CFD BASED PERFORMANCE ANALYSIS OF MODIFIED PUMP USED AS TURBINE

A DISSERTATION

Submitted in partial fulfillment of the requirements for the award of the degree
of
MASTER OF TECHNOLOGY
in
ALTERNATE HYDRO ENERGY SYSTEMS

By
ANUJ KUMAR

ALTERNATE HYDRO ENERGY CENTRE
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE -247 667 (INDIA)
JUNE, 2012

CANDIDATE'S DECLARATION

I hereby declare that the work which is presented in this dissertation entitled, "**CFD BASED PERFORