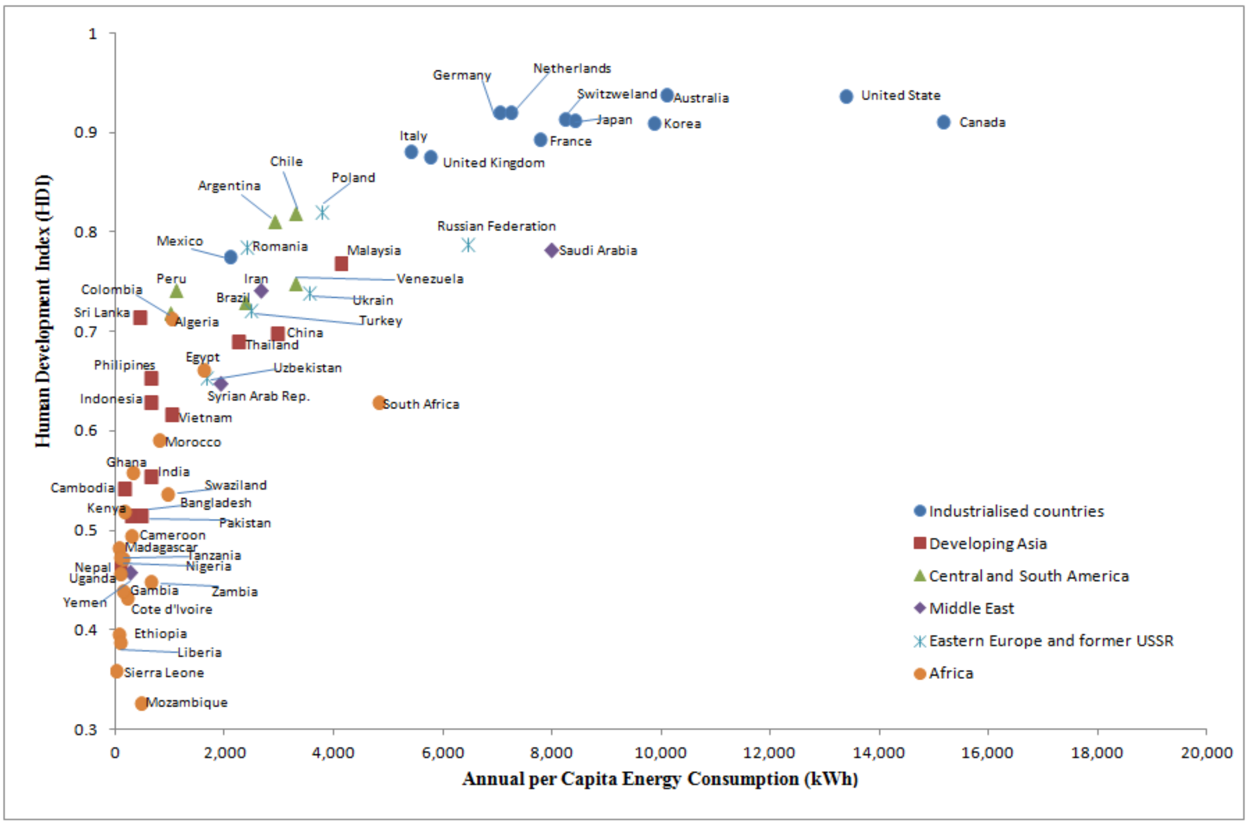

Fig. 1.1 Human development index v\s annual per capita energy consumption [4]

Energy is produced by resources like coal, crude oil, biogas, nuclear fuels, hydropower, natural gas, solar power, wind power, tidal energy, geothermal energy etc. from these resources some are conventional and non-renewable resources like coal, crude oil, nuclear fuels, natural gas and these resources are depleting fast and also affect the environment. But on the other side there are non-conventional and renewable resources which are clean in nature and doesn't pollute environment and also can't be depleted.

Hydro energy is much cleaner source than other sources as it produces power without any harmful pollutants and wastes. It produces GHG (greenhouse gas) but in small amount as compared to other sources. Hydro energy is a renewable source as water gets replenished due to water cycle. The main issue related to hydropower is that when a dam is created on a river then it changes the regime of river and also affects existing wildlife. Hydropower also reduces some water quality, but all these problems can be resolved by using fish ladder, silt extraction components, and by maintaining plant operations.

1.2 ENERGY SCENARIO IN INDIA

India's ranking in 2000 was at seventh position but now it is at fourth position. The country has the fifth-largest power generation portfolio worldwide [1] India has the availability of all type of energy resources including renewable. Coal as a fuel is used mostly. At present in India 60.3% of the installed electricity generation capacity is coal based. Out of total thermal installed capacity 86.6% capacity is coal based [2]. Other renewables such as wind, geothermal, solar, and hydroelectricity represent a 28.07 percent share of the Indian fuel mix. Nuclear has a 2.23% percent share [2]. Total installed capacity in the country stands at 258.7 GW [2] of which

(i) Thermal power accounts for 60.3 %

(ii) Renewable energy accounts for 28.07% [2]

In India energy production by the resources according to the C.E.A. report Jan 2015 are as in Table 1.2 and in Fig. 1.2:

Table 1.2: India power productions by the resources according to the C.E.A. [2]

s.no.		Power in GW
1.	Coal	156.19
2.	Gas	22.90
3.	Diesel	1.19
4.	Nuclear	5.78
5.	Hydro	40.86
6.	Other renewable energy resources (wind, solar, tidal etc)	31.69
	Total energy	258.70

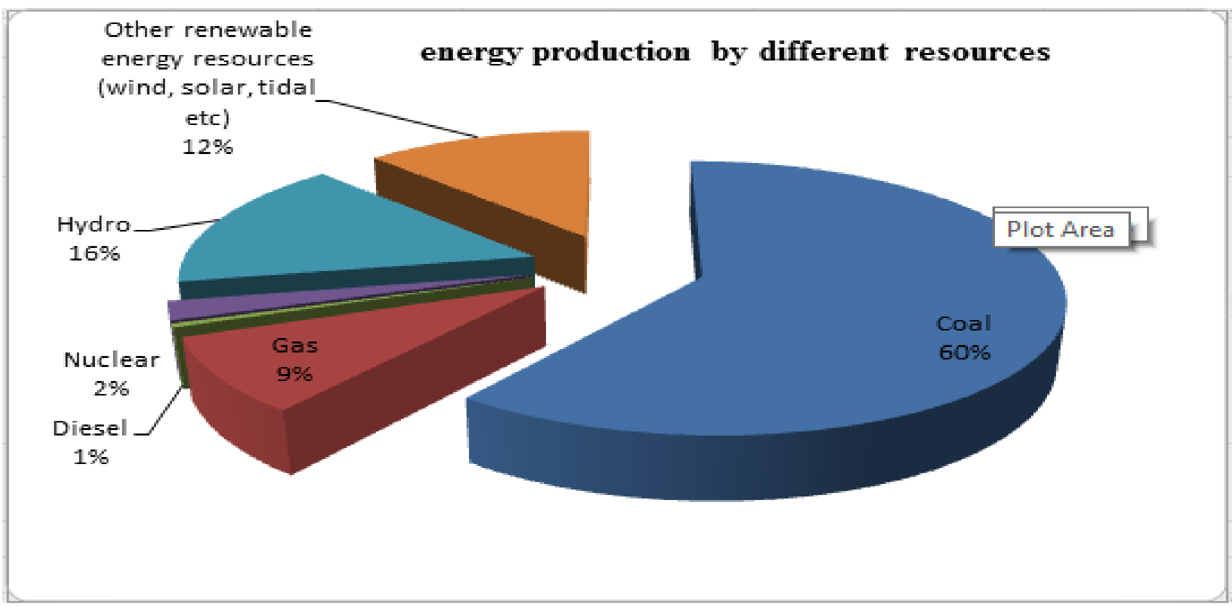

Fig. 1.2 Energy productions by different energy resources

To increase the generation capacity we can't rely on conventional resources but increase it by increasing renewable resources like hydropower, solar power, wind energy, biogas, and geothermal energy as the fossil fuels are depleting. Among the total renewable energy consumption the percentage of each type of energy is as in Table 1.3 and in Fig. 1.3 according to M.N.R.E. Feb 28, 2015 [5]:

Table 1.3: Installed capacity of renewable energy according to M.N.R.E [5]

s.no.	Renewable energy	Installed capacity in (MW)
1.	Wind power	22664.63
2.	Small hydro	4025.63
3.	Biomass power and gasification	1365.20
4.	Bagasse cogeneration	2818.35
5.	Waste to power	115.08
6.	Solar power	3382.78
	Total	34351.39

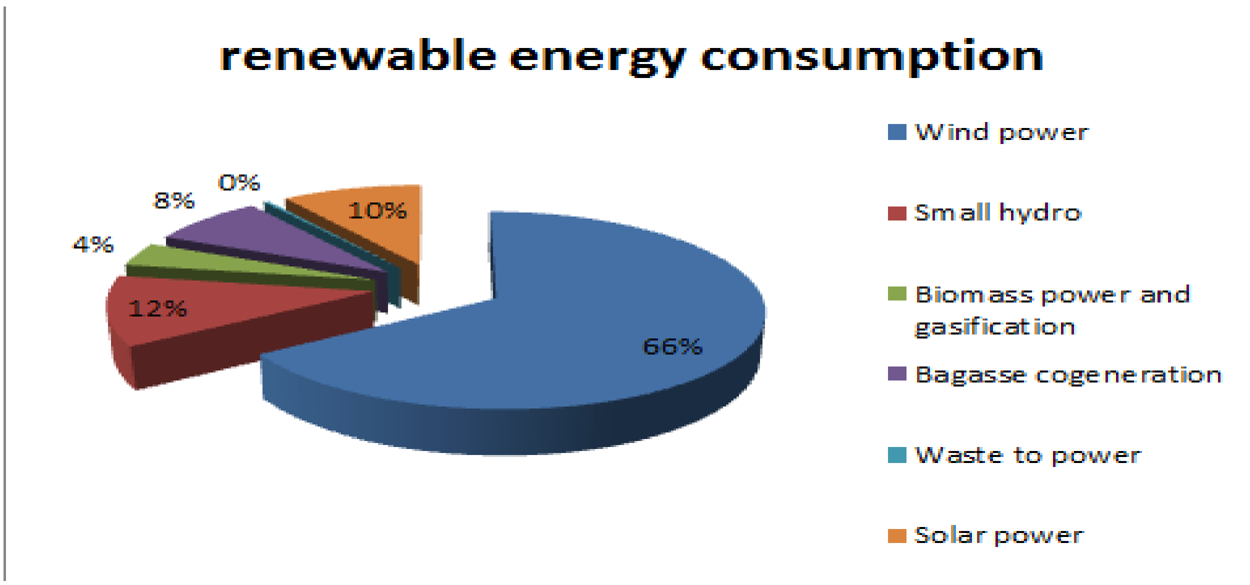

Fig. 1.3 Renewable energy production by different resources

Small hydro (<= 25MW) potential in India is approximated as 20,000 MW, but only 4025.63 MW is harnessed till date [5]. There are many type of schemes according to which small hydropower plant can be installed successfully in remote areas and can increase rural electrification. For conventional electricity distribution network small is low cost renewable energy source. Small hydro can be built at remote areas which has potential for small hydro so that we can avoid large electricity network for distribution to far areas. As there is no need for damming of river in small hydro so there is no GHG production and no wildlife get affected so it has a low environmental effect. This decreased environmental impact depends strongly on the balance between stream flow and power production [6].

1.3 SMALL HYDRO POWER PROGRAMME

Ministry of New and Renewable Energy has the responsibility of developing Small Hydro Power (SHP) projects up to 25 MW station capacities. The estimated **potential** for power generation in the country from such plants is about 20,000 MW [5] Most of the potential is in Himalayan States i.e. uttarakhand, himachal Pradesh and arunachal Pradesh as river-based projects and in other States on irrigation canals. SHP can be from natural resources (run of river scheme) or by diversion structure like dams and weirs.

Hydropower converts energy in from flowing water into electricity. That produced electricity is based on the net head over turbine and discharge .A typical hydropower includes dam, reservoir, penstocks, powerhouse and electrical substations. Dam stores water and create the head, penstocks carry water from reservoirs to turbines inside the power house, the water rotate the turbine inside the power house, which drives generator and produce electricity. The electricity is then transmitted to a substation where transformers increase voltage to allow transmission to homes, business and factories.

Different countries have set different capacity to define small hydro power shown as below in Table 1.4 [5]:

Table 1.4: Capacity limit of SHP according to different countries [5]

U.K.	<=5MW
UNIDO	<=10 MW
INDIA	<=25 MW
SWEDEN	<=15 MW
COLOMBIA	<=20 MW
AUSTRALIA	<=20 MW

CHINA	<=25 MW
PHILLIPINES	<=50 MW
NEW ZEALAND	<=50 MW

These days most of the SHP projects are developing with the help of private investment. Project development is economical these days and therefore many private sectors are looking for these types of projects.

Fig. 1.4 Picture of a small hydro power plant [5]

The focus of the SHP program in MNRE is to lower the cost of equipment, increase its reliability and set up projects in areas which give the maximum advantage in terms of capacity utilization. [5]

An estimated potential of about 20,000 MW of small hydro power projects exists in India. Ministry of New and Renewable Energy has created a database of potential sites of small hydro and 6,474 potential sites with an aggregate capacity of 19,749.44 MW for projects up to 25 MW capacity have been identified.[5]

Advantages:

- The gestation period is small as compared to large hydro projects
- Very low initial cost.
- No harm to the environment.

1.3.1 Hydro Power Project Classification

Hydro power projects are generally categorized in two segments i.e. small and large hydro. In India, hydro projects up to 25 MW station capacities have been categorized as Small Hydro Power (SHP) projects. While Ministry of Power, Government of India is responsible for large hydro projects, the mandate for the subject small hydro power (up to 25 MW) is given to Ministry of New and Renewable Energy. Small hydro power projects are further classified as:

1.3.1.1 On the basis of head:

This classification of the SHP based on head as per Central Electricity Agency shown in Table 1.5.

Table 1.5: Hydro power classification on the basis of head [5]

Classification	Head
Ultra low head	<3m
Low head	3-40m
Medium head/high head	Above 40 m

1.3.2 Components of SHP

Components of SHP can be categorised as follow:

1.3.2.1 Civil works components

It includes the following:

1. Weir
2. Intake works
3. Sand trap
4. Forebay
5. Power channel
6. Penstock
7. Powerhouse

1.3.2.2 Electro mechanical components

It includes the following:

1. Hydro turbine
2. Generator
3. Governor
4. Gate valves and other auxiliaries

1.4 HYDRO TURBINES

Hydro turbines are the mechanical equipment which convert potential energy of water at higher altitude into mechanical energy and this mechanical energy is then converted into electricity with the help of generators.

Hydraulic potential is present in the water flowing from higher level to lower level. This potential flow can be utilized by converting it into mechanical power on the shaft of turbines. On the way to achieve this conversion some losses are encountered which occur in the machine and some of them occur in the machine and some of them occur in the water conduit connected machine and water source.

On the basis of action of water on blades hydraulic turbine can be classified as which is as follows and can be further subdivided as shown in Fig. 1.5.

i. Impulse turbine
ii. Reaction turbine

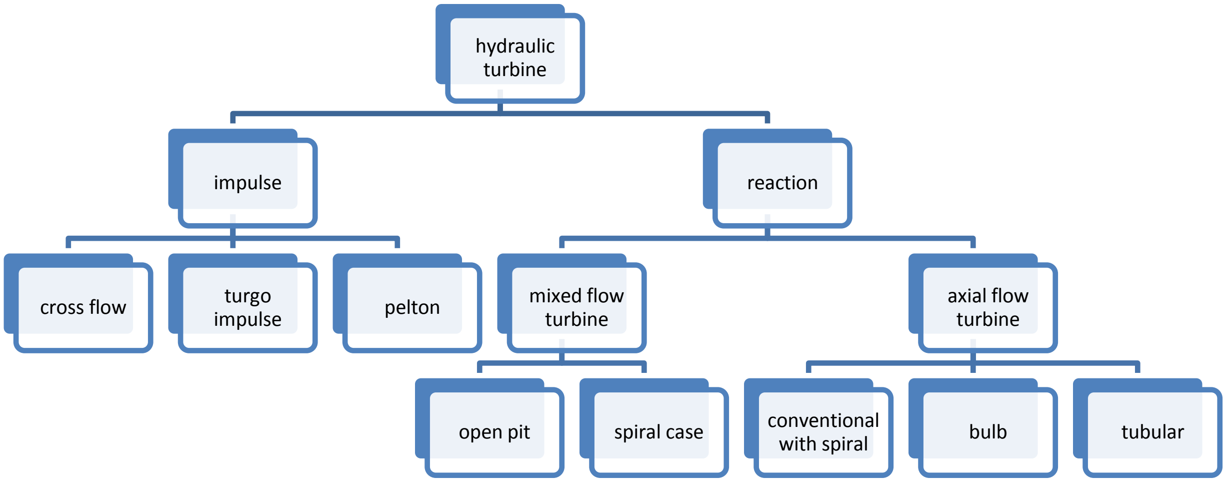

Fig 1.5 Classification of hydraulic turbines

Our work is on francis turbine which is a reaction turbine

1.4.1 Reaction Turbine

Reaction turbine encased in a casing full of water and water enters into casing with kinetic as well as pressure energy and due to shape of casing as in volute casing pressure energy converts into kinetic energy gradually, when water acts on the blades then due to reaction of the water the blades start moving and helps generator to move and produce electricity

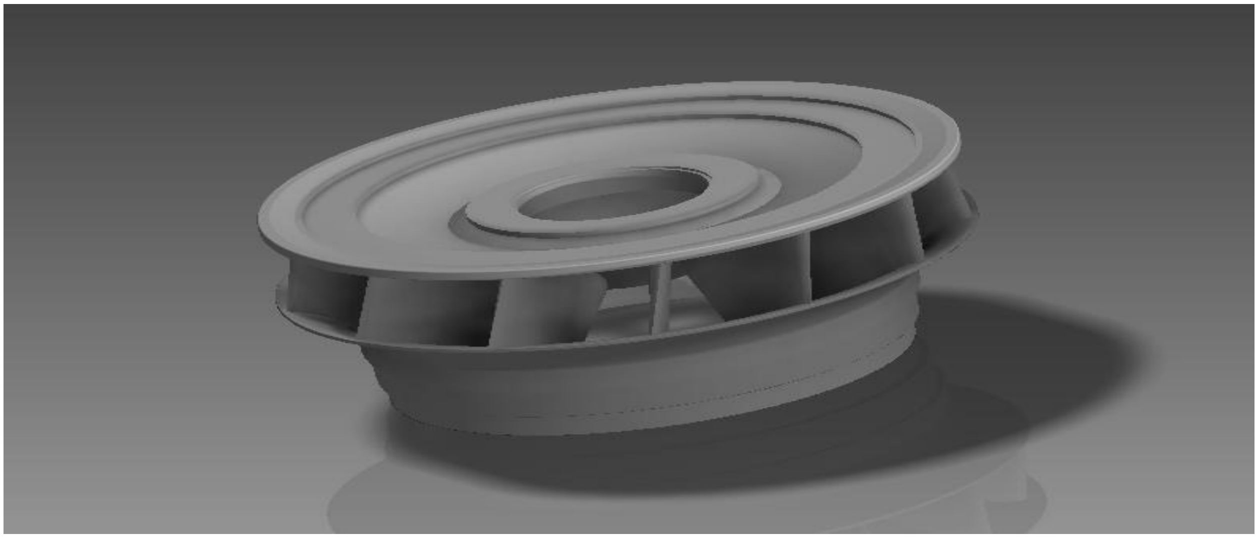

Fig. 1.6 Francis turbine

Principle of reaction turbine based upon newton third law. Most water turbines used under head lower than 30m are low head turbine and between 30 and 300m head are medium head turbine both type of turbines are reaction turbines. There is a drop in pressure in moving as well as fixed blades of reaction turbines. It is used mostly in medium and large hydro plants [7].

For example: - Francis turbine as shown in Fig. 1.6, Kaplan turbine, bulb turbine, tubular turbine, propeller turbine.

Reaction turbine are the turbine which has a combination of kinetic energy and pressure energy the inlet of casing and water when moves over the vanes then transfer their energy to vanes and vanes start rotating. The main types of reaction turbines are:-

1. **Radially outward flow reaction turbine**:

   An outward flow reaction turbine has flow of water from inlet to outlet and during entry of water at inlet it passes through the guide vanes to make the flow smooth. The function of guide vanes is to control the discharge passing through turbines. As the water flow over blades then it generate a torque and turbine start rotating [8].

2. **Radially inward flow reaction turbine**:

   It is similar to the outward flow turbine in construction and guide vane are outside the runner vanes and as flow move from outside to inside. The energy of the water decreases from outside to inside and transfer it to the runner vanes [8].

   Reaction turbine main components are:

   (1) Casing, (2) Guide vanes (3) Runner with vanes (4) Draft tube

   Casing: The casing is type of volute or spiral type. Casing is full of water and casing has a decreasing cross sectional area and it helps to enter the water from all sides at equal velocity. The shape of casing reduce the formation of eddies [8].

   Guide vanes: it is specially used for guiding water and makes the flow smooth and reduces the formation of eddies.

   Runner: runner has many blades of a fixed geometry and at equal angular distance. Runner is connected with hub, as water from casing enters into runner it has kinetic and pressure energy due to reaction of blades this energy dissipates and helps to rotate the runner.

Draft tube: it is used between turbine and the tail race, when water leaves the turbine then its pressure is vaccum pressure due to which it cannot leave the turbine so a draft tube at the end of turbine is used due to its increasing area the velocity of flow decreases and pressure of flow increases and water has higher pressure and negative head increases.

3. **Mixed flow reaction turbine:**

   In mixed flow turbine water enter radially and escapes axially, but function as same as inward flow turbine. It is a reaction turbine. Example is **Francis turbine** [8].

4. **Axial flow reaction turbine:**

   This is a reaction turbine in this type of turbine water enter in turbine axially and also leaves axially and flow is parallel to axis of rotation. Lower end is generally bigger to make hub or boss. No. of vanes are attached to the boss. Turbine can be vertical or can be horizontal.to control the discharge of water wicket gates are provided if blades are adjustable and so the wicket gates then turbine is **Kaplan turbine** as in Fig.1.7, if both are fixed then turbine is **propeller turbine**, if vanes are adjustable but the gates are not then the turbine is **semi Kaplan turbine** [8]

As our topic is concentrated to cavitation resistance and the cavitation prone turbines are reaction turbines [9] so here only reaction turbines are fully explained.

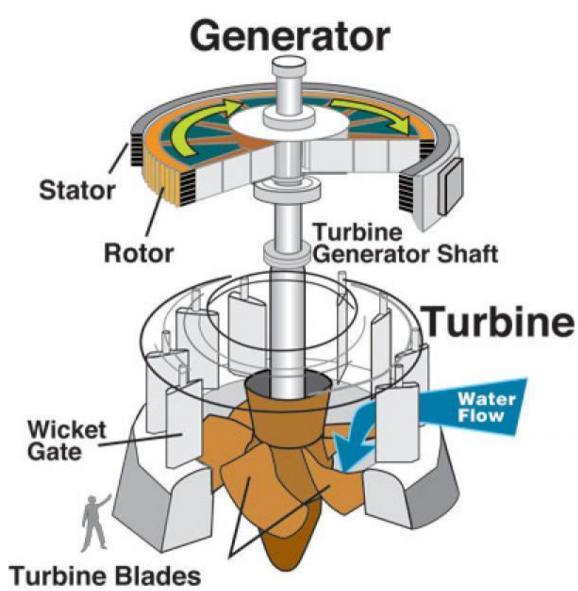

Fig. 1.7 Kaplan turbine [7]

There are two types of reaction turbines which are mostly used and more prone to cavitation i.e. Francis turbine, Kaplan turbine. But we opted Francis turbine for study of cavitation.

1.5 CAVITATION IN HYDROTURBINE

Cavitation is regarded as only limiting factor for design of high specific speed turbines working under either higher head, as well as for pumps supplying a larger amount of discharge. In fact the pumps with lowest possible dynamic depression (net positive suction head) have become the need of the day [9].

1.5.1 What is Cavitation?

As the pressure increased then boiling temperature also increases. Thus, if the pressure of flowing liquid become lower than the vaporization pressure of the liquid then it will start boiling , formation of bubbles starts, which then travel downstream resulting in two phase mixture flow and in actual practice these formation of bubbles takes place earlier due to dissolution of gases in the flowing liquid, the actual value of vaporization pressure is higher than the value of normal fluid [9]

1.5.2 Net Positive Suction Head

Pressure of any liquid can be measured in two ways [9].

1. In absolute scale in which the pressure also below the atmospheric pressure will be shown as positive.
2. In gauge scale there is pressure measured from atmospheric pressure.

If pressure on suction side of any pump is lower than atmospheric pressure then it is called as suction pressure or negative gauge pressure.

The pressure required at suction side to reduce cavitation should be more than vapour pressure, so the pressure required at suction side to reduce cavitation above the vapour pressure is called as NPSH (Net Positive Suction Head)

The suction side pressure of a turbine or pump is usually negative or suction and is called as total suction head.

$$H_s = h_s \pm h_{ls} + c_s^2/(2g) \quad \text{... (1.1)}$$

Where

H_s= total suction head

h_s= static suction lift

h_{ls} = loses in suction conduct

c_s = velocity in sump or tail race

Cavitation inception in liquids when NPSH =0

1.5.3 Net Positive Suction Head Required ($(\mathrm{NPSH})_R$)

- There is increase in pressure of liquid from inlet to outlet in case of pumps
- There is increase in pressure of liquid from inlet to outlet in case of turbines [9]
- So the two sides of blades of a runner cannot be at equal pressure. Therefore, trailing edge of pumps and leading edge of turbines blades are at lower pressure.
- So there is change in pressure from one end to another and at the point of lowest pressure , the pressure ΔH below the suction side pressure and it is the $(\mathrm{NPSH})_R$ or dynamic depression ΔH and its value depends on geometry of the machine.
- To reduce cavitation $(\mathrm{NPSH})_A$ > $(\mathrm{NPSH})_R$

  $(\mathrm{NPSH})_A$ = available NPSH
- $\Delta H = \Delta h_s + \Delta h_1$ ....(1.2)

  $\Delta h_s =$ Pressure loss due to internal friction and pressure change due to change in kinetic energy = $\lambda_1 C_2^2/(2g)$

  Δh_1 = $\lambda_2(U_2^2/(2g) + C_2^2/(2g))$ = fall in Head between the suction side and point of lowest pressure side of blade
- The dynamic depression depends on geometry of machine, on discharge^2 and $speed^2$.
- $\Delta H = \frac{(\lambda_1+\lambda_2)C_2^2}{2g} + \lambda_2 U_2^2/(2g)$ …(1.3)
- Discharge varies as C_2, and speed varies as U_2

1.5.4 Thoma's Cavitation Factor

Prof. D. Thoma suggested a dimension-less number, called as Thoma's cavitation factor σ (sigma), which can be used for determining the region where cavitation take place in reaction turbines. When it is applied to $(\text{NPSH})_A$ then it is called as plant coefficientσ_A, if it applied to $(\text{NPSH})_R$ then the thoma coefficient is σ_R[10]

NPSH=$H_b - H_s - H_v$; …(1.3)

$\sigma_A = (\text{NPSH})_A / H$ ... (1.4)

$\sigma_R = (\text{NPSH})_R / H$ ... (1.5)

To prevent cavitation $\sigma_A > \sigma_R$

As long as Cavitation does not take place till then cavitation does not depends upon the value of suction height of a machine.

- If (Q,H,N) are kept constant then machine will work at its best efficiency, even though suction height is changed till such time cavitation effect its performance.
- If $\sigma_A = \sigma_R$ then cavitation inception will take place, then cavitation factor at this stage is called as critical cavitation factor (σ_{CH}).
- If $\sigma < \sigma_{CH}$ upto some $\sigma = \sigma_{CH}$

where H_b is the barometric pressure head of water in m, H_s is the suction pressure at the reaction turbine outlet in m of water or distance between tail water race level and turbine level, H denotes net head on turbine of water in m.

Thoma's cavitation factor (σ) value for a particular type of turbine is to be calculated from above equation. This value of Thomas cavitation factor (σ) when compared with critical cavitation factor(σ) for a particular type of turbine.

For cavitation free operation of turbine, it is necessary that:

$\sigma > \sigma_{CH}$

The σ_{CH}can be obtained by the empirical relations as given below for different turbines.

For Francis turbines,

$\sigma_{CH} = 431 * 10^{-8} * N_s^2$ ... (1.6)

For propeller turbines,

$\sigma_{CH} = 0.28 + [\left(\frac{N_s}{380.78}\right)^3 / 7.5]$ ... (1.7)

Where N_s is the specific speed of turbine.

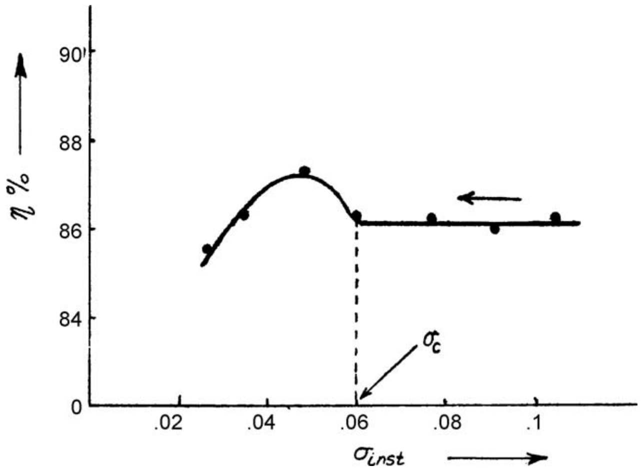

Fig. 1.8 Variation of efficiency with respect to cavitation factor, σ [10]

Fig.1.8 shows the change in efficiency with Thoma's cavitation factor (σ). As suction head of turbine is increased then there is corresponding decrease in the σ and we will obtain a constant efficiency trend and below critical cavitation factor efficiency increase for some lower value due to formation of bubble cushion in flow and after that cavitation increase severely and efficiency starts decreasing rapidly.

1.5.5 Causes and Types of Cavitation

The main cause due to which cavitation problem happens in hydraulic turbine are due to the conditions as given below:

- Design geometry of the hydraulic turbine.
- To tackle various type of load conditions by doing changes in operating conditions. [10].

The structural vibrations, sound wave emissions and hydrodynamic pressures measured are the basis of analysis of cavitation and formation of cavities in the turbine. Accordingly, in Francis turbine the cavitation is caused by the change in fluid flow when the flow encounters the blade. The velocity increase during the flow causes a decrease in level of dynamic pressure which causes cavitation due to formation of different flow pattern.

- Formation of fluid flow behavior over the leading edge of the runner blade.
- Formation of fluid flow behavior over the trailing edge of the runner blade.
- Once the fluid passes over the leading edge the it forms a swirl in draft tube.

- Formation of inter-blade vortex.

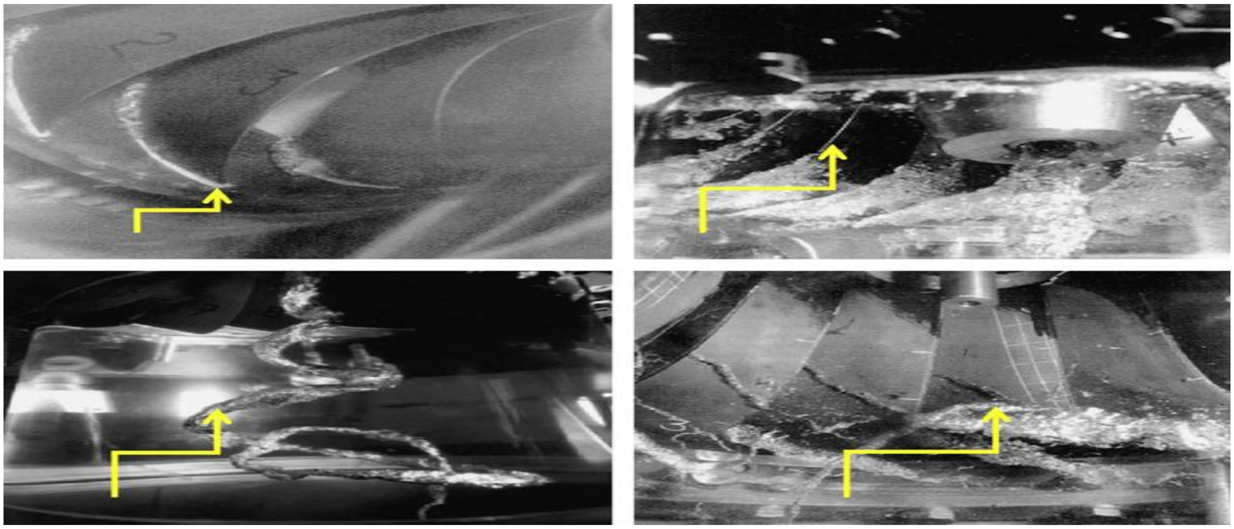

Fig 1.9 Cavitation damages in various parts of Francis turbine (a) leading edge cavitation, (b) trailing edge cavitation, (c) draft tube swirl and (d) inter-blade vortex cavitation.[10]

The four types of cavitation detected in Francis turbine are shown in fig.1.9 above which shows how leading edge cavitation, trailing edge cavitation, draft tube swirl and inter blade vortex cavitation can damage the Francis turbine.

- The leading edge cavitation as shown in (a) is a serious issue that is likely to erode the blades deeply. It is due to liquid flow obstruction and change in water pressure at the point of contact and then water bubbles formation.
- The trailing edge cavitation as shown in (b) is a noisy type of cavitation that minimises the machine performance and initiates blade erosion.
- Draft tube swirl in draft tube as shown in Fig. (c) Produces low pressure pulse which leads to hydraulic resonance and production of high amplitude vibration due to which power plant structure gets disturbed.
- Fig. (d) shows inter blade vortex type cavitation or sometimes we say von Karman vortex cavitation and mostly occurs in trailing edge of turbine components.

1.5.6 Turbine Parts Susceptible To Cavitation

The different type of turbine at different plant operating conditions leads to different cavitation affected turbine parts are summarized as below [10]

In case of Francis turbine cavitation may be severe in the blade trailing edge zone and blade leading zone and as shown in Fig. 1.10. Other components such as guide vanes, draft tube and wicket gates have lower possibility of cavitation damages as compared to the blade. Moreover in many studies, the cavitation defects in Francis turbines are described only on the profile of blade [13].

Cavitation affected turbine parts are described below:

- In Pelton turbines:

Bucket: The geometry of the Pelton bucket is free from cavitation. Due to initiation of rough asperities on the bucket by frequent impacts of erosive material or silt in water there is an increase in cavitation.

- In Bulb turbine:

Blade surface, result of CFD simulation for flow through the bulb turbine in Fig.1.10 shows the

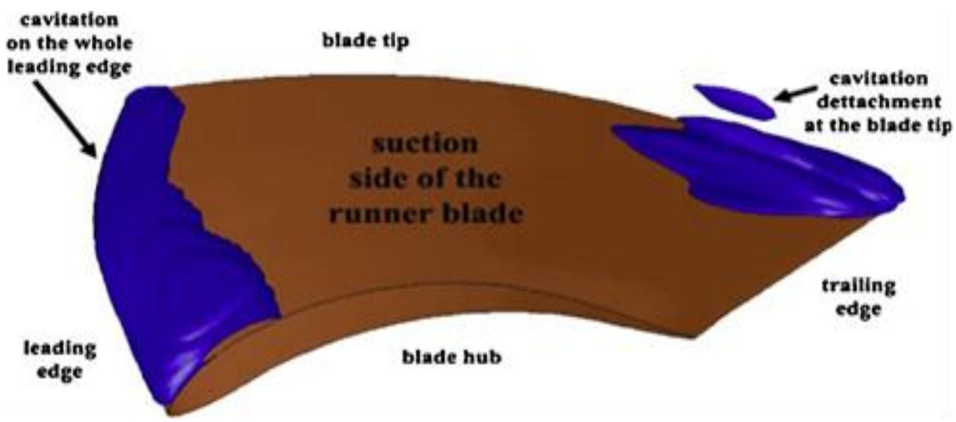

Fig. 1.10 Typical eroded areas of a Francis runner.[13]

Cavitation over blade profile and it indirectly affecting the plant efficiency

- In Francis turbine:

Leading edge, trailing edge, draft tubes, wicket gates, guide vanes.

- In Kaplan turbine:

Guide vanes, Runner blades.

1.5.7 Cavitation Damage

Cavitation damage may be due to following reason [9]:

1. Erosion: it is removal of material due to impact of suspended solid particles on the metallic surface. It happens mainly in Pelton turbine due to high flow velocity. It also occurs in reaction turbines when water has silt in it.
2. Corrosion: Due to chemical or electrolytic action of liquid on the material of the machine. Cavitation accelerates corrosion damage.
3. Cavitation pitting: this is the erosion of material under the pressure produced due to collapse of bubbles, formed by cavitation.

   The harmful effect of cavitation is:

   1. The pitting of the solid entity of turbine
   2. Fall in efficiency of turbine
   3. Variation in hydraulic machines caused by periodic nature of cavity collapse.

1.5.8 Cavitation Prevention

Motive of this study is to completely eliminate the cavitation or to reduce it to a minimum value. So most effective way is to stop the cause. It is impossible to design the hydro machines free from cavitation with only theoretical analysis but practical knowledge is also required.

Cavitation prevention at design stage

To design a hydraulic machine these practical points are important:

- Relation between runner type and cavitation coefficient must be found out with safe suction height.
- Sharp curvatures, abrupt corners, which produces vortices, eddies, separation are all responsible for cavitation should be eliminated.
- Model testing should be used to design turbine against cavitation.
- Cavitation resistant material should be used also help in minimising cavitation damage at the designing stage.

  Rubber or various type of sprayed material are quiet good for mild cavitation, when cavitation is moderate then 18-8 stainless steels, ampco bronze and 13% chromium gives good performances. For large damages use stellite but not giving upto mark performance.
- Good surface finish is also reduces cavitation and it is good to be used against cavitation.

Cavitation prevention of installed hydraulic machine

In these cases we can't make large changes to prevent cavitation. These methods are seems good to prevent cavitation:

- Replacement of turbine part by a tougher material. This can be done by remove out the earlier material and replace it by more tougher material like stainless steel 301, stainless steel 308, stellite 21,cavitec , stainless steel 1020 . Properties of all these materials are shown in Table 1.6.
- Cavitation sometimes allows happening when not at drastic stage so that it can arrest itself by creating a liquid cushion.
- One of the most effective solutions is to admit air at low pressure region. Larger air content in the liquid has been seen to cushion the collapse and reduce its destructive effects. It reduces the cavitation by simply breaking the vaccum.

Table 1.6: Mechanical properties of some coating used to reduce cavitation

Properties / Coating	Density (g/cm^3)	Tensile strength(MPa)	Hardness (rockwell)	Poisson ratio	Elastic modulus(GPa)
Stainless steel 301	7.88	120	B86	0.26-0.27	193
Stainless steel 308	8	585	B80	0.27-0.30	193
Stainless steel 1020	7.87	394.72	B64	0.29	200
Cavitec	8.0	860	B25	0.3	-
steel 13/4	7.33	579	B81	0.288	206
Stellite 21	8.33	1000	B27-40	0.27	245

1.6 LITERATURE REVIEW

1.6.1 Cavitation in Hydro Turbines

Pradeep Kumar and R.P. Saini [10]

Different Reaction turbines like Kaplan turbines and Francis turbines are used for low and medium head hydro plant locations. Cavitation is the main reason behind the decreasing efficiency of the reaction turbines. Reaction turbines like Francis have more chances of cavitation and zones in the turbines that have more threat of cavitation are called as forbidden zone. Cavitation is the problem in which there is formation of cavities on the turbines due to which pitting of the turbines starts. Cavitation is phenomena in which there is a formation of vapour bubble in low pressure region and they collapse on metallic surface in high pressure region and material from that part get pitted. Some of the investigators said that at the place of changing the blade profile in turbines to overcome cavitation we can use different materials and coating over them to reduce cavitation. So experimental and theoretical analysis are necessary to check the effect of cavitation as cavitation is different at different parameters. CFD based analysis is a cost effective and producing up to the mark results as compared to model testing.

Peng Yu Cheng And CHEN Xi-Yang [11]

In three gorges power plant erosion on large area such as rust and cavitation on guide vanes was found out. At first, a numerical analysis is done using the **Reynolds-Averaged Navier-Stokes (RANS) equations** on the operating point in actual operating condition on turbine given by ALSTOM. Passage of flow from inlet of casing to the outlet of draft tube is included in the fluid domain. According to the result produced the static pressure on the guide vanes are much higher than critical cavitation pressure.

Then a tiny protrusion is taken on guide vane and the detailed flow in 2-D is studied. Results from the RANS simulation are considered and then the flow under LES is studied. Then in results it is shown that there are some regions which have lower pressure than water vapour pressure.

Xavier Escalera,, Eduard Egusquizaa , Mohamed Farhatb , Franc-ois Avellanb , Miguel Coussirata [12].

In order to detect cavitation a experimental study has been done on the hydraulic turbine. The method of experiment is based on structural vibration, hydrodynamic pressure and acoustic emission in the machine. These techniques have been examined on the real prototype which was affected by the cavitation. One Kaplan, two Francis and one Pump-Turbine have been examined in this field. Other than this, one Francis has been tested in a laboratory. As leading edge cavitation leads to erosive losses, bubble cavitation leads to hamper the machine performance and draft tube swirl leads to the disturbance in plant operation. The model is based on the fact that an isolated bubble will remain spherical throughout the liquid. So, the generalized **Rayleigh–Plesset equation** is a valid equation for the bubble growth and to find the value of radius of the bubble generated, R_B(t) we can use this, provided that the bubble pressure, p_B(t), and the infinite domain pressure, $p_\infty(t)$; are known:

$$\frac{p_B(t) - p_\infty(t)}{\rho} = R_B \frac{\mathrm{d}^2 R_B}{\mathrm{d}t^2} + \frac{3}{2}\left(\frac{\mathrm{d}R_B}{\mathrm{d}t}\right)^2 + \frac{4\nu}{R_B}\frac{\mathrm{d}R_B}{\mathrm{d}t} + \frac{2\gamma}{\rho R_B}$$

Hongming, Zhanga, Lixiang Zhang [13].

This paper tells about numerical analysis of Francis turbine working under part load operation having cavitating turbulent flow. Open FOAM perform this code. RANS equation is employed and the turbulence model used is **k–ω SST**. Model used was of mixture type and there was one mode of mass transfer. The fluid domain consists of spiral casing, guide vanes, runner and draft tube and consists of 3D mesh which is unstructured tetrahedral mesh. The finite volume method is used to solve the generating equation.

Liangliang Zhan ,Yucheng Peng ,Xiyang Chen [14]

Cavitation is a common problem in hydraulic turbines. To reduce this problem of cavitation, cavitation monitoring, an experimental investigation, on the Kaplan turbine to check high vibration produced by cavitation. Four 53 kHz natural frequency high-level accelerometer mounted on the turbine, and corresponding to them four data acquisition channels is used. The experiment is carried out for output of no- load, 30 MW, 66 MW, 100 MW, 115 MW and 130

MW. The results show that the adopted experimental setup can precisely overcome the cavitation signal, and the 115 MW output in the 21.05-meter working head is suitable for the continued operation.

William Duncan, Jr. [15]

This paper tells us about the runner material, cavitation, its mitigation , its repair procedures, cavitation damage inspection, frequency of inspection and repair, cause of pitting, runner modification, decision on best approach to repair .

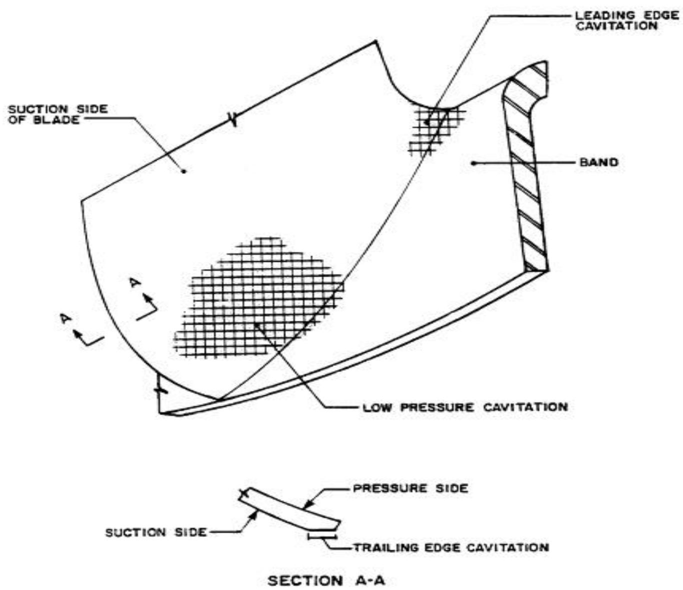

Fig. 1.11 Low pressure cavitation on Francis blade [16]

Branko Bajic et al. [16]

In this cavitation is investigated in prototype and model experiment by checking noise sampling, signal processing and analysis, data processing in vibro-acoustic diagnostics of turbines. Different development and improvement are done in these methods and also weak points are checked. These techniques detect many causes of cavitation. Brief assessments of new techniques, weak points of the practice and developed improvements as well as examples of uses, are represented in paper. The Fig. 1.11 shown above shows the different parts for cavitation on the Francis runner blade.

Here, noise power, I(P), at the turbine power P, used as an estimate of the cavitation intensity, could be simply modeled by the exponential rule

$$I(P) = \sum_{m=1}^{M} I_m(P)$$

Sebastian Muntean et al. [17]

The paper represents behavior of Francis turbine and numerical analysis of increase in cavitation. A two phase mixture approach is used for cavitation flow, and a model is employed for checking. This model is validated on a benchmark problem for the hemispherical ogive. Then Francis turbine cavitation flow model is used to investigate the cavitation. In starting, the steady non-cavitating relative flow is analyzed in a runner inter blade channel using the mixing interface method. Second, the cavitation model is used. Results for cavity size, shape and extent and pressure distribution on blade without or with cavities are discussed cavity shape and extent, as well as for the pressure distribution on the blade with and without cavity are presented and discussed.

1.6.2 Coating on Hydro Turbines To Reduce Cavitation

T. Pramod, R.K. Kumar, S. Seetharamu and M. Kamaraj [18]

Cavitation can be seen in both pumps and under-water components like some under water turbines of hydro power plants. There is also wastage of turbine material due to particle induced erosion when silt laden water is used.

In order to reduce the severe erosion due to silt erosion and cavitation, HVOF (high velocity oxy fuel) is used which has hard coating of 300 to 500 microns. Cavitation resistance of **tungsten carbide coating processed through HVOF** is represented in this paper.

Spray rate of powder and spray distance like process parameters affect the porosity of the coating. Cavitation erosion resistance doesn’t get affected by Small variation in hardness. Porosity in coating affects the rate of metal losses due to cavitation and it will be high in porous region. So actual metal loss during the initial stage of cavitation is different for different coatings, so metal loss is function of porosity of coating. Coating should have higher toughness and low porosity to reduce cavitation. Metal loss increases with increase in roughness of cavitated surface with time and follow similar trends.

Hart D. and Whale D. [19]

A weld surfacing alloy has been made and examined to prevent cavitation erosion in hydro turbines. A high strain, work hardening austenitic stainless steel gives good resistance to cavitation damage shows by a metallurgical evaluation.

Cavitation-erosion resistance increases up to 8 times as compared to 308 stainless steel shows by field testing. Cavitation erosion rate is a parameter to check which material ranks good as a repair material such as austenitic stainless steel. This led to the development of highly resistive material known as cavitec

The chemistry of the weld has been studied to produce an economical iron-base matrix, which has same type of planar deformation as observed in cobalt based alloys. To continuously guard material from cavitation characteristics like surface hardening and martensitic transformation helps a lot, due to which coating material become efficient surface barrier. CaviTec weld surfacing has shown best resistance toward cavitation resistance. Due to availability of high resistant alloys and increase in cost of energy and increase in damage due to cavitation leads to decrease in allowable cavitation metal loss.

1.6.3 CFD Analysis For Hydro Turbines

Prasad et al [20]

This paper presents that in earlier days model testing was used to evaluate turbine performance, it is costly and time consuming for different design alternatives while optimizing design of turbine. Detailed information of flow can be assessed by making a best design in space in CFD. This paper compare efficiency at different guide vane angle CFD analysis and through experimental setup and its comes out approximately same. The turbulence model used for analysis is using $\kappa - \omega$ SST turbulence model in ANSYS CFX.

Drtina and sallberger [21]

The present paper discusses the basic working and characteristic of hydraulic turbines, with special work on the use of computational fluid dynamics (CFD) as a tool which is being fastly used to study 3D flow inside the fluid machinery. Two examples have been taken for study. In first example it compares experimental data and 3D Euler and 3D Navier–Stokes results for Francis runner flow. The second example shows how to calculate performance of Francis turbine by numerical simulation. It also shows about flow separation, loss source and detection of low pressure level leads to cavitation and improved with the aid of CFD.

Wu et al [22]

In this study runner and guide vanes are set to maximum extent under geometrical constraint of machine and CFD code and automatic mesh generator are enabled for efficient design optimization of turbine components. It is acted on rehabilitation project of Francis turbine with 3% increase in peak efficiency, decrease in cavitation and 13% increase in power. The performance of the new designed guide vanes are checked by model tests, and exceeds required improvement.

Lipej [23]

In this paper it optimizes the design of axial flow turbine by using CFD. It describes the complete procedure for the design optimization of axial runner and this procedure consists of a design program, numerical flow analysis. This approach advantage is that it reduced design time. This result of the optimization method helps us show how each design process parameter influences the energy production and cavitation behavior of axial runners. By using the algorithm in combination with 3D numerical flow analysis was found to be very useful for the design of fluid machinery, as initial geometry from the designed program has a relatively high efficiency.

Khare et al [24]

Computational fluid dynamics (CFD) has emerged as a most effective tool for finding flow equation solution whose analytical solution is not possible but numerical solution is possible. CFD also minimizes the need of requirement of model testing. Its work deals with the 3D flow simulation of the Francis turbine passage i.e. casing, stay vanes, guide vanes, runner and draft tube and then computation of various losses and efficiency at different operating regimes. The computed values and variation in performance parameters are found to produce close comparison with experimental results.

Viscanti et al [25]

In this paper it shows about fluid dynamics optimization of new blade profiles and meridonial blade channel is achieved through CFD. To increase blade efficiency and to reduce cavitation phenomena in the new runner a trial and error approach is used to modify blade geometry. Runner's performance was approximately 90% at design point as shown by analysis. Existing results from CFD simulation pointed out that swirl flow occurred at the exit of runner. By comparison of runner's performance between simulation results and experimental data from previous work, it can be used to simulate runner's performance of the Francis turbine.

Patel et al [26]

It tells about Francis Turbine having high head and specific speed (Ns=80 to 90) is very complicated in nature. Basic design geometry of Francis turbine is made on basic fluid dynamics turbo machinery principal. 3-D flow steady state, single-phase mixture CFD analysis is conducted in all the components (Runner, Draft tube, Spiral Casing, Guide vane) of Francis turbine and fluid flow behavior along with losses is checked. CFD analysis is used to optimize the required performance of Francis turbine. At BEP and part load turbine performance is checked and it shows advancement over previous design.to check performance of the design created CFD analysis is used.

1.7 OBJECTIVE OF THE PRESENT STUDY

According the literature reviewed it is found high specific speed turbine like Francis turbine is much prone to cavitation. Depending on different discharge and variation of pressure the cavitation region changes, turbines are the most costly components in power plants. So they require coating to reduce cavitation. It is proposed to carry out CFD based analysis on Francis turbine. The following objectives are proposed as:

i. To design a Francis turbine for the given data.

ii. To make 3-D model of Francis turbine in the Autodesk inventor software.
iii. To investigate the effect of discharge or (guide vane opening) over the hydraulic efficiency of turbine.
iv. To check the type of meshing suitable for the Francis turbine modelled.
v. To visualize the region affected due to cavitation on change of discharge.
vi. To compare the different coatings on the runner to reduce effect of cavitation on the runner.

CHAPTER 2

COMPUTATIONAL FLUID DYNAMICS IN HYDRO TURBINES

2.1 INTRODUCTION

Computational fluid dynamics (CFD) is a branch of science in which it deals with problems of fluid flow and solves it by various numerical techniques. The laws which governs a fluid flow problem are represented by systems consist of partial differential equation of continuity equation and Reynolds average nervier stokes equation. CFD these days is used widely to design fluid machinery like (turbine, pump) by investigating about the various problem during the simulation of the 3-D fluid flow inside the fluid domain of runner, guide vanes, casing and draft tube. The main focus in the design of reaction turbines is on blades as they convert the pressure energy of water into kinetic energy which is used to generate power and the generation of blade profile can be done with the software in CFD which leads to decrease in experimenting loads. The fluid flow behavior is analyzed by the colored contours, plots and some quantitative analysis to fully understand the mechanics behind the flow. This CFD technique is essential these days for the design, optimization and testing of the machinery. CFD works on the laws which include the following laws:-

i. Newton's second law
ii. Law of conservation of mass
iii. Law of conservation of energy

In CFD partial derivative and integral of these equations discretize and converted into algebraic form and then solve it at different point of space and time by finite volume method and show result after integral of the results of different points.

2.2 APPLICATION OF CFD

Application of CFD is as given below

i. Construction of design: The design of fluid machinery or any other system can easily be made in the CFD which helps in the actual visualisation of system and phenomena acting within the system

ii. Performance prediction: With the help of CFD, performance of the system designed can be predicted, i.e. under different discharge, different suction head and different guide vanes how the performance of the system will change.
iii. Saving in cost and time due to better and faster analysis at the place of model testing which is costly and time consuming.

2.3 BASIC CONCEPT OF CFD

CFD works on the three basic laws and these laws governs the fluid flow inside the system:

i. Conservation of energy: energy neither be created nor be destroyed but change from one form to another like potential energy to kinetic and in some heat energy.
ii. Conservation of mass: As the mass of the flow will remain conserved.
iii. Conservation of momentum: As rate of change of momentum of any fluid particle is equal to the addition of forces on the particle in that direction.

These laws generally expressed in basic mathematical equation which are either partial differential equation or integral equation.

2.3.1 Equations Which Governs CFD

With the application of law of mechanics we find out the governing equation for fluid flow problem.

Mass conservation equation is :

$$\frac{\partial\rho}{\partial t} + \nabla.(\rho V) = 0 \qquad \text{... (2.1)}$$

Momentum conservation equation:

$$\frac{\rho\partial V}{\partial t} + \rho(V.\nabla)V = -\nabla p + \rho.g + \nabla.\tau_{ij} \qquad \text{... (2.2)}$$

Where

V= velocity of fluid in m/s

τ_{ij}= time factor in sec

ρ = density of fluid in kg/m^3

Energy conservation equation along with above equation can’t be solved analytically for many engineering problems. But approximate solution can be computed by computer solution from these governing equations.

2.3.2 Discretization in CFD

The solution in CFD generally governed by Nervier-Stokes and sometimes Euler equation and these are partial differential equation. These partial derivatives are expressed as algebraic quotients and in CFD this algebraic quotient are expressed as flow field variable at discrete grid points. Discrete grid points are generally created by the following two methods.

i. **Finite volume method (FVM)**

FVM is popular in CFD as it has many advantages. As discretisation by FVM follow the three basic laws i.e. energy conservation, mass conservation and momentum conservation law. It doesn't require any coordinate system to be applied on irregular meshes. So FVM can be applied on meshes which are unstructured in three dimensions and some irregular polygons in two dimensional.

ii. **Finite element method (FEM)**

It is used in the case of structural mechanics problem and solid mechanics related problems. FEM is generally used to multidimensional complex irregular geometries. In this method domain in divided into small elements called as finite elements. An appropriate interpolation function is used, and approximation of function at nodes to get values is carried out. Then approximation is substituted into governing equations, which are then solved by computers.

iii. **Finite difference method (FDM)**

First step is to discretize the geometric domain to obtain a numerical simulation, so we have to define the numerical grid. In FDM the grid is structurally locally, i.e. each grid node may be considered the origin of a local coordinate system, whose axis coincides with the grid lines. According to this two grid line family belongs to the same family, say x_1 = constant, and x_2= constant, intersect only once. In 3D grid system, three lines will intersect at each node, none of these will intersect each other at any other node.

Each node can be identified separately by a set of indices, like (i , j) in two dimension and (i,j,k) in three dimension. The neighbour nodes can be obtained by increasing or decreasing one unit.

Mathematical model of differential equation are given by,

D (ϕ) = k

D is the differential equation,

ϕ is the parameter,

K is the source term,

$$(\partial\phi/\partial x)_{x_i} = \lim_{\Delta x \to 0} \frac{\phi(x_i+\Delta x)-\phi(x_i)}{\Delta x}$$

Differential can be approximated as:

$$\left(\frac{\partial\varphi}{\partial x}\right)_i \approx \frac{\varphi_{i+1}-\varphi_i}{x_{i+1}-x_i} \qquad \text{Forward difference} \qquad \text{…. (2.5)}$$

$$\left(\frac{\partial\varphi}{\partial x}\right)_i \approx \frac{\varphi_i-\varphi_{i-1}}{x_i-x_{i-1}} \qquad \text{Backward difference} \qquad \text{….. (2.6)}$$

$$\left(\frac{\partial\varphi}{\partial x}\right)_i \approx \frac{\varphi_{i+1}-\varphi_{i-1}}{x_{i+1}-x_{i-1}} \qquad \text{Central difference} \qquad \text{…. (2.7)}$$

Similarly for solving higher derivatives to convert them into algebraic system which are solved by computations we uses finite difference approximations. FDM generally uses structural grids.

2.3.3 Grid Generation

The division of flow field throughout into discrete points is called grid. The basic grid shape is rectangular type. But this is not compulsory some physical applications uses different type of grip for them we need a transformation for the derivative in the standard partial differential equation. There are different type of grids like structured grids, elliptical grids and adaptive grids as the application demands. After the formation of grid the mesh is created between them which can be easily visualised as mesh of finite volume cells. The meshes generated can be structured or unstructured type

2.3.4 Model Available In FLUENT

There are two types of flow problems like laminar or turbulent and steady or unsteady. Many types of models are present for steady as well as turbulence flow. Laminar flows have smoothly varying velocity fields in space and time in which individual laminar flow past one another without generating the currents. In laminar flows the viscosity is very large. These flows generally have low values of Reynolds no. Turbulent flows have large fluctuation in velocity and pressure in both space and time. These change in velocity and pressure leads to instabilities that grow until interactions are nonlinear and these fluctuations convert the flow into finer and finer whirls. The different turbulence models available in fluent:

1. K $-$ ε models
   - Standard K $-$ ε model
   - Renormalization group (RNG) K $-$ ε model
   - Realizable K $-$ ε model

2. $K - \omega$ models
    - Standard $K - \omega$ model
    - Shear stress transport (SST) $K - \omega$ model
3. Reynolds stress model (RSM)
4. Detached eddy simulation (DES) model
5. Large eddy simulation (LES) model
6. V2-f model
7. Spalart-Allmaras model

From the above models $K - \varepsilon$ model is widely used for turbulence modeling and it also gives good results, hence this model is used for flow analysis of francis turbine. But for analysis of cavitation Shear stress transport (SST) $K - \omega$ model is widely used.

CHAPTER 3

CFD ANALYSIS OF FRANCIS TURBINE

3.1 GENERAL

In this study, a Francis runner with casing, guide vanes and draft tube for large hydro range has been designed. A Francis turbine having rated head of 269m and discharge of 112.22 cumec is considered. The detail of design is discussed as below:

3.2 DESIGN OF TURBINE PARTS

The main characteristics are the data on which the design of runner is based. To calculate the sizing of turbine some adaptation mechanism is needed, for sizing of turbine we need three data parameters which are discharge available at the site, head available at the site and rotational speed of turbine which is defined by the generator rpm. So data which we needs is as given below. Site data of a hydro power plant is taken which is as follow in table 3.1:

Table 3.1: Site Data of power plant

Power 100%	P100% = 255MW
Power 120%	P120% =306MW
Nominal Head	H_n =269m
Maximum Head	H_{max} =300m
Nominal Discharge	Q_n =112.22 m^3/s
Speed of Turbine	N =214.29 rpm
Runner Diameter	D_2 =3570 mm
Number of runner blades	Z =13

Number of wicket gates	Z =24
Height of gate mechanism	B_o = 669mm
Sense of rotation	Counter clockwise

POWER HOUSE

- GROSS HEAD - 293 M
- NET HEAD - 269 M
- GENERATING UNITS - 4X250 MW
- TURBINE - FRANCIS
- SPEED - 214.3 RPM

3.3 CALCULATE DIMENSIONS FOR THIS FRANCIS TURBINE

All the calculation are done according to the instruction given in IS_12800 (1) [27]

3.3.1. Specific Speed and Rotational Speed

The hydro turbines can be classified by their specific speed, it is a dimensionless parameter and characterizes by hydraulic properties of a turbine in terms of speed, power and head. It is based on similitude rules.

All the turbines having the similarities like (dynamic, geometric and kinematic) between prototype and model, working under same k_u and φ, will have the same specific speed , no matter what power they develop under what head and what size they have. So both model as well as turbine will have same specific speed. Equation 3.1 of specific speed is as given below:

Specific Speed of Turbine (N_s) = $\frac{N\sqrt{P*1.358}}{H^{5/4}}$ …(3.1)

H= 269m, N=214.29 rpm, P= 255 MW

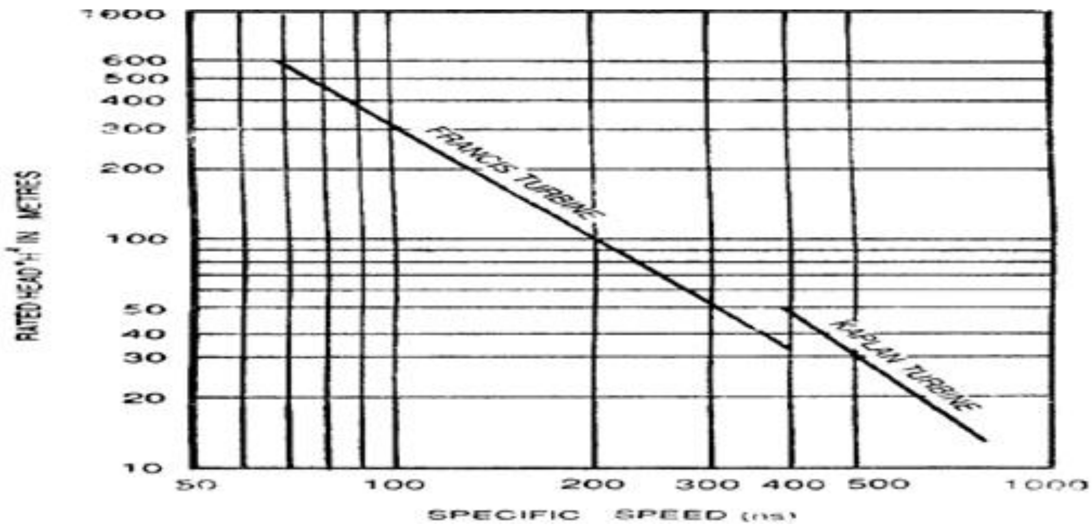

Fig. 3.1 Relationship between Specific Speed And Rated Head [27]

So N_s =115.75 rpm, verify it from graph given below in Fig. 3.1

3.3.2. Turbine Setting

In reaction turbines, the distance between turbine level and minimum tail race level decides cavitation will happen or not in turbine. The suction height above the minimum tail water level can be determined by the given formula by Equation 3.2:

$$H_s = H_b - \sigma H - H_v \qquad \ldots(3.2)$$

H_s= Suction head in metres

$H_b =$ Atmospheric or barometric pressure in metres of water column

H_v = Vapour pressure; and

In case we do not know about $H_b - H_v$, then it can be determined From Fig. 3.2 for a specific altitude above mean sea level and for a known temperature which is generally taken as 30°C.

σ = Thomas's cavitation coefficient, which can be which can be obtained from Fig. 3.3

So height of water column =7.65 m at 1810m altitude and at 30 degree Celsius.

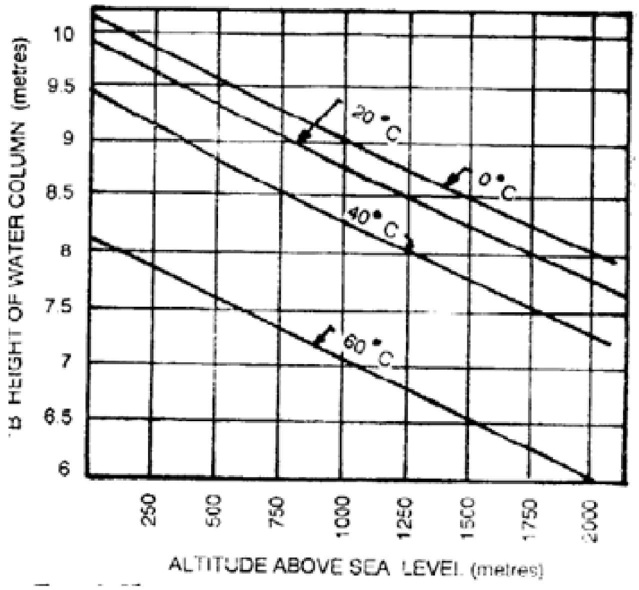

Fig. 3.2 Height of Barometer Water Column [27]

As specific speed is 115 rpm so the $\sigma = 0.055$ from Fig.3.3

So calculated H_s is -8.58 m and as H (net head) changes then will H_s also change.

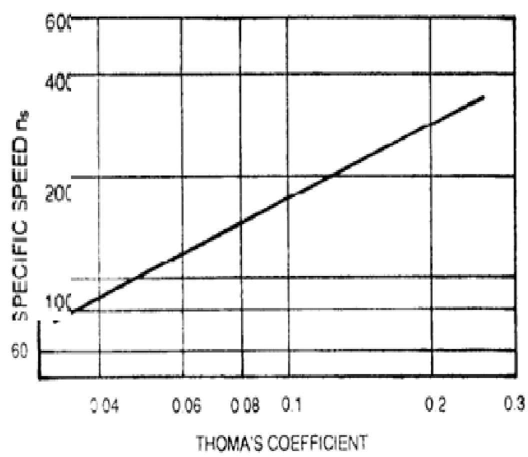

Fig. 3.3 THOMA'S Cavitation at Different Specific Speed for Francis Turbine

3.3.3 Runner diameter (D_3)

Diameter of runner has been calculated from the formula as given under

$K_u = (\pi D_3 N_s)/\sqrt{(2gH)}$

D_3 = outer diameter of runner

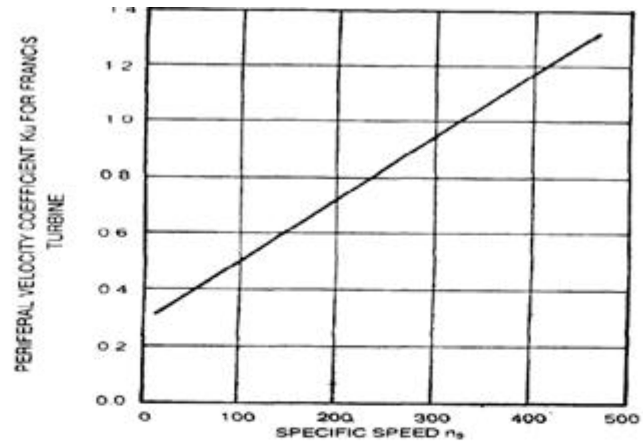

Fig. 3.4 Relationship between specific speeed $N_s$and peripherial velocityK_u cofficient for francis turbine [27]

K_u from above Fig.3.4 is about 0.51 at Ns= 115 rpm and H= 269 m

So D_3= 4.74 m

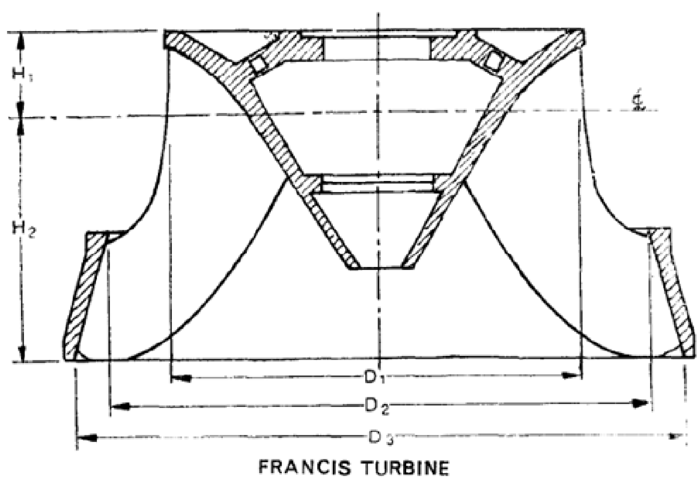

Fig. 3.5 Typical shapes of reaction turbine runners [27]

These following dimensions are shown in Fig. 3.5

$H_2 = 0.35D_3$ = 1.66m

$H_1 = 0.1D_3$ = 0.474m

$D_2 = 1.2D_3$ = 5.688m

$D_1 = D_3$ = 4.74 m

These dimensions will also used for generating blade profile in ANSYS bladegen.

3.3.4 Spiral Casing

An involute casing is provided around the runner and have guide vanes in between the two. the dimension of the casing is as given below and shown in Fig. 3.6.

Above 30 m gross head we use mettalic spiral casing.

$A = 1.02D_3$ = 4.834m

$B = 1.58D_3$ = 7.489m

$C = 1.78D_3$ = 8.4372m

$D = 1.9D_3$ = 9.006m

E= $1.5D_3$ = 7.11m

$F = 2.15D_3$ = 10.19m

$G = 1.75D_3$ = 8.295m

$H = 1.55D_3$ =7.35m

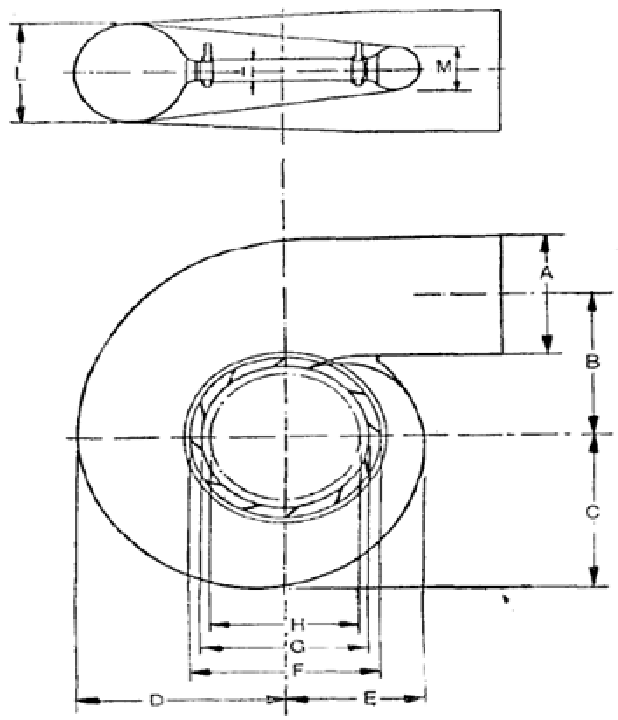

Fig 3.6 Major dimensions of the spiral casing [27]

I$= 0.17D_3$ = 0.81m

$L = 0.95D_3$ = 4.5m

$M = 0.6D_3$ = 2.85m

3.3.5 Draft tube calculations from IS 5496 [28]

We are using elbow type draft tube because it is efficient where excavation is difficult to do.

Draft tube dimension are depend upon the specific speed, size and spacing of the unit and and all of these depends upon diameter of runner.

The design should be such that exit loss and draft tube loss should be minimum. The retardation of water should be gradual and so that change in flow should be gradual with smooth surfaces.

Preliminary design of draft tubes can be derived by referring to either of the Fig. 3.7 and 3.8 by taking the values of ***H, L*** and ***B*** given in IS code 5496 graphs

The angle of inlet cone shod be between 6 to 10 degree. A bed slope should be 1:10 (H:V). However, this slope can be maximum up to 1:6.

The height of draft tube should be in between 0.94 D to 1.32 D (D being the inlet diameter of the runner) according to specific speed of the turbine, the lower specific speeds has corresponding lower values.

Depth of the draft tube is measured from the Centre line of the guide apparatus.

For Francis turbine a depth of 2.5 to 3.3 D is used.

The draft tube length measured from turbine axis is 4D to 5 D.

The draft tube exit width (excluding the pier) is normally recommended as 2.6 to 3.3 D. In exceptional cases it may be even more or lower than 3.9D.

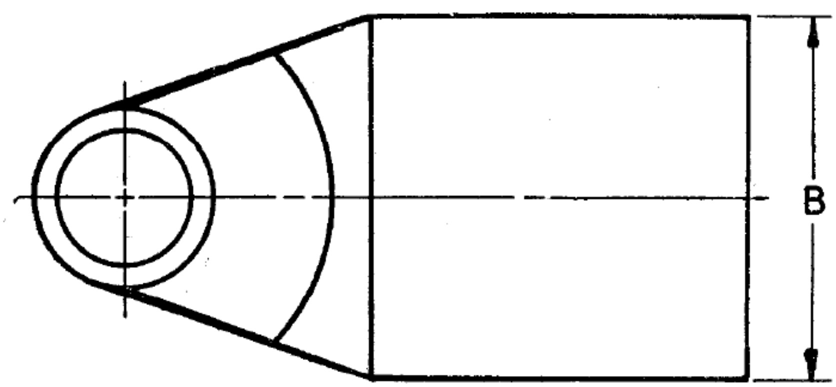

Fig 3.7 Top view of an elbow type draft tube [27]

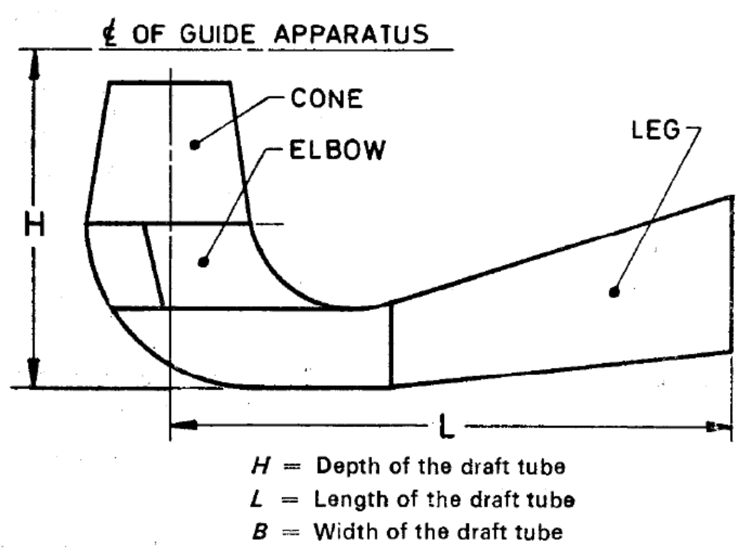

Fig 3.8 Front view of an elbow type draft tube [27]

As D = 4.74m

Following dimensions are shown in Fig. 6.7 & 6.8

Depth of draft tube H= 2.5 to 3.3 D

Width of draft tube B= 2.6 to 3.3D

Length of draft tube L=4 to 5D

So, H = 11.85m to 15.46m

B = 12.32m to 15.64m

L = 18.96m to 23.7m

3.3.6 Guide vane design [29]

Net head on turbine H=269m

Discharge through turbine Q =112.22 cumec

Nominal Speed of turbine N= 214.29rpm

$$D_o = D_1(0.29\Omega + 1.07)\varpi\omega\Omega$$

$$\Omega = \omega'\sqrt{Q'}$$

$$\omega = (2\pi N)/60 = 22.44 \text{ rad/s}$$

$$\omega' = \frac{\omega}{\sqrt{2gH}}$$

$$Q' = Q/\sqrt{2gH}$$

Put both values in equation of Ω

So $\Omega = 0.122$

Put it in equation of D_o

So $\frac{D_o}{D_1}$ =1.105

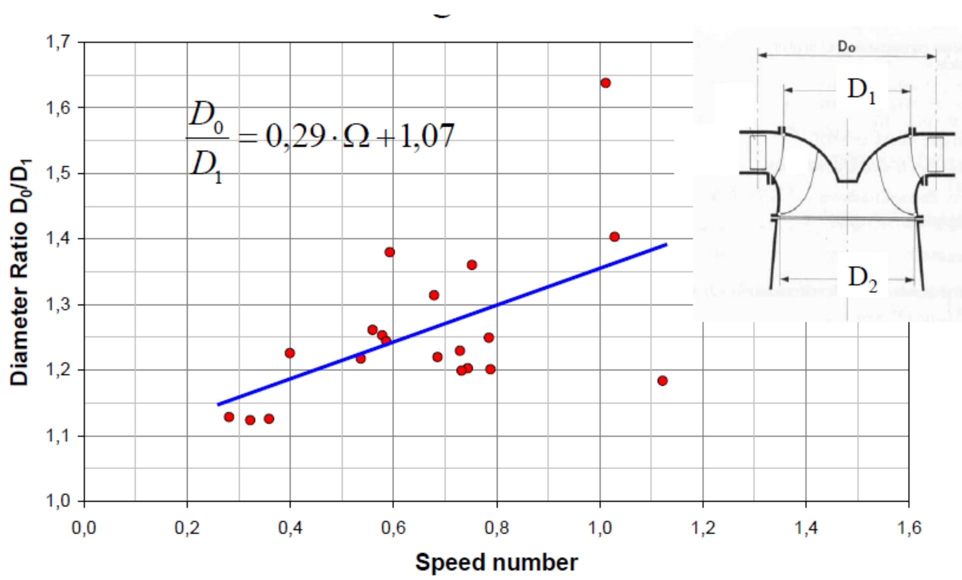

Fig 3.9 Graph between diameter ratio and speed no. [29]

So, from the graph in Fig 3.9 speed no. = 0.22

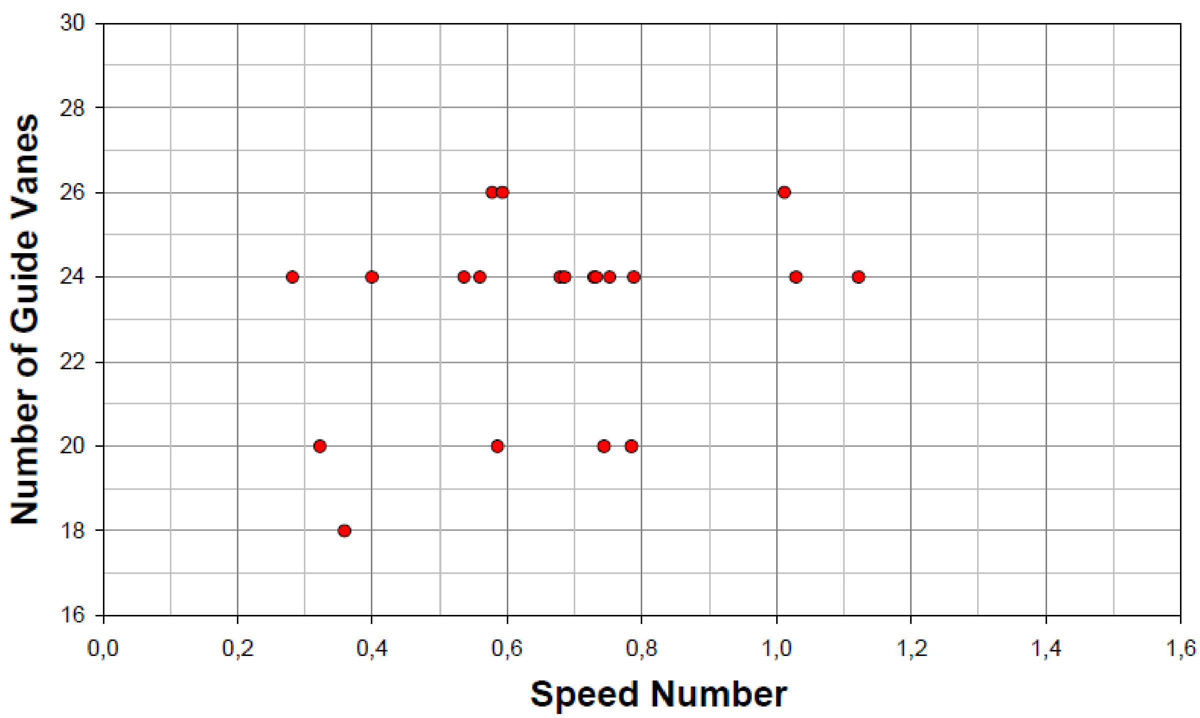

Fig 3.10 Graph between no. of guide vanes and speed no. [29]

So no. of guide vane from the graph should be =24 from graph in Fig.3.10.

And the ratio of no. of guide vanes to runner vanes should not be an integer

- Guide vane maximum angle α_o at full load = 4(-4Ω^2+13 Ω+1)
  Maximum angle at full load is 21.6^0

3.3.7 Blade design

For the blade many factor affect the performance and play a significant role, the leading edge is thicker than trailing edge for streamlined flow. The bade should be as thin as possible to improve the cavitation characteristics it means it should be thicker near the hub, the thickness of each blade is 44mm.

3.4 CFD ANALYSIS

In this section Francis turbine is modeled according to the sizing given earlier. AUTODESK INVENTOR has been used for modeling of turbine parts and 3D modeling of turbine and converted from .iam format to igs. Format which is used in design modeler of ANSYS-15 software. The .igs file imported into the geometry on the workbench tool. The geometry material in geometry section has been defined and fluid domain inside the material through which fluid will flow is generated in the design modeler and sent file into the mesh generator in which meshing of the fluid domain is generated.

The overall efficiency of Francis turbine has been determined based on fundamental equation. The various parameters used in equation depend on the type of boundary equation used for numerical simulation. Simulation under the present study has been done by mass flow inlet and pressure outlet set of boundary conditions.

3.5 PARAMETERS CONSIDERED FOR PRESENT STUDY

A 255MW Francis turbine with involute casing under designed head of 269 m and 112.22 cumec discharge has been considered in the present study. The other parameter of the Francis turbine has been computed by using design discussed in this chapter. The salient feature of Francis turbine considered as given in Table 3.1

Table 3.2: Parameters for present study

S.no.	Parameters	Values
1	Rated head	269m
2	Rated discharge	112.22 m^3/s
3	Rated turbine output	255MW
4	Rated speed	214.29 rpm
5	No. of runner blades	13
6	Diameter of runner	4.740m
7	No. of guide vanes	24

3.6 METHODOLOGY

The present study has been followed the certain methodology which provide optimum approach to finish the work. For CFD analysis of Francis turbine, Methodology is as shown below in Fig. 3.11:

First step: To generate CAD model which has been done in AUTODESK INVENTOR PROFESSIONAL, this stage is preprocessor stage for simulation.

Second step: is to assemble the part and generate the fluid domain.

Third step: is to generate the fluid domain out of the assembly of the parts.

Fourth step: is to generate the meshing of different turbine components with the help of ANSYS software.

Fifth step: to setup the boundary condition for the simulation or numerical computation with the help of FLUENT software. This stage is called solver stage.

Final step: is the analysis of result

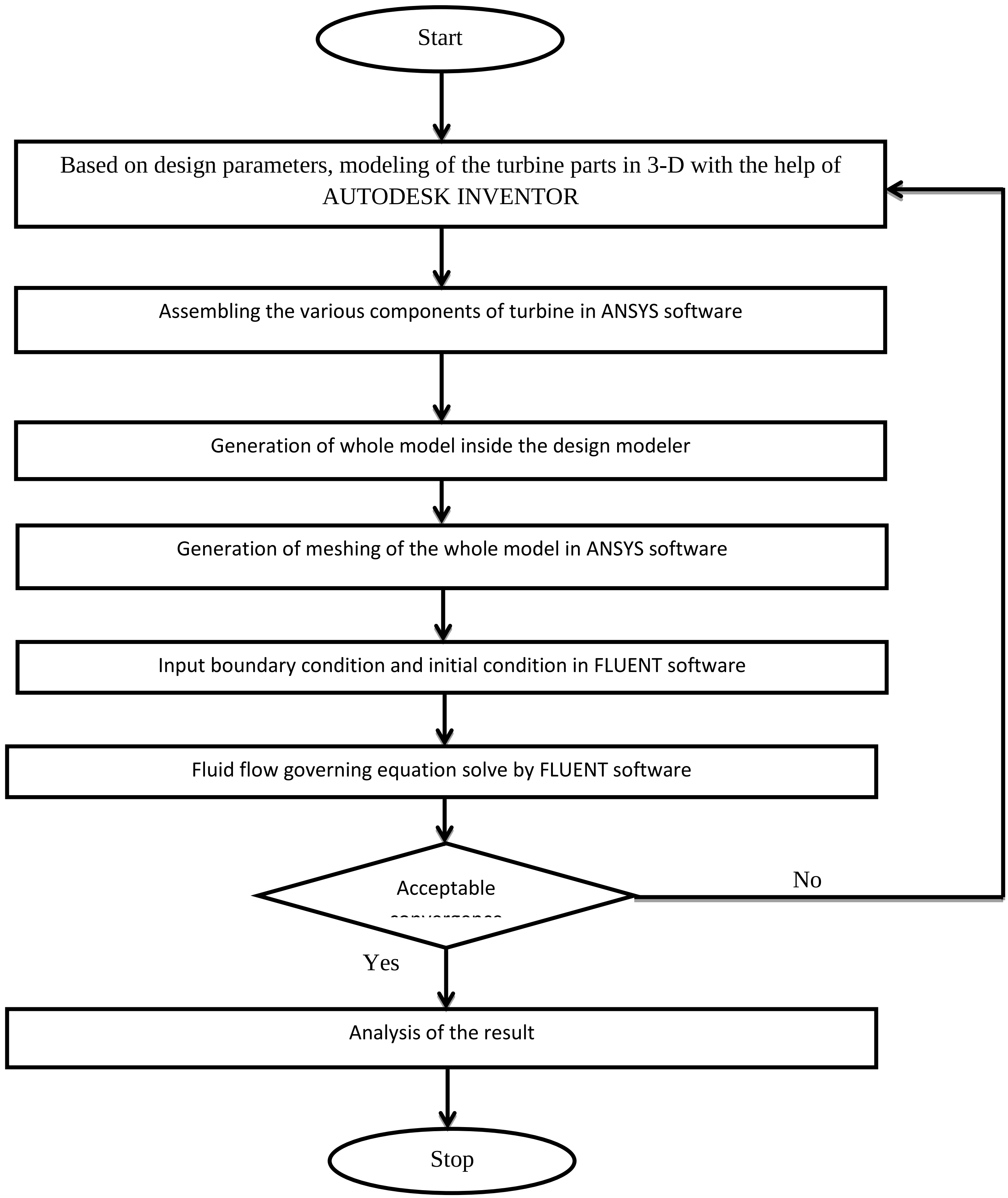

Fig. 3.11: Flow chart for methodology

.

3.7 MODELING OF TURBINE COMPONENTS

Three dimensional modeling of Francis turbine is the first step of CFD analysis. For the present work we have used AUTODESK INVENTOR for 3D modeling of turbine. It is most user friendly software.

3.7.1 Tools for creating part on AUTODESK INVENTOR

There are many tools for creating the 3D model of turbine. Some of the basic tools which are useful are described below:

I. Open the INVENTOR part modelling

Firstly open the INVENTOR software which provide INVENTOR work screen and open new file which gives window in which select the part modelling. Set the parameters on which the model have been prepare. In the present study, .ipt file is used for part solid.

II. Work screen of INVENTOR

The work screen opened after selecting the part modelling. This is a platform on which model has been prepared. In work screen various tools such as sketch, view management, edit toolbar, constraint toolbar, and 3-D feature creation toolbar are there which is used for 3D model creation of turbine.

III. Sketching toolbar

To prepare any 3D model first work to prepare the 2D sketch, this is created by sketching tool. At first select the 2D plane on which we have to sketch the 2D design, for this click on the sketch tool icon in the work screen sketch then window will come. In this present study line, rectangle, circle, arc and circle etc. tools have been used.

IV. 3D part generation toolbar

Then the 2D sketch is converted into 3D model with the help of 3D generation tool. Extrude, revolve, loft sweep, bend, torus, chamfer tool has been used for generation of 3D models of turbine.

V. **Edit toolbar**

Then use the edit toolbar to make changes in the 3D design and modifying Exiting features. Used for removing existing part in the model of the turbine and also used for developing symmetrical bodies or pattern on the body, like to make all blades out of one blade of runner. Extrude cut and pattern command has been used.

3.7.2 Creating 3D Model of Blade And Runner

I. Start ANSYS-15 software and open workbench in it.

II. Open the bladegen module and give dimension of blades according to Z and R parameters demanded by ANSYS in a meridonial plane also provide wrap angle, no. of blades. and thickness as shown in Fig. 3.12 below.

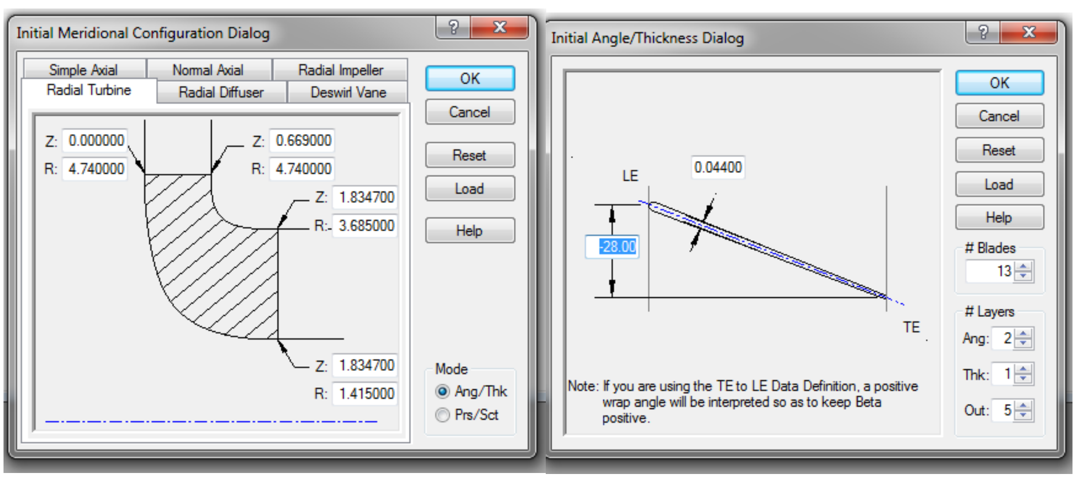

Fig. 3.12 Input of blade outer dimensions, thickness and wrap angle of blade

III. Then adjust blade angles at leading edge and trailing edge and then click OK to Generate the blade profile as shown in Fig. 3.13.

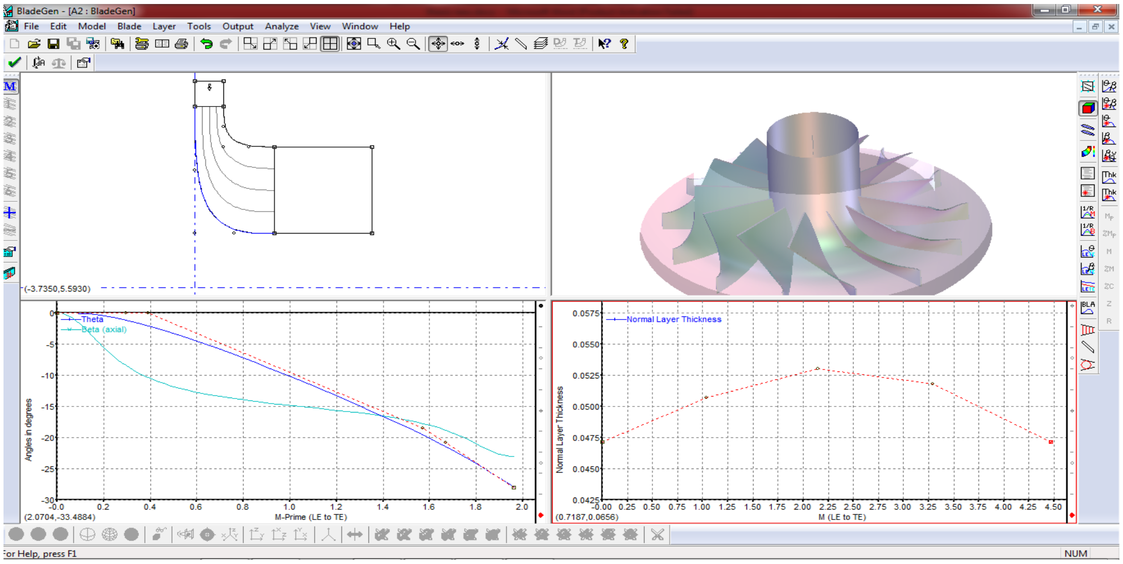

Fig. 3.13 Meridonial view of the blade and blade angles

IV. After generation of blade profile pass it to the design modeler then it a we have to fill some information as given below in Table 3.2 :

Table 3.3: The parameter for importation of blade profile in design modeler

Details of ImportBGD1	
ImportBGD	ImportBGD1
Unit Preference	Meter
Create Hub	Yes
Hub Offset	0.05
Create Blades	1
Merge Blade Topology	Yes
Blade Loft Direction	Streamwise
Create Shroud Clearance	No
Create Fluid Zone	No
Blade Extension (%)	2

V. Then click on generate button to make the blade profile. Convert into .igs file and save the file and Fig. 3.14 shows blade profile below:

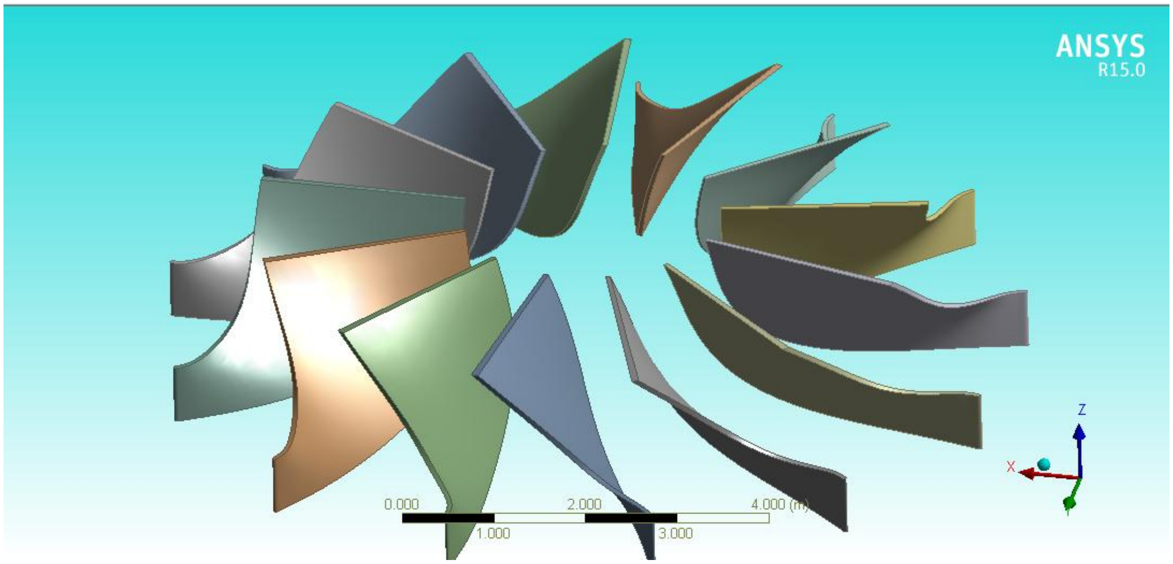

Fig. 3.14 The blade profile in design modeler

VI. Open this file in inventor software and generate the geometry of runner as shown below in Fig.3.15.

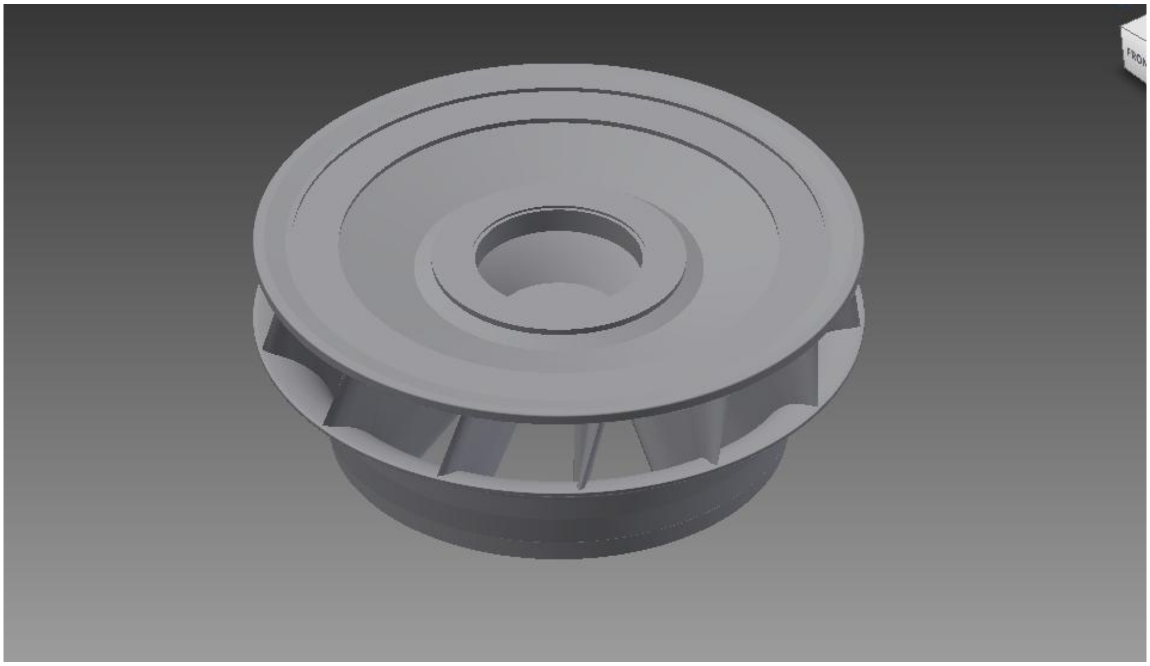

Fig. 3.15 3-D Model of turbine runner

3.7.3 Creating 3D Model For Guide Vanes.

I. Start inventor and select the part modelling button.

II. Select the sketcher and take the plane if runner as reference to make the 2D sketch for guide vanes.

III. Make that sketch and click on OK button

IV. Extrude that guide vane sketch up to the thickness of turbine inlet and then pattern the guide vane and make 24 guide vanes at a regular interval around the runner as shown in Fig. 3.16.

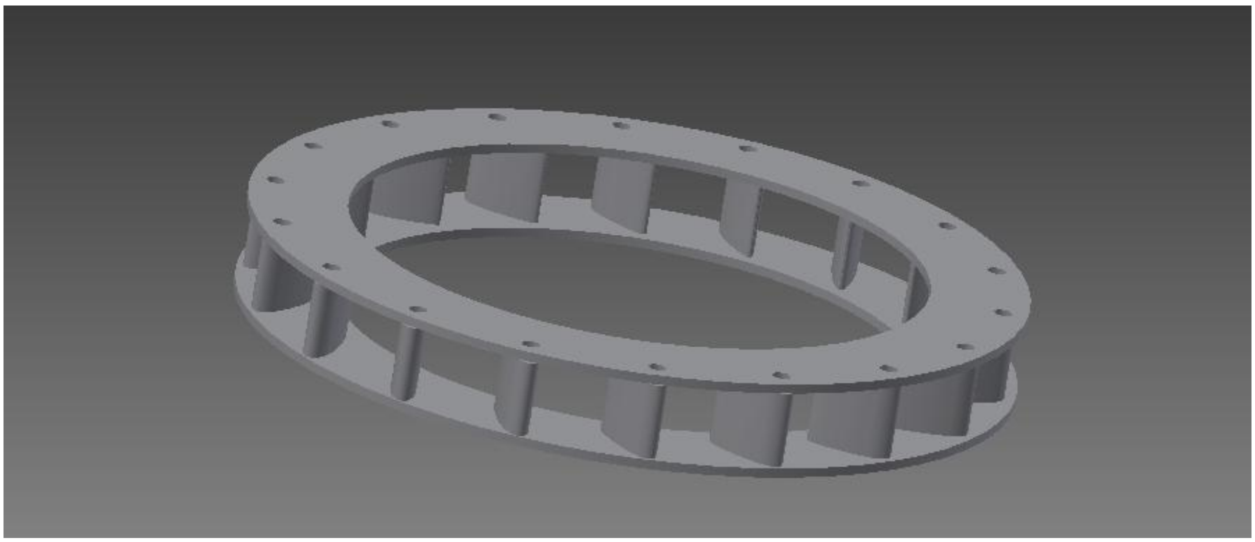

Fig. 3.16 3-D Model of guide vanes

3.7.4 Creating the 3D Model for Casing

I. Start inventor and select part modelling.

II. Select sketch tool and select the plane runner as reference plane.

III. Make sketch and click OK button.

IV. Use loft command to construct the casing. The model of casing is as shown in Fig. 3.17 given below.

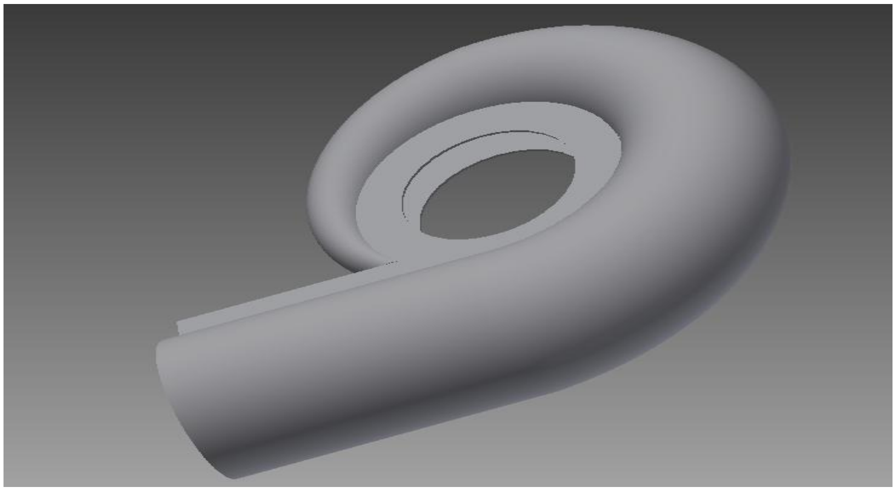

Fig. 3.17 3-D model of casing of turbine

3.7.5 Creating the 3D Model of Draft Tube

I. Start inventor and select part modelling.

II. Select sketch tool and select the bottom of runner for reference plane.

III. With the help of revolve and loft command make draft tube.

IV. Then use shell command to make the draft tube hollow as shown in Fig. 3.18

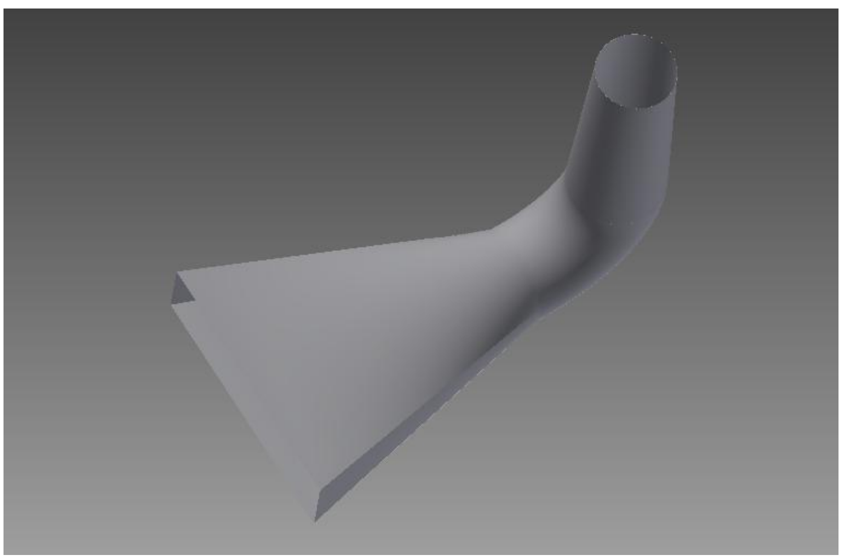

Fig. 3.18 3-D model of draft tube

3.8 CREATION OF FLUID DOMAIN

I. Export file from inventor to design modeller in the form of .igs file.

II. Use fill command from the tool button and click on fill by caps.

III. Create fluid domain by click on OK as shown in Fig.3.19 below.

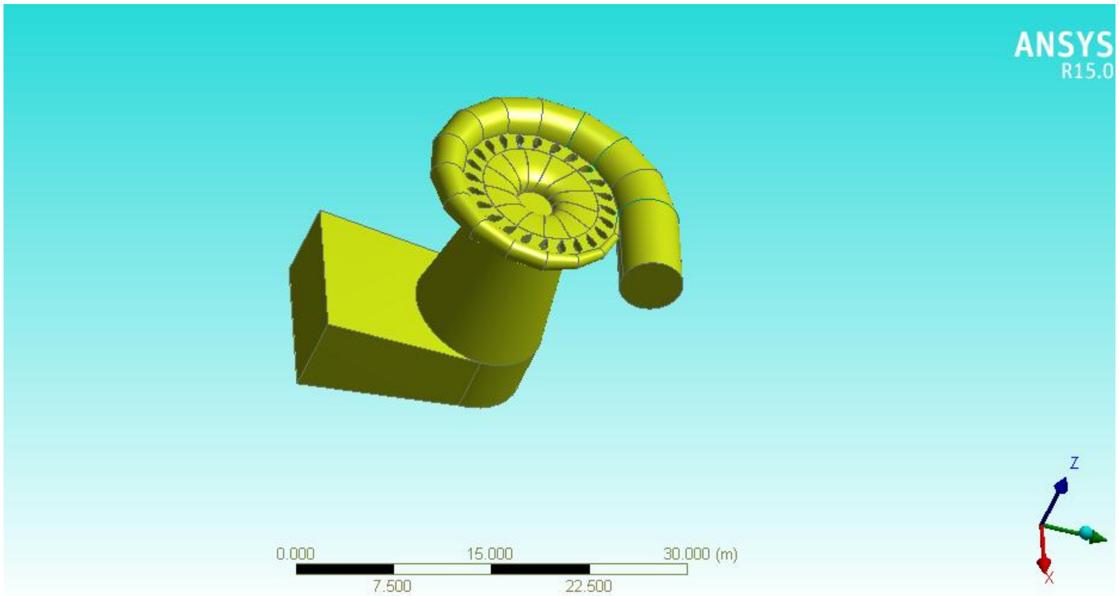

Fig. 3.19 The fluid domain of whole model

3.9 MESHING OF THE FLUID DOMAIN MODEL OF TURBINE

Meshing or grid generation of 3D model is a necessary task and it is takes too much time. If the geometry is very complex then unstructured grid mesh is used which consist of tetrahedral and triangular element.

Now for generation of mesh ANSYS mesh module is used for CFD FLUENT. FLUENT is a solver used for simulation and producing fluid flow solution.

3.9.1 Steps for mesh generation in 3D model

I. Open the window of ANSYS workbench on which we can information of the whole project.

II. Send file of fluid domain from design modeller to mesh module which is in .agdb format.

III. Set the insertion method as tetrahedral patch conforming and the parameters used is as given below and put smoothing as low as shown in Table 3.3.

Table 3.4: Parameter for sizing used to generate mesh

Details of "Mesh"	
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Fun...	On: Proximity and Curvature
Relevance Center	Medium
Initial Size Seed	Active Assembly
Smoothing	Low
Transition	Slow
Span Angle Center	Fine
Curvature Normal A...	Default (18.0 °)
Num Cells Across Gap	Default (3)
Min Size	40.0 mm
Proximity Min Size	40.0 mm
Max Face Size	500.0 mm
Max Size	500.0 mm
Growth Rate	Default (1.20)
Minimum Edge Length	3.8356e-002 mm

IV. Click on generates mesh and also define name of surfaces by named selection as inlet, outlet, int_ij (for interface between i and j) as shown in Fig.3.20.

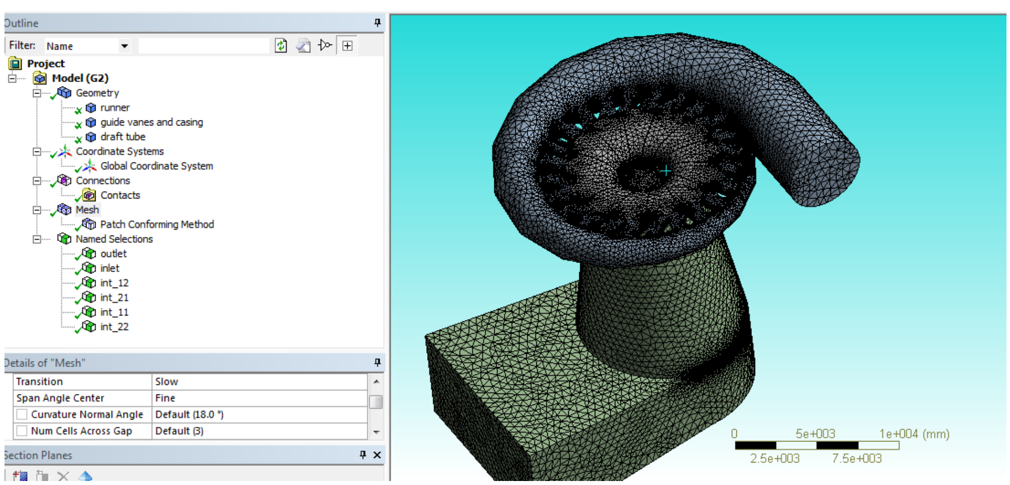

Fig.3.20 Mesh model of the whole fluid domain

3.9.2 Meshed 3D Model for Turbine Parts

I. Runner

The meshed model of runner is shown below as in Fig. 3.21. Total no. of nodes and element are 219940 and 1268261 respectively generating after the meshing of the runner.

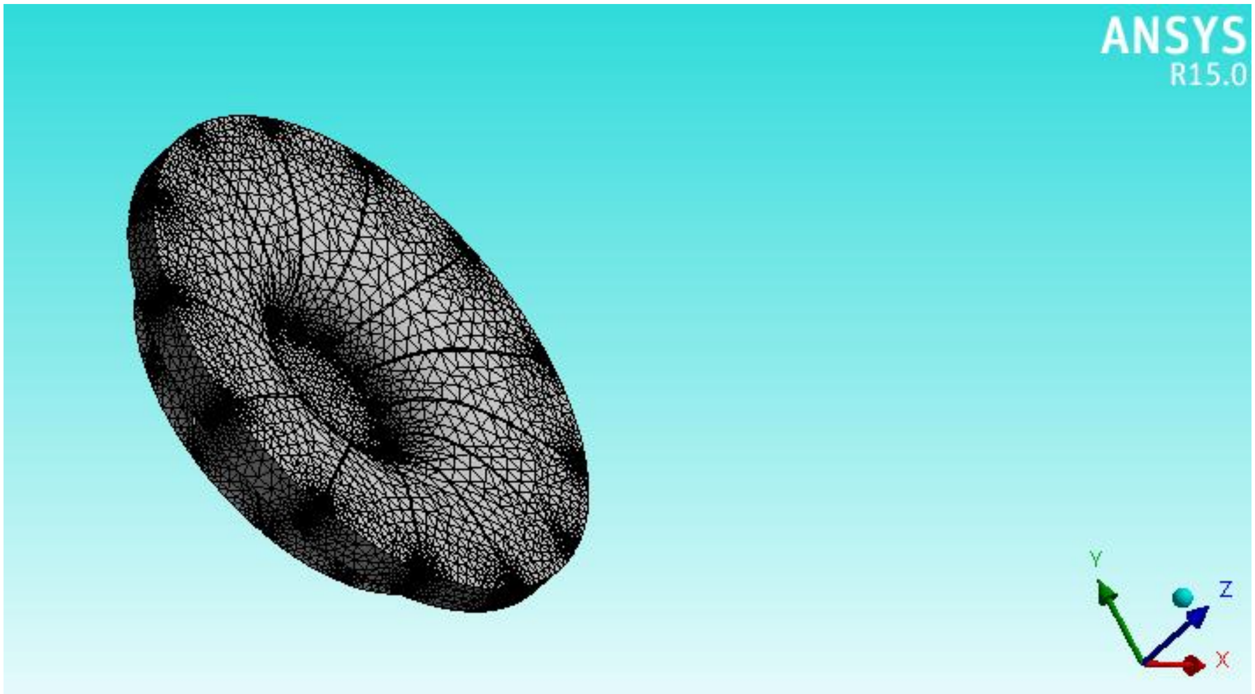

Fig. 3.21 Mesh model of turbine runner

II. Draft Tube

The meshed model of Draft Tube is shown below as in Fig 3.22. Total no. of nodes and element are 251146 and 1268261 respectively generating after the meshing of the Draft Tube.

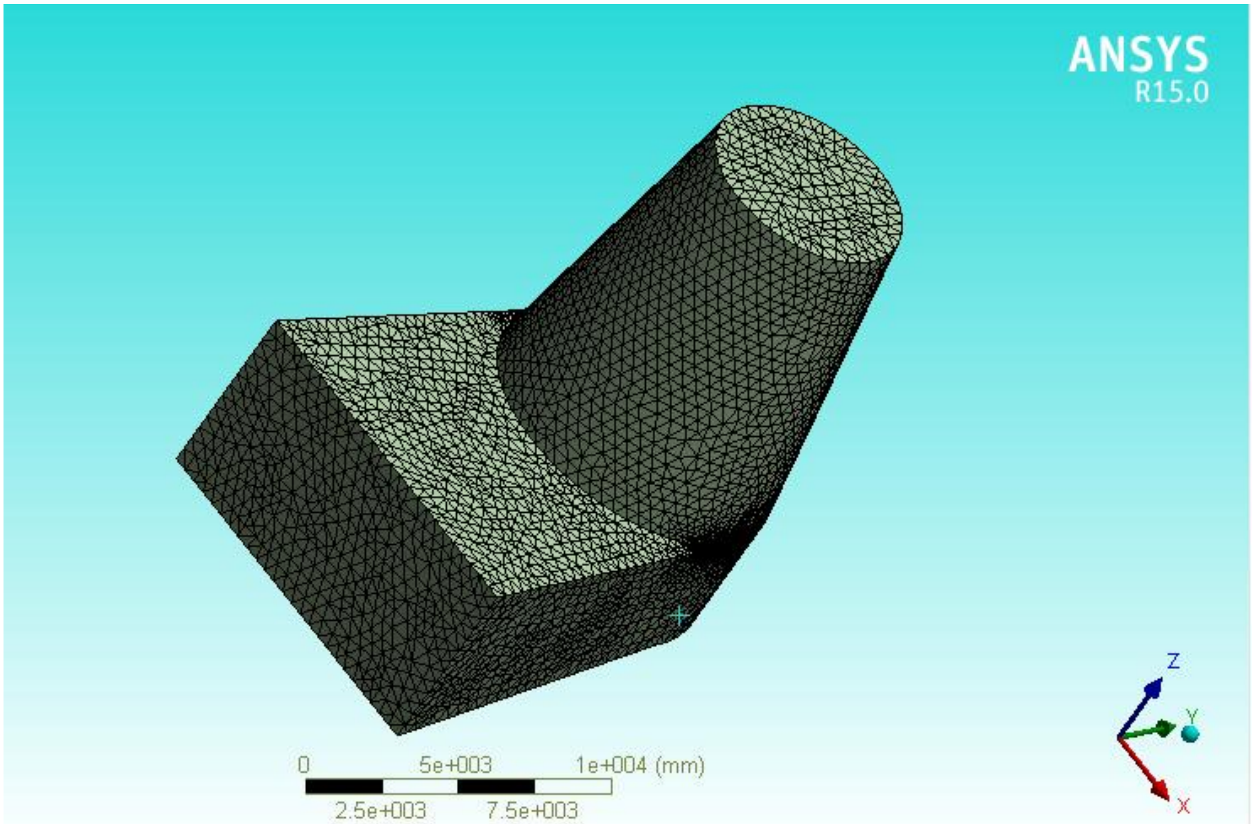

Fig. 3.22 Mesh model of draft tube

III. Guide Vanes And Casing

The meshed model of Guide vanes and casing is shown below as in Fig.3.23. Total no. of nodes and element are 118007 and 596016 respectively generating after the meshing of the Guide vanes and casing.

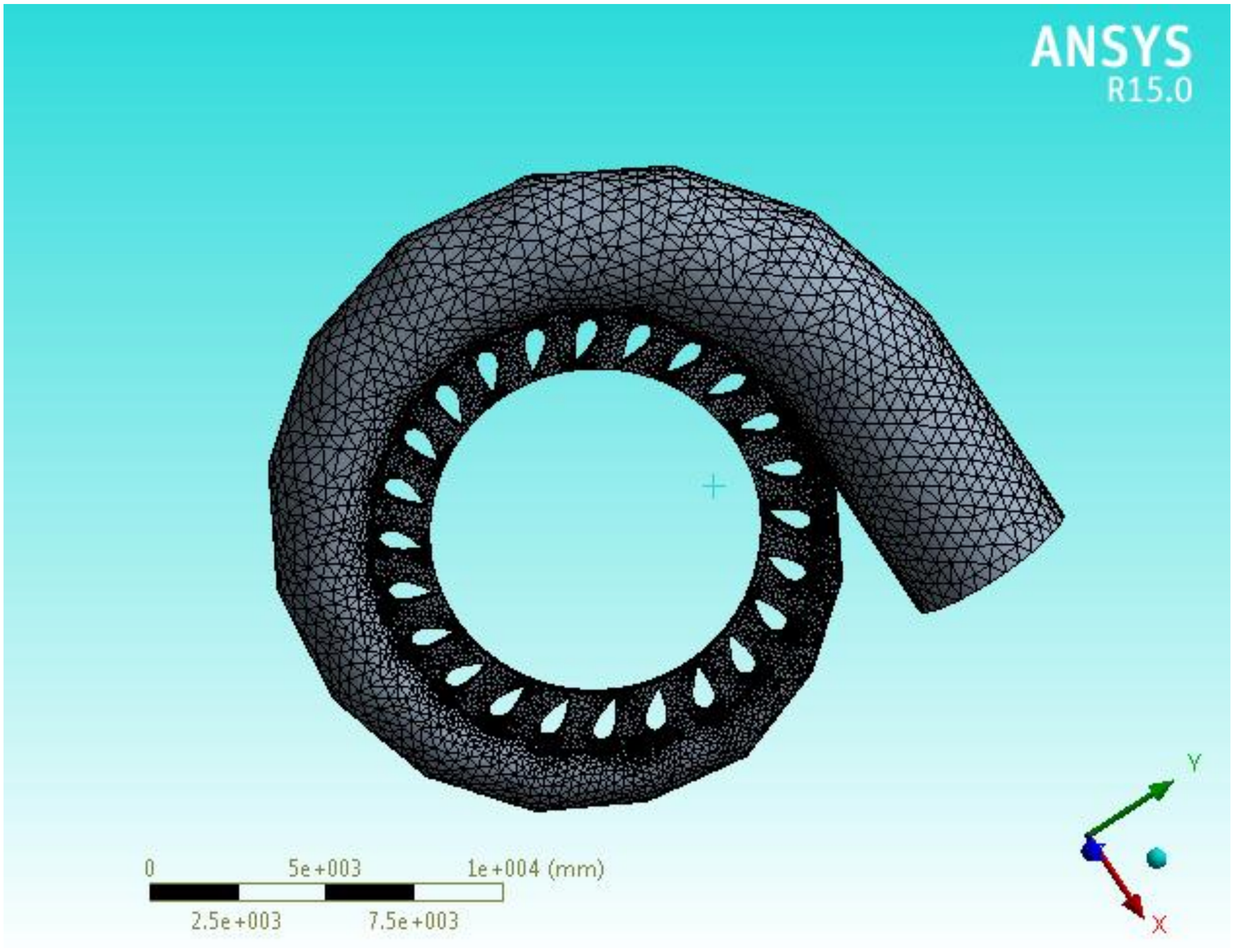

Fig. 3.23 Mesh model of turbine casing

Modeling and meshing of turbine component is the pre-processing part of CFD simulation. After this solver stage comes in picture in which software has to solve the fluid flow governing equation with the help of FLUENT module in ANSYS software.

3.10 FLUENT SOLVER SETUP

For solving the problem or doing the simulation of the fluid flow, we have to setup the condition for FLUENT like, material of flow, boundary condition, type of phases, mesh interfaces, monitor, cell zone condition, solution initialization, solution controls, solution methods, type of model for the turbine or hydraulic machine under consideration.

3.10.1 When Fluid is Pure Water

Import mesh to the fluent setup to proceed further as shown in Fig. 3.24 below.

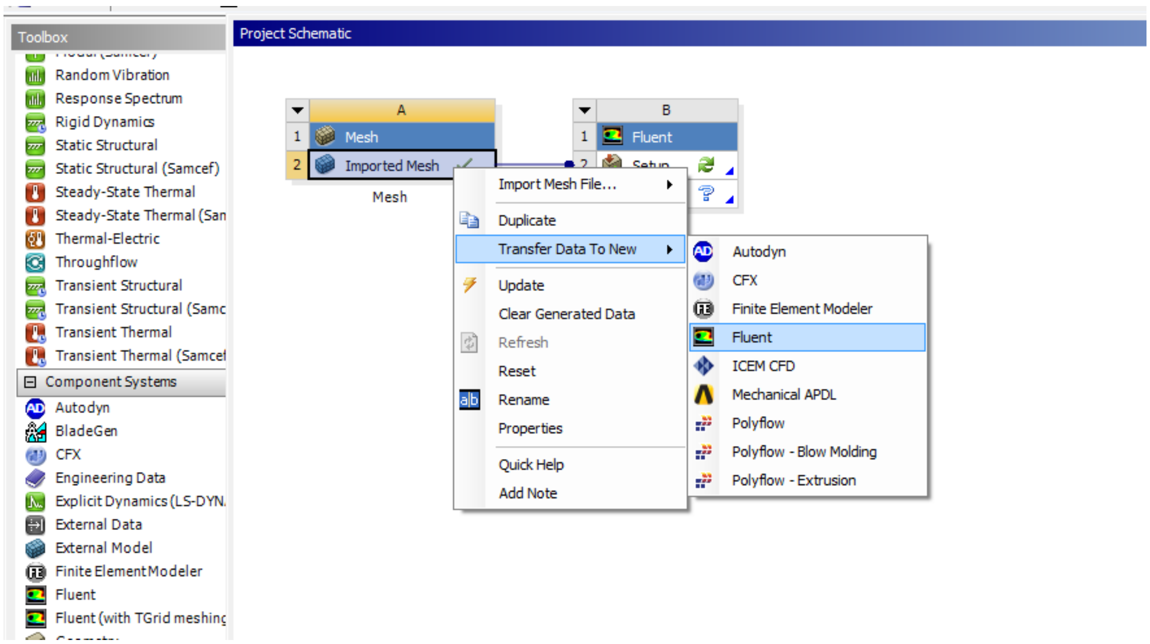

Fig. 3.24 Data transfer from mesh to fluent

I. Now open FLUENT module by double clicking the setup under the fluent dialogue box as shown in Fig.3.25. Accept the default setting but click on double precision.

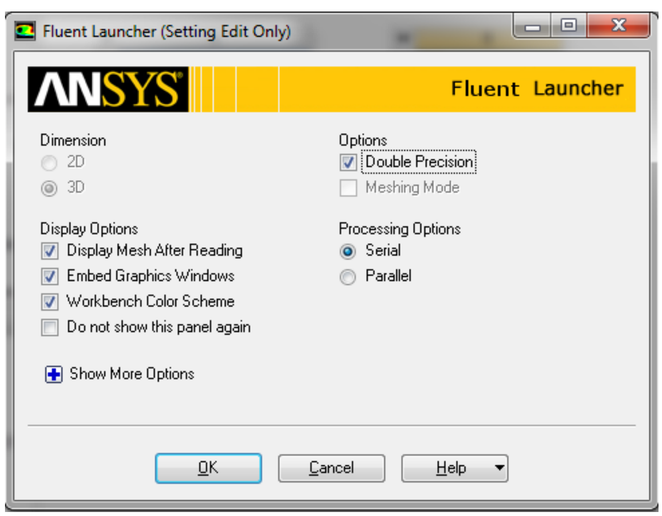

Fig. 3.25 Dialog box on opening of fluent

II. General setting: the main FLUENT window will appear, click on the general tab in problem setup column. Then at right side check and scale the mesh grid by clicking the respective tab. Fig.3.26 shown below will make it clear with the problem setup window.

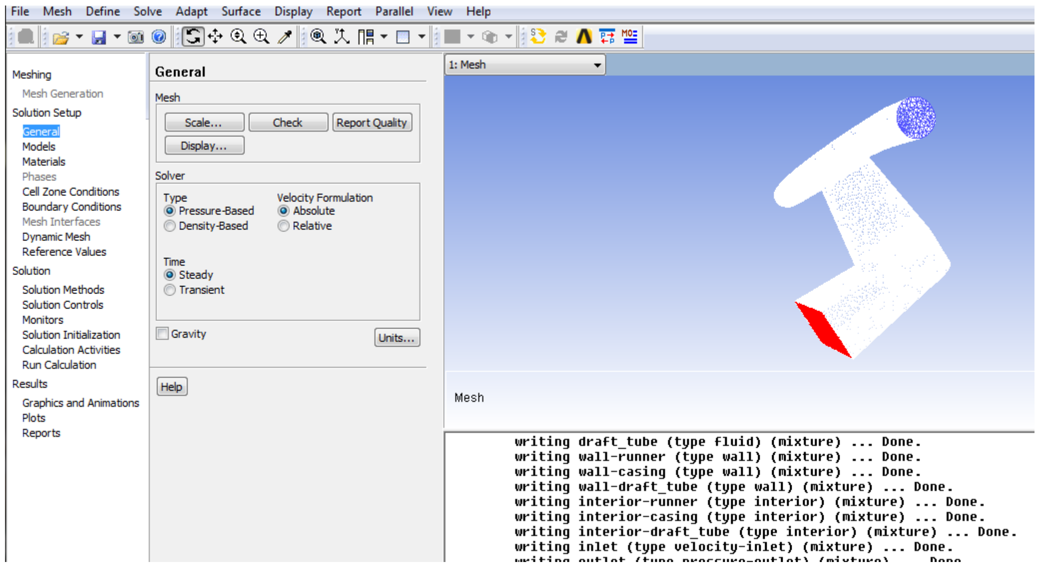

Fig. 3.26 General tab of fluent

III. Setup of model: Choose one the model from viscous, energy, multiphase etc. for the present work we are opting viscous model ($K - \varepsilon$,Realizable model, standard wall function) as shown in Fig. 3.27.

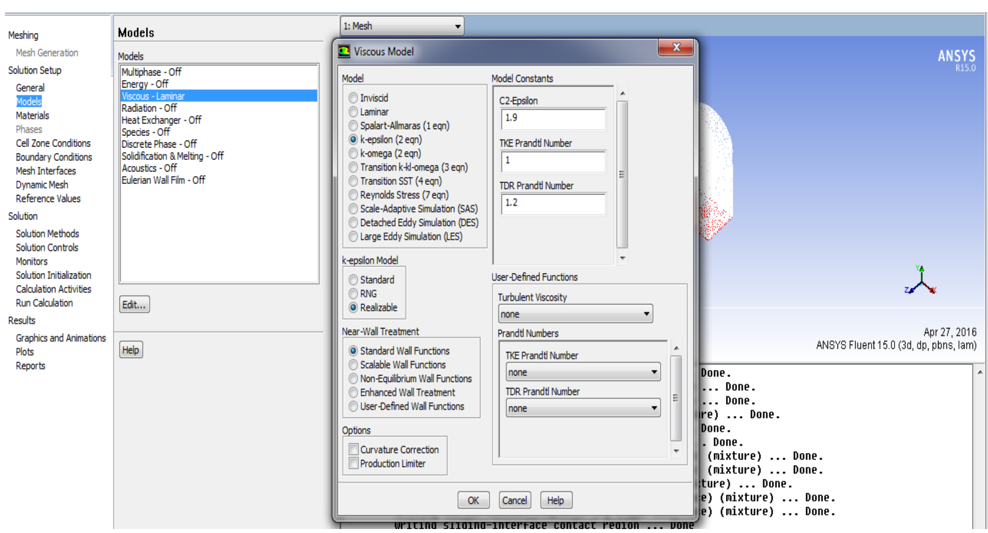

Fig. 3.27 Turbulence model used for pure flow

IV. Setup of material: Choose which type of solid and liquid we have to choose then define its properties like density and viscosity as shown in Fig.3.28 below we choose water- liquid as a material for fluid.

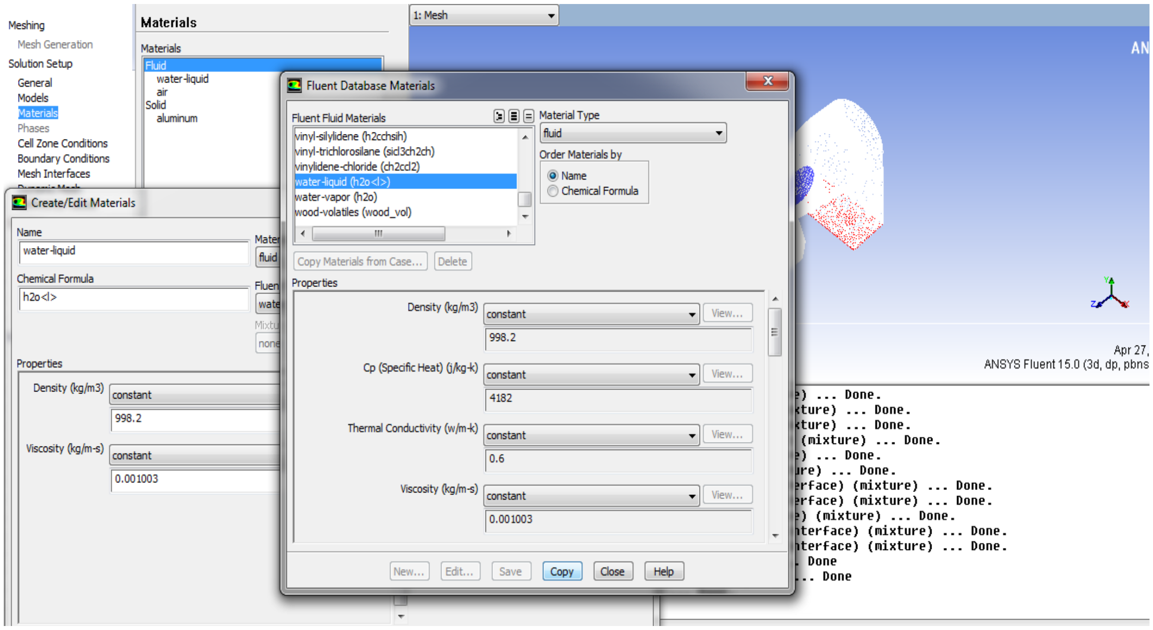

Fig. 3.28 Material included for fluid

V. Cell zone and operating conditions: Cell zone condition is defined to set up whether the frame is moving or stationary. In the present study. As in this study our runner is rotating at 214.29 rpm but all other are stationary so we have to fill this information as in Fig.3.29 below

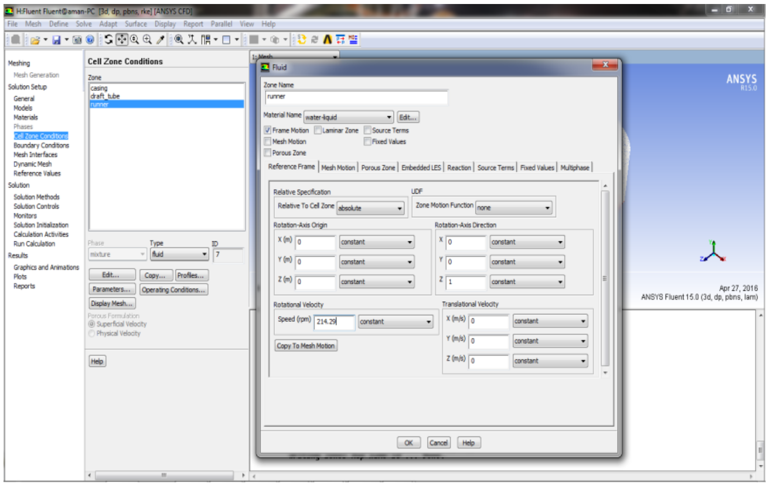

Fig.3.29 Cell zone conditions of different parts of turbines

VI. Boundary conditions: For the present study we have to put inlet boundary condition as mass flow rate as shown in Fig.3.30, outlet boundary condition as pressure outlet as shown in Fig.3.31 and wall of runner as rotating .

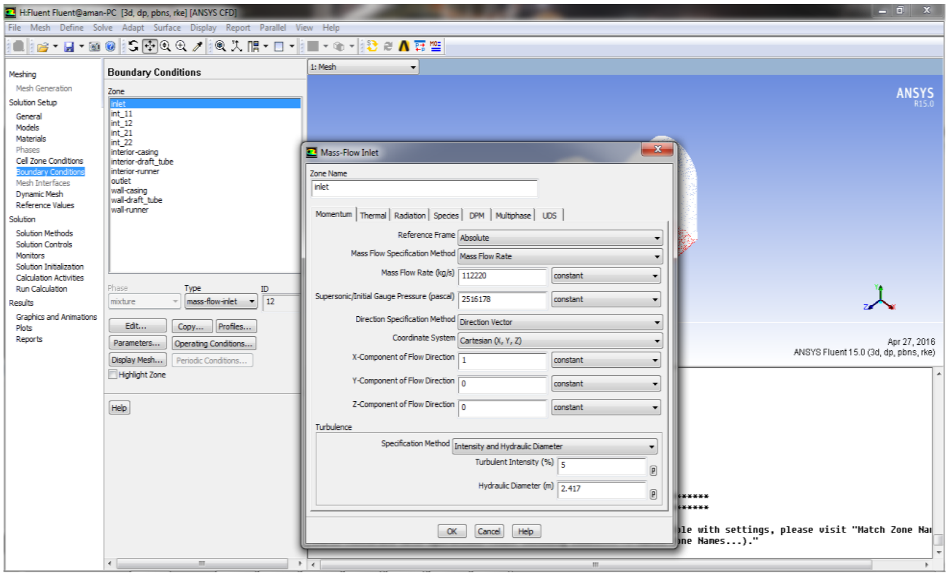

Fig.3.30 Boundary condition at inlet

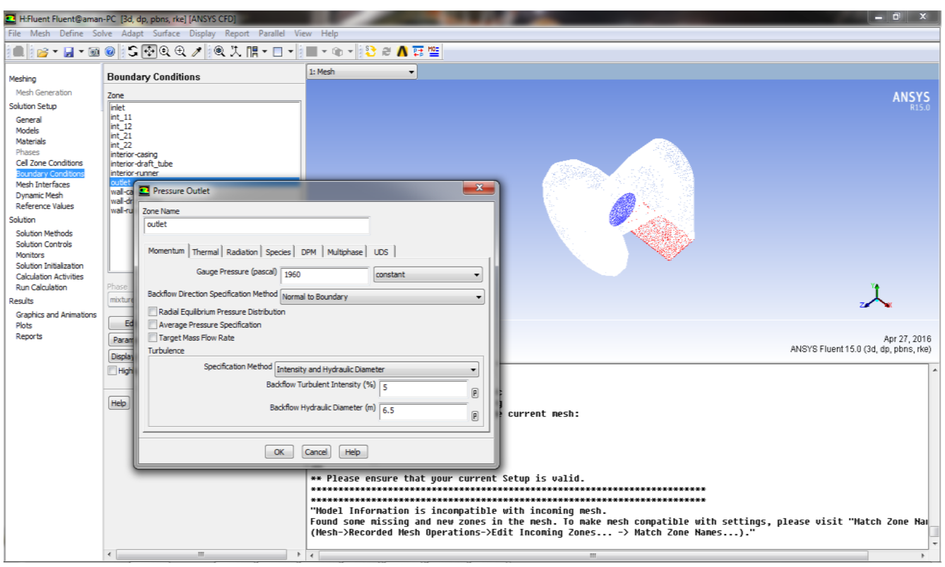

Fig. 3.31 Boundary condition at outlet

VII. Mesh interface: As there are two interfaces so we have to define their relation as shown in Fig. 3.32

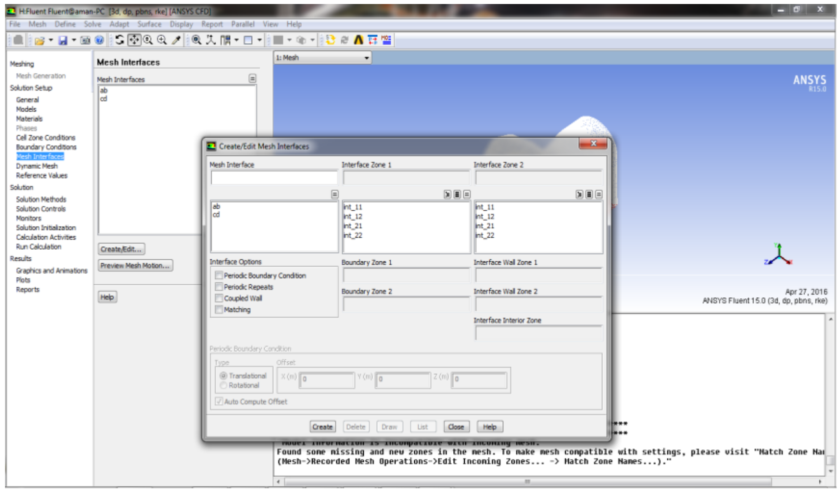

Fig. 3.32 Interaction between different interfaces

VIII. Solution method: we have used simple method for pressure velocity coupling, in spatial discretisation method for gradient is least square cell based method is used, for pressure, momentum, turbulent kinetic energy, turbulent dissipation rate second order upwind used as shown in Fig. 3.33 below

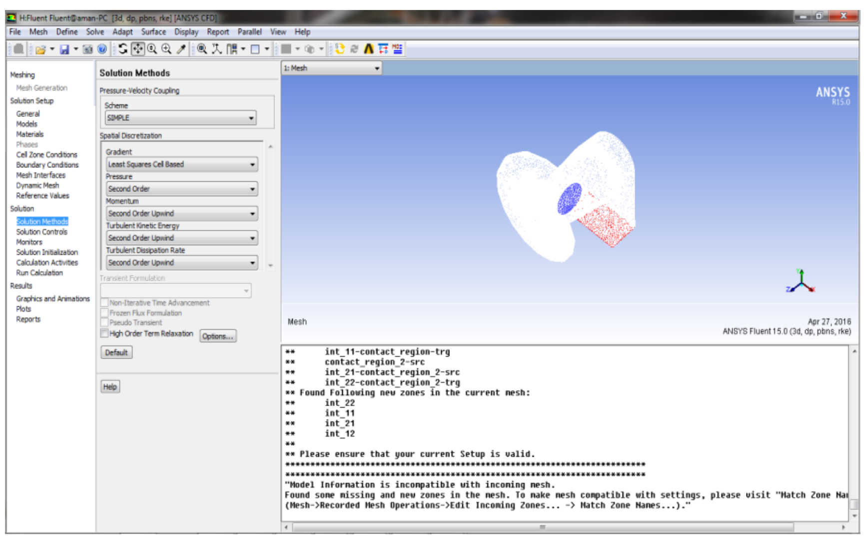

Fig. 3.33 Solution method used to solve the fluid flow problem

IX. Solution initialisation: standard initialisation is used, reference frame taken is absolute and value will be computed from inlet. The Fig. 3.34 is as shown below.

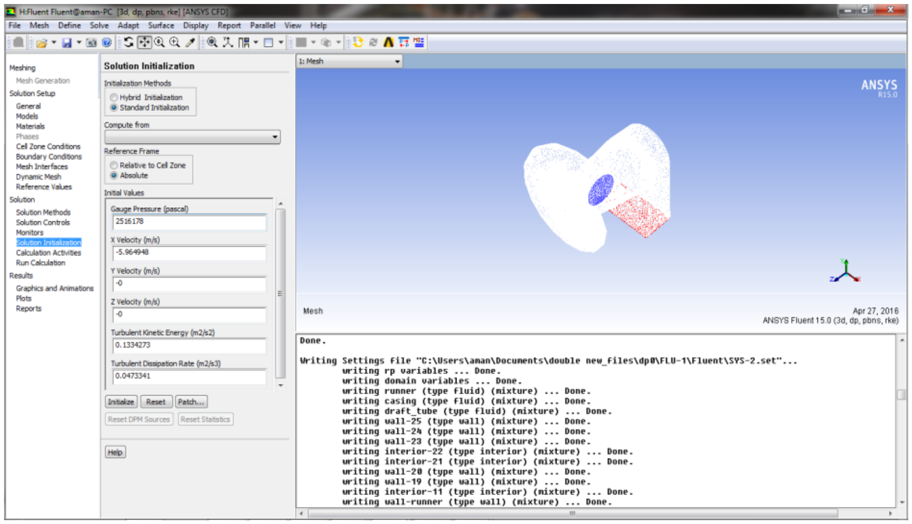

Fig. 3.34 Solution initialisation in FLUENT

X. Run calculations: set up the no. of iterations to get required accuracy. For the present study we are using 500 iterations. The setup is shown as below in Fig 3.35.

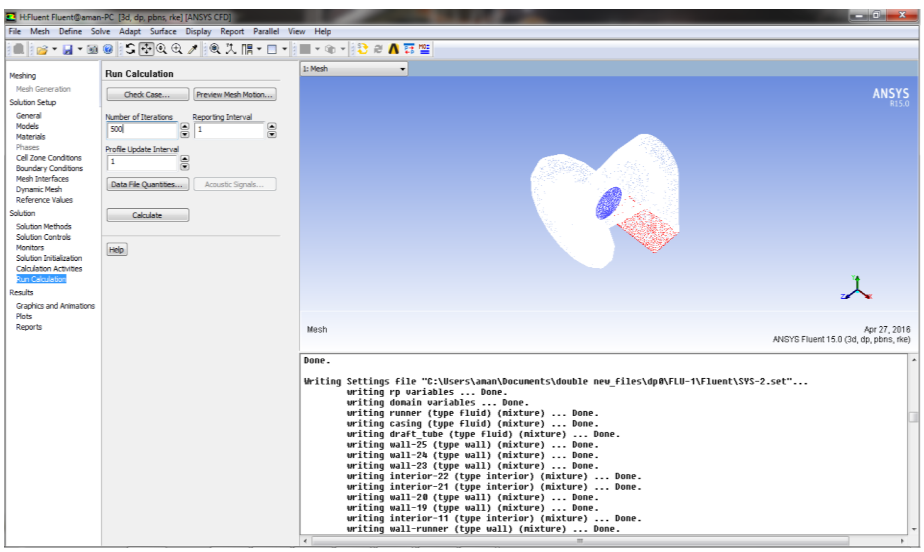

Fig. 3.35 No. of iteration and reporting interval in FLUENT

3.10.2 When Fluid Has Water Vapour Mixture (For Cavitation Model)

To set the cavitation model we have to do following changes in pure water simulation. Changes are mentioned below:

I. The model used in the cavitation analysis is a multiphase model as water-liquid converted into water-vapour and no slip velocity is used as shown in Fig.3.36.

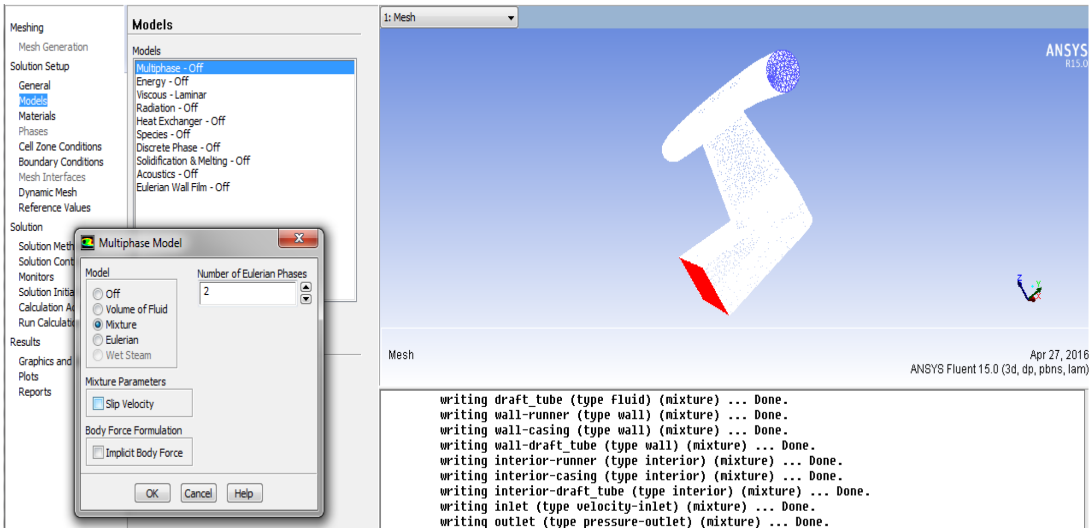

Fig 3.36 Model for cavitation

II. The turbulence model used is Shear stress transport (SST) $K-\omega$ model which is suitable for cavitation analysis as shown in Fig. 3.37

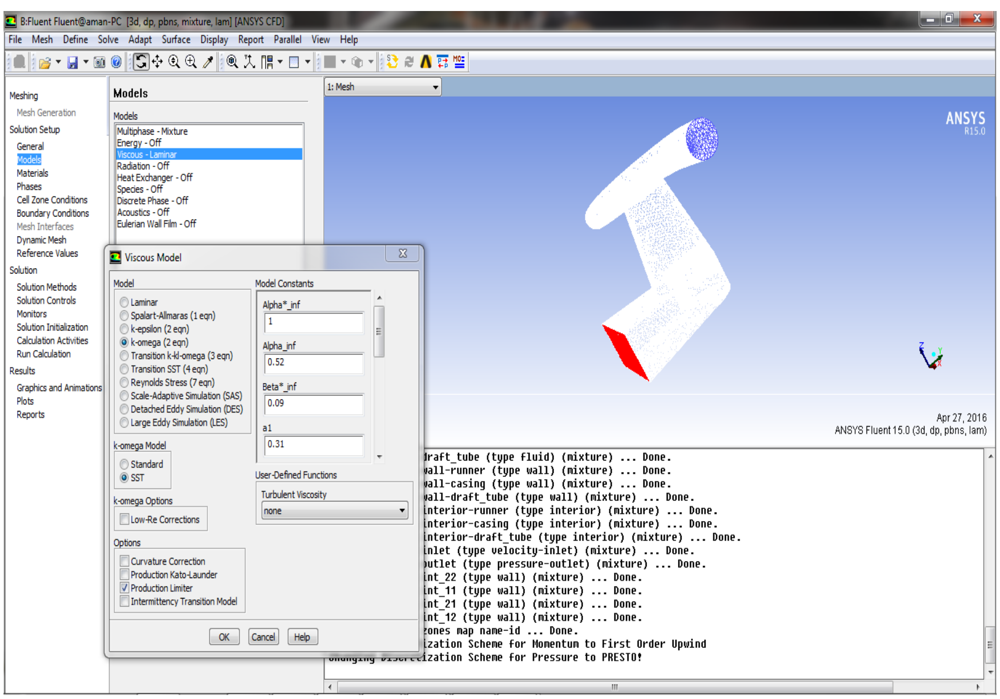

Fig 3.37 Turbulence Model for cavitation

III. There is change of phase from liquid to vapour so we have to fill the type of interaction between phase 1 and phase 2. As shown in Fig.3.38 & 3.39

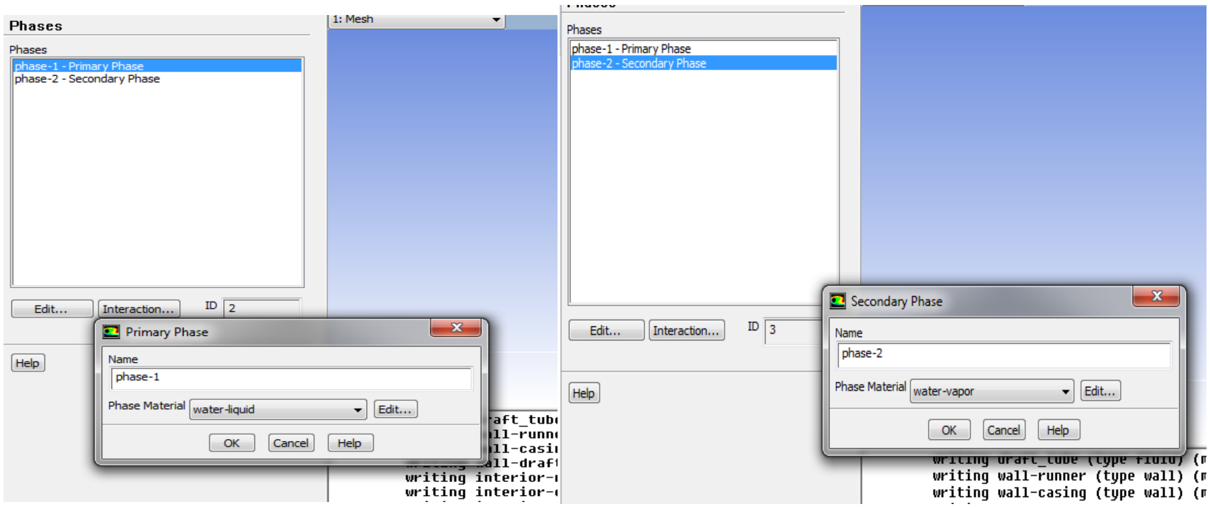

Fig. 3.38 Definition of phase 1 & phase 2

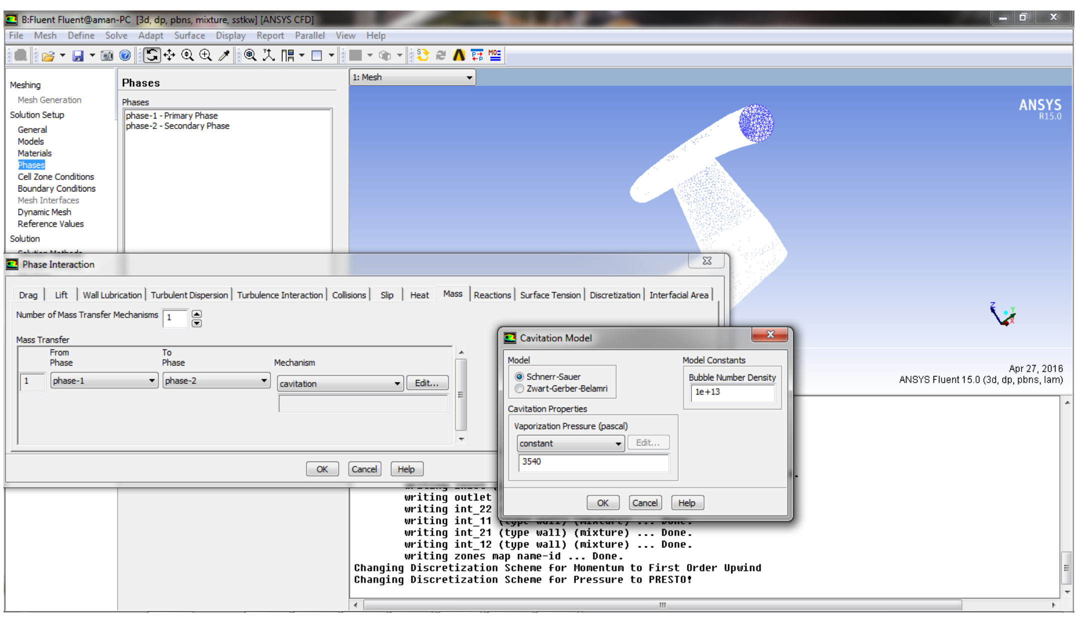

Fig 3.39 Interaction between phase 1 & phase 2

IV. Solution method used for cavitation is as given below i.e. pressure-velocity coupling uses simple scheme, for spatial discretisation gradient is least square node based, pressure is presto!, momentum is second order upwind, volume fraction is quick, turbulent kinetic energy and specific dissipation rate is first order upwind as shown in Fig. 3.40

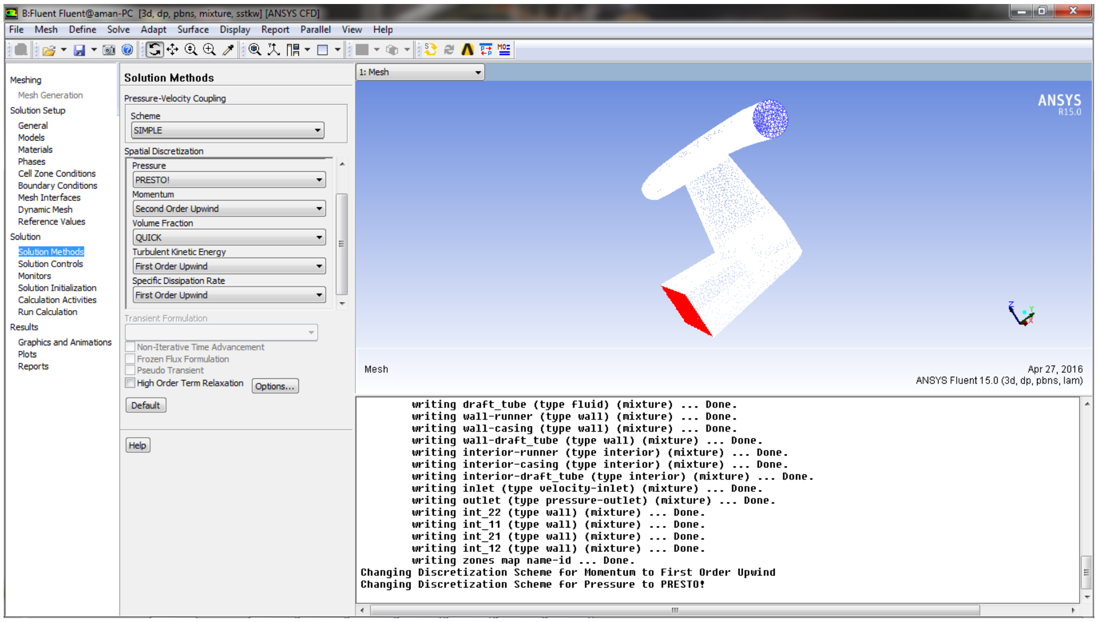

Fig. 3.40 Solution method for cavitation model

CHAPTER 4

RESULT AND DISCUSSION

4.1 GENERAL

The method adopted for cavitation analysis of Francis turbine using CFD package Fluent which is the module of the ANSYS software, is represented in previous chapter 3. The simulation has been carried for the pure water flow parameters and also for vapour-liquid flow parameters. The simulation for mixture of vapour-liquid parameter uses Shear stress transport (SST) $K-\omega$ model turbulence model and for pure water parameters the $K-\varepsilon$ Realizable turbulence model is used. The simulation has been generated for design conditions, various flow conditions and conditions under cavitation for Francis with the help of inlet and outlet boundary condition and for different coating used at runner. The operating characteristic curve, predicted by the numerical simulation are drawn and discussed as below.

4.2 EFFICIENCY COMPUTATION

Efficiency of a turbine is calculated as the ratio of work delivered by turbine to the energy inside the flow of fluid for the incompressible working fluid. The ratio of power output to power input is as given below as:

$$\eta = \frac{T\omega}{Q(P_{t1}-P_{t2})} \quad \text{... (4.1)}$$

Where,

T is the net torque acting on the runner (N-m),

ω is angular speed in (rad/s),

Q is the discharge through turbine in m^3/s,

P_{t1} is Total pressure at the inlet of turbine in Pascal,

P_{t2} is Total pressure at the outlet of turbine in Pascal,

Different parameters have different values at different conditions which are required to calculate efficiency of Francis turbine can be obtained by post processor of fluent software. The input parameters for calculating efficiency are shown below

- Average total pressure at turbine outlet (P_{t2}) in Pascal.
- Discharge in turbine in m^3/s.

- The rotational speed of runner in (N) in rpm.

The output parameters for computing the efficiency are shown below

- Total pressure at turbine inlet (P_{t1}) in Pascal.
- The net torque acting on the runner (T) in Nm.

4.3 SIMULATION ANALYSIS FOR DESIGN CONDITION

The simulation of Francis turbine has been carried out with design parameters which are the discharge is 112.22 m^3/s and head 269 m. For design condition, 100 % discharge has been used. The design condition provides the maximum efficiency for the turbine as compared to the different flow conditions.

4.3.1 Contours for Design Conditions

The total pressure variation in the turbine components obtained through the numerical simulation come in the form of contours as shown below in the Fig.4.1. The runner is the most critical part for the most critical part of turbine for the simulation.

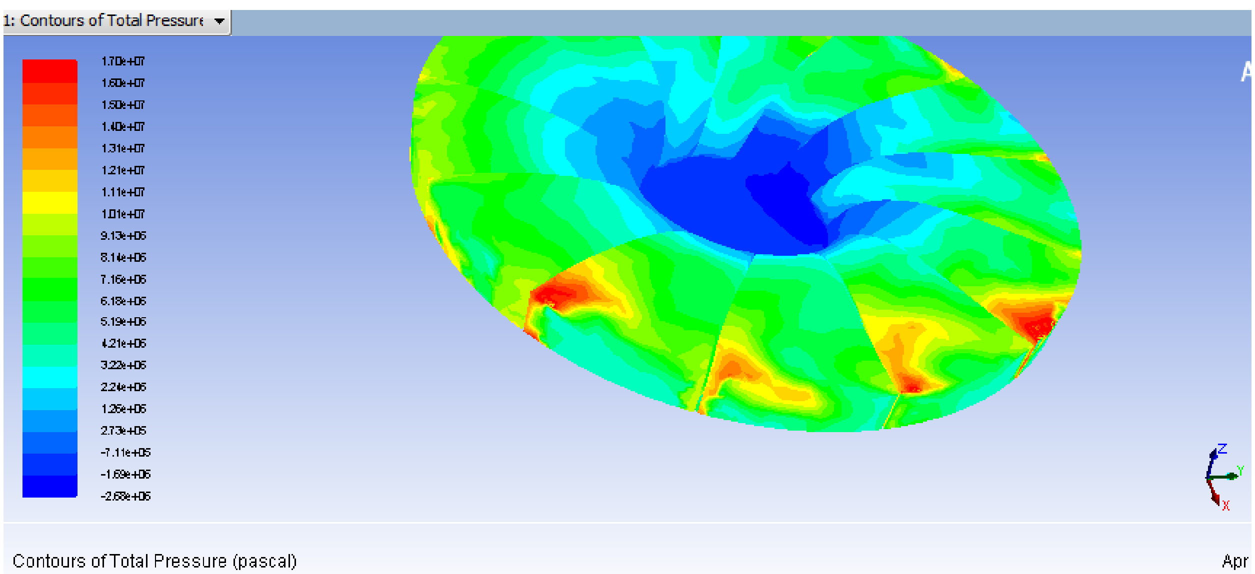

Fig. 4.1 Pressure contour of the turbine runner at rated discharge

The variation of density of water on the runner blade has shown in Fig. 4.2 and it as found that the density of water is same in every point on the surface of all blades.

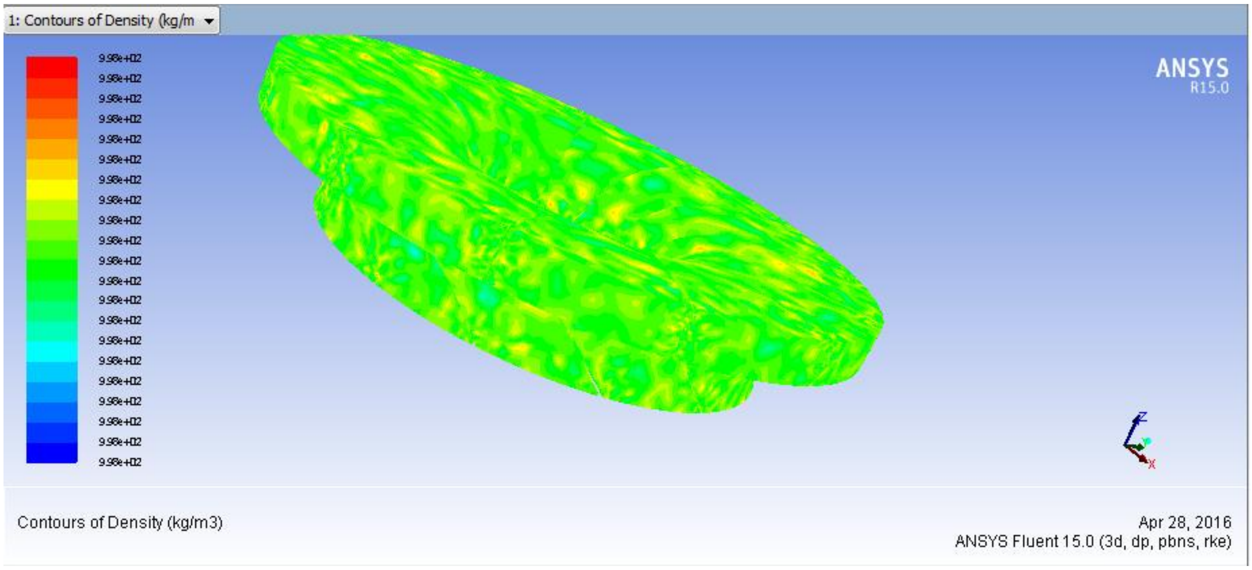

Fig. 4.2 Density contour of fluid in runner

The variation of pressure from the inlet to the outlet point of draft tube has been shown in Fig.4.3 and has been found that there is recovering of pressure in tube so that water can leave the turbine.

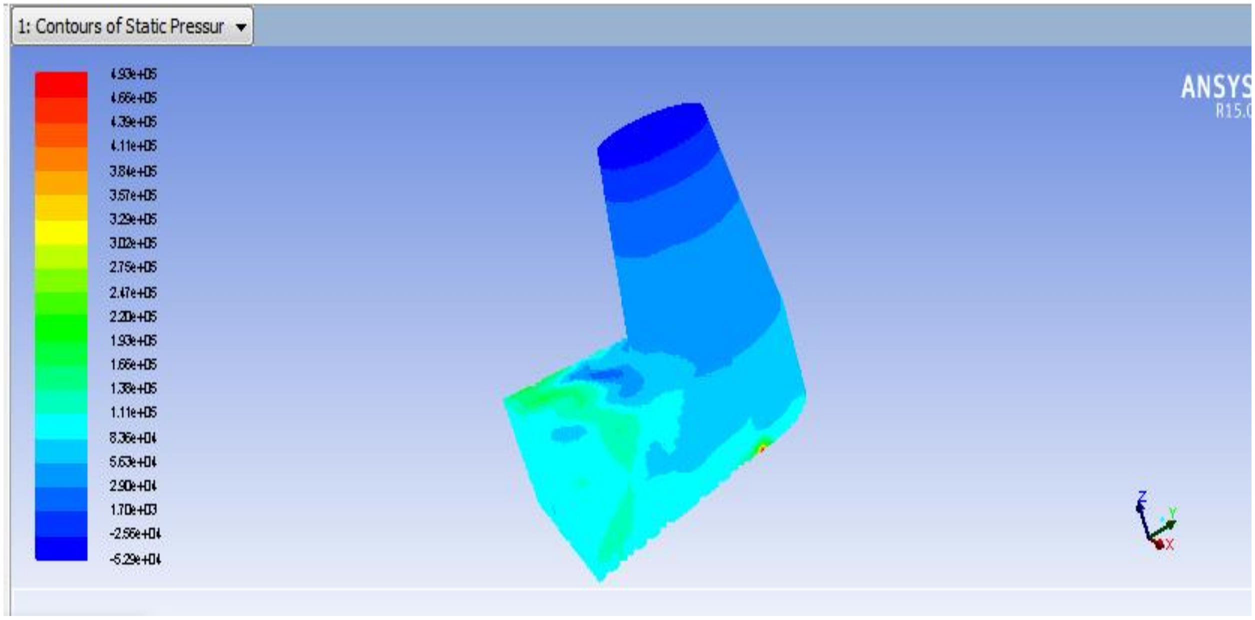

Fig.4.3 Pressure variation in draft tube

4.3.2 CASE (I) Efficiency at Design Condition

The values of input parameters for the fluent software i.e. pressure at outlet of the turbine, rotational speed and discharge is shown below in Table 4.1

Table 4.1: The value of input parameter for design conditions

Parameters	Discharge Q in (m^3/s)	Rotational speed N in (rpm)	Pressure at outlet of turbine (p_{t2})in pascal
Value	112.22	214.29	1960

The output parameter torque acting on the runner and pressure at inlet of the turbinehas been calculated by FLUENT software is shown below in Table 4.2

Table 4.2 The output parameter generated through fluent software

Pressure of inlet of turbine (p_{t1}) in Pascal	Torque in (Nm)
2516178	10368318.13

$$\text{So Efficiency } (\eta) = \frac{T\omega}{Q(P_{t1}-P_{t2})}$$

$$= \frac{10368318.13*2*3.14*214.29}{60*112.22*(2516178-1960)}$$

$$= 82.4\%$$

So the efficiency of Francis turbine for 100% discharge in design condition has been found to be 82.4 % from the equation (4.1)

CASE II for a flow of 110%

Input parameters: discharge Q= 112.22*1.1=123.442 m^3/s

P_{t2} = 1960 Pa

Output parameters: Torque (T) = 11504110.66 Nm

P_{t1} = 2767795.8 Pa

$$\text{So Efficiency } (\eta) = \frac{T\omega}{Q(P_{t1}-P_{t2})}$$

$$= \frac{11504110.66*2*3.14*214.29}{60*123.4*(2767795.8-1960)}$$

$$= 75.6\ \%$$

CASE III for a flow of 80%

Input parameters: discharge Q = 112.22*0.8= 89.776 m^3/s

P_{t2} = 1960 Pa

Output parameters: Torque (T) = 6421441.703 Nm

P_{t1} = 2212942.56 Pa

So Efficiency (η) = $\frac{T\omega}{Q(P_{t1}-P_{t2})}$

$$= \frac{6421441.703*2*3.14*214.29}{60*89.776*(2212942.56-1960)}$$

=72.56 %

CASE IV for a flow of 60%

Input parameters: discharge Q = 112.22*0.6= 67.332 m^3/s

P_{t2} = 1960 Pa

Output parameters: Torque (T) = 4971005.225 Nm

P_{t1} = 1809706.5 Pa

So Efficiency (η) = $\frac{T\omega}{Q(P_{t1}-P_{t2})}$

$$= \frac{4971005.225*2*3.14*214.29}{60*89.776*(1809706.5-1960)}$$

= 68.7 %

4.3.3 The Efficiency vs. Discharge Curve:

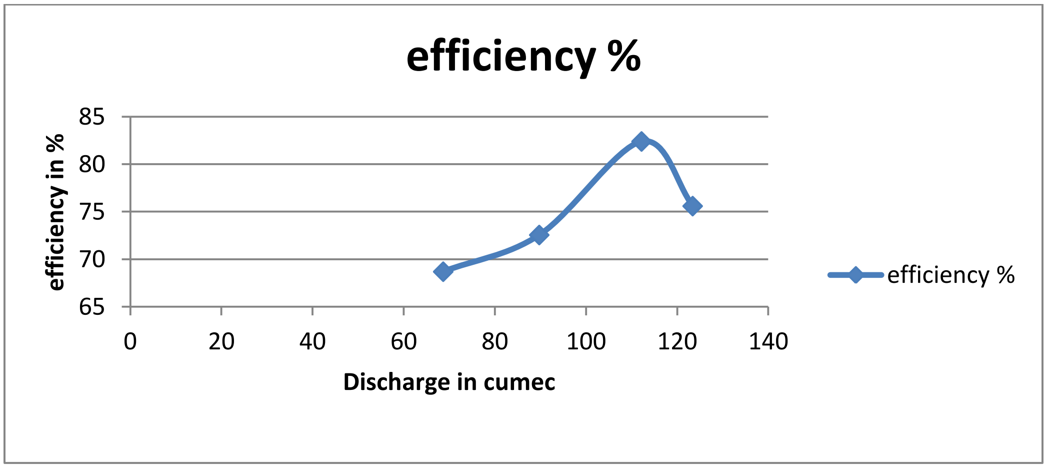

Fig. 4.4: Efficiency – Discharge curve

Fig. 4.4 shows the variation of simulated efficiency with discharge curve.

4.4 SIMULATION ANALYSIS OF DIFFERENT COATING MATERIALS

In an installed plant setup, coating on the turbine parts is one of the methods to prevent turbine from cavitation, there are some other methods like change of turbine setting or change of blade geometry but these cannot be changed after the installation of plant. Turbine is the heart of a hydro power plant so to prevent turbine from cavitation we use many type of coating but in this study we have done some comparison between many types of coatings with the help of ANSYS CFD. So we have used turbine material as material of coating one by one and then checked the mass eroded from turbine and compared different type of coating used for turbine. As the minimum blade thickness is 44mm. We have taken 1000 particle and 1 hour duration for simulation due to which some of the volume from the parts of low vapour pressure has been eroded which is expressed in the volume fraction of runner. Full opening of guide vane is used.

Case I When Cavitec is the coating material

When cavitec is used as coating material on the turbine and applied the cavitation condition in the flow as explained in the last chapter. Volume eroded from turbine is shown in the contours as shown in Fig. 4.5 below for cavitec. Volume eroded from cavitec coating is 813.74 mm^3 during simulation

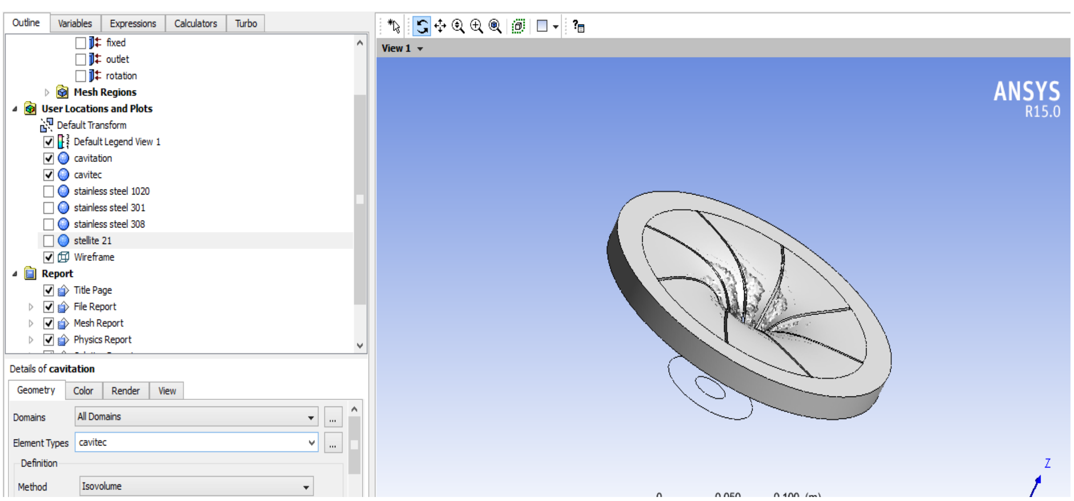

Fig. 4.5 cavitation erosion area when cavitec as a material is used

Case II When Stellite 21 is the coating material

When stellite 21 is used as coating material on the turbine and applied the cavitation condition in the flow as explained in the last chapter. Volume eroded from turbine is shown in the contours as shown in Fig. 4.6 below for stellite 21. Volume eroded from stellite 21 coating is 2034.1373 mm^3.

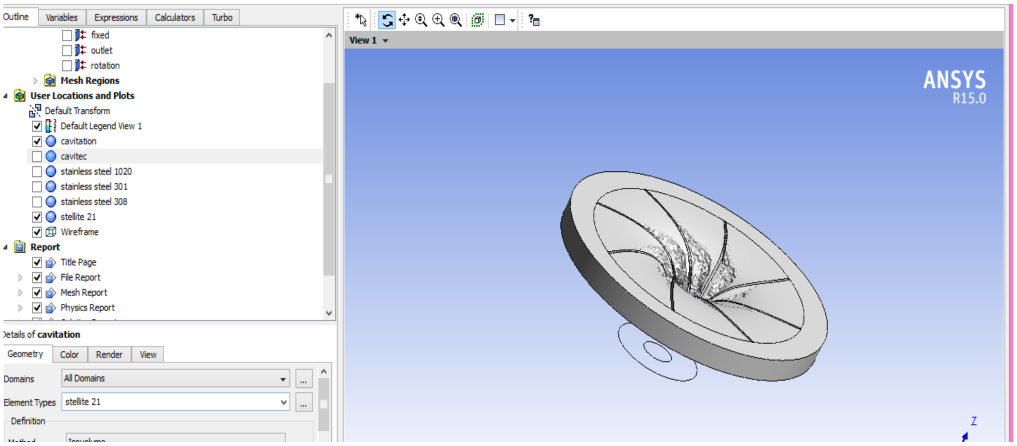

Fig. 4.6 cavitation erosion area when stellite 21 as a material is used

Case III When Stainless steel 301 is the coating material

When Stainless steel 301 is used as coating material on the turbine and applied the cavitation condition in the flow as explained in the last chapter. Volume eroded from turbine is shown in the contours as shown below in Fig 4.7 for Stainless steel 301. Volume eroded from Stainless steel 301 coating is 6105.07 mm^3.

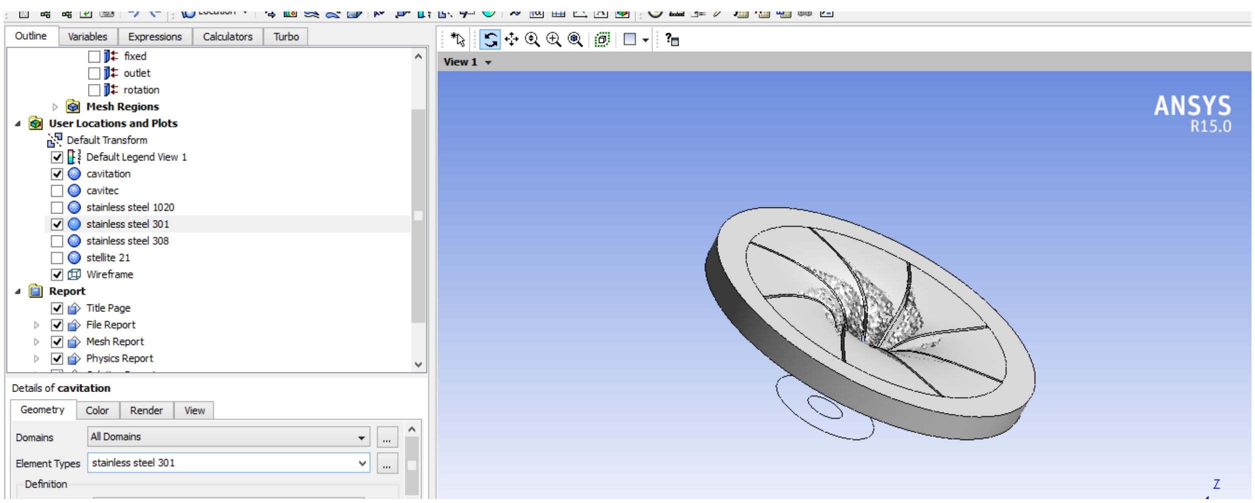

Fig. 4.7 cavitation erosion area when stainless steel 301 as a material is used

Case IV When Stainless steel 308 is the coating material

When Stainless steel 308 is used as coating material on the turbine and applied the cavitation condition in the flow as explained in the last chapter. Volume eroded from turbine is shown in the contours as shown below in Fig.4.8. for Stainless steel 308. Volume fraction eroded from Stainless steel 308 coating is 8391.38 mm^3.

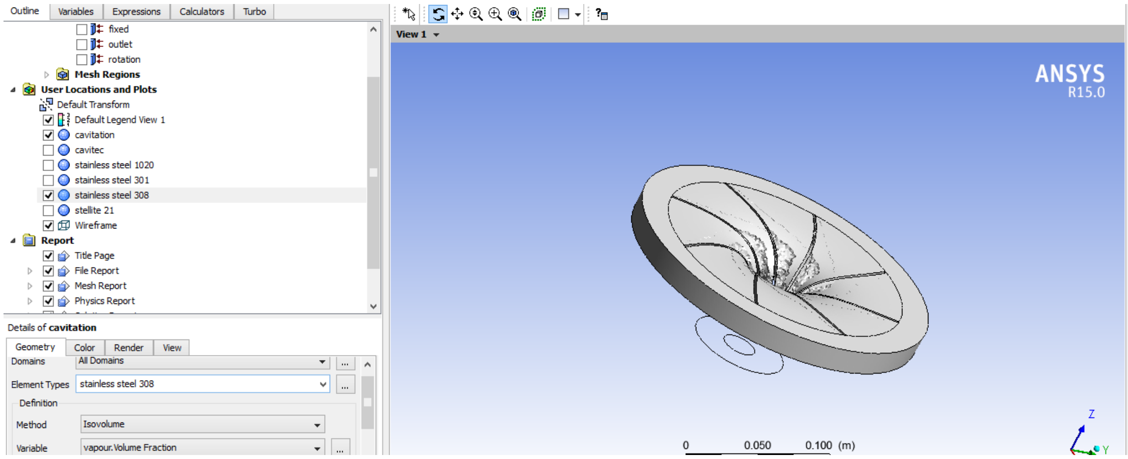

Fig. 4.8 cavitation erosion area when stainless steel 308 as a material is used

Case V When Stainless steel 1020 is the Coating Material

When Stainless steel 1020 is used as coating material on the turbine and applied the cavitation condition in the flow as explained in the last chapter. Volume eroded from turbine is shown in the contours as shown in Fig. 4.9 below for Stainless steel 1020. Volume eroded from Stainless steel 1020 coating is 12850.7 mm^3.

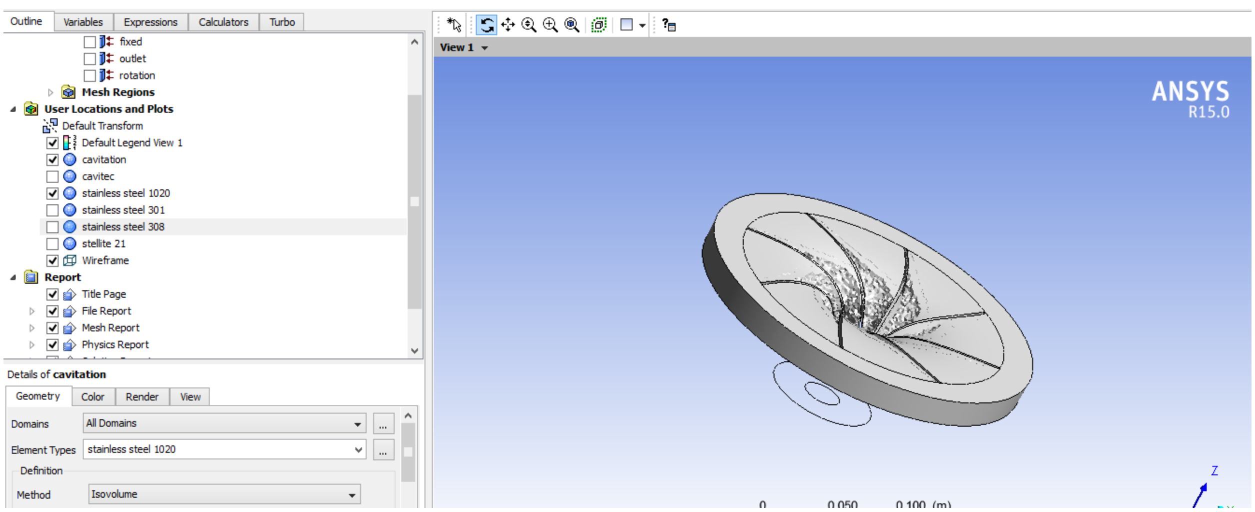

Fig. 4.9 cavitation erosion area when Stainless steel 1020 as a material is used

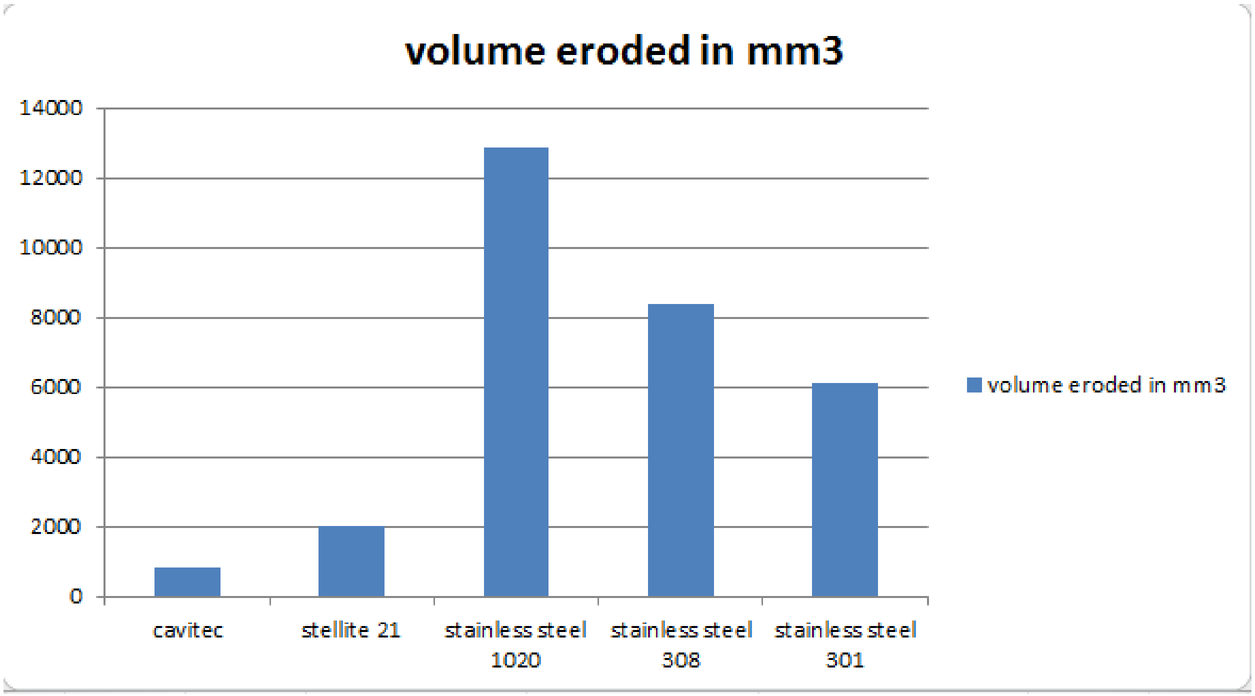

Fig. 4.10 Comparision between different coating materials to prevent from cavitation

According to the comparison in Fig. 4.10 the result is similar to the experimental result given in literature review from paper "A review of cavitation-erosion resistant weld Surfacing alloys for hydro turbines" as if we use material cavitec then there will be less cavitation erosion. Blade profile remains same and there is no loss of efficiency.

CHAPTER 5

CONCLUSIONS

5.1 CONCLUSIONS

In the present study, flow analysis through a Francis turbine has been carried out to investigate the effect of cavitation, using ANSYS CFD package. The rated capacity of the turbine is 255 MW under 269 m head and discharge as 112.22 m^3/s has been considered. The 3-D model of turbine has been generated in AUTODESK INVENTOR software with actual dimensions meshed model was created for different flow conditions. In order to determine the turbine efficiency without cavitation. Based on the study following conclusions are drawn.

I. Pure water flow and $K - \varepsilon$ turbulence models was used in one case and the maximum efficiency of the turbine was found out as 82.4% with 112.22 cumec of discharge at 21.6^o guide vane opening angle. High pressure zone has been observed at the inlet of runner blade.

II. Different type of coating has been used on the runner to prevent runner from cavitation and different rate of volume erosion is there. So according to CFD study the material should be used in the order as observed
Cavitec> stellite 21 > stainless steel 301> stainless steel 308> stainless steel 1020 in order to prevent runner from erosion due to cavitation.

III. In case of large model for better quality of meshing tetrahedral patch conforming method should be used. Smoothing parameter should be kept at low condition.

CHAPTER 6

REFERENCES

1. http://www.ijareeie.com/upload/2014/february/9_Overview.pdf accessed on 26th July 2015
2. http://www.cea.nic.in/reports/monthly/inst_capacity/jan15.pdf accessed on 28th july 2015.
3. http://data.worldbank.org/indicator/EG.ELC.ACCS.ZS accessed on 1st august 2015.
4. https://rigorandrelevance.wordpress.com/2014/03/27/the-value-of-energy-consumpti--on accessed on 5th august 2015
5. http://www.mnre.gov.in/schemes/grid-connected/small-hydro/ accessed on 8/08/2015
6. http://www.iitmandi.ac.in/ireps/images/SmallHydro%20Power%20Generation-IIT%20 Mandi%20may16.pdf accessed on 11/08/2015
7. http://en.wikipedia.org/wiki/Water_turbine accessed on 13/08/2015
8. http://elearning.vtu.ac.in/P6/enotes/CV44/Reac_Tur-MNSP.pdf accessed on 15th august 2015
9. "Hydraulic machine theory and design" by v.p. vasandani
10. Kumar P., Saini R.P., "Study of cavitation in hydro turbines - A review", renewable and sustainable energy reviews, Accepted 13 July 2009.
11. 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 hydrodynamics 2010,22(5):703 -708 DOI: 10.1016/S1001-6058 (09) 60106-2.
12. Xavier Escalera,, Eduard Egusquizaa , Mohamed Farhatb , Franc-ois Avellanb , Miguel Coussirata, ''Detection of cavitation in hydraulic turbines'', Mechanical Systems and Signal Processing 20 (2006) 983–1007 ,accepted 11 August 2004.
13. Hongming Zhanga*, Lixiang Zhang, ''Numerical simulation of cavitating turbulent flow in a high head Francis turbine at part load operation with Open FOAM '', International Conference on Advances in Computational Modelling and Simulation; Procedia Engineering 31 (2012) 156 – 165.

14. Liangliang Zhan ,Yucheng Peng , Xiyang Chen, “Cavitation Vibration Monitoring in the Kaplan Turbine”, Conference Location: Wuhan ; DOI:10.1109/APPEEC.2009.4918211
15. William Duncan, Jr. turbine repair facilities instructions, standards, & techniques volume 2-5, September 2000
16. Branko Bajic et al., “methods for vibro-acoustic diagnostics of turbine cavitation”, Journal of Hydraulic Research, Published online: 01 Feb 2010.
17. Sebastian Muntean et al., “numerical investigation of 3d cavitating flow in Francis turbines ”, Conference on Modelling Fluid Flow (CMFF’03) The 12th International Conference on Fluid Flow Technologies Budapest, Hungary, September 3 - 6, 2003
18. T. Pramod, R.K. Kumar, S. Seetharamu and M. Kamara, “Effect of Porosity on Cavitation Erosion Resistance of HVOF Processed Tungsten Carbide Coatings”, International Journal of Advanced Mechanical Engineering. ISSN 2250-3234 Volume 4, Number 3 (2014), pp. 307-314.
19. Hart d., and whale d, “A review of cavitation-erosion resistant weld Surfacing alloys for hydro turbines” http://www.castolin.com/publication/review-cavitation-erosion-resistant-weld-surfacing-alloys-hydroturbines.
20. Prasad , V. gahlot & krinnamachar P., “CFD approach for design optimisation and validation for axial flow hydraulic turbine” Indian journal of engineering & material science vol. 16, august 2009,pp 229-236
21. P Drtina* andM Sallaberger, “Hydraulic turbines—basic principles and state-of-theart computational fluid dynamics applications” Proc Instn Mech Engrs Vol 213 Part C 1999
22. Jingchun Wu, Katsumasa Shimmei, Kiyohito Tani & Kazuo Niikura, “CFD-Based Design Optimization for Hydro Turbines” Journal of Fluids Engineering FEBRUARY 2007, Vol. 129, pp 159-168
23. A Lipej, “Optimization method for the design of axial hydraulic turbines” Journal Power and Energy 2004 218:43
24. Ruchi Khare, Vishnu Prasad and Sushil Kumar, “ Derivation of Global Parametric Performance of Mixed Flow Hydraulic Turbine Using CFD” hydro Nepal issue no. 7 july,2010

25. Suthep Kaewnai and Somchai Wongwises. “ Improvement of the Runner Design of Francis Turbine using Computational Fluid Dynamics” American J. of Engineering and Applied Sciences 4 (4): 540-547, 2011
26. Kiran Patel, Jaymin Desai, Vishal Chauhan and Shahil Charnia, “ Development of Francis Turbine using Computational Fluid Dynamics” The 11th Asian International Conference on Fluid Machinery Paper ID: AICFM_TM_015
27. IS 12800 part 1 (1993), “ guidelines for selection of turbine preliminary dimensioning and layout of surface hydroelectric power houses”
28. IS 5496, “Guide For Preliminary Dimensioning And Layout Of Elbow Type Draft Tubes For Surface Hydroelectric Power Stations”
29. www.ivt.ntnu.no/ept/fag/.../8%20-%20**Guide**%20**Vanes**%20in%20Francisturbines. Accessed on 13th October 2015.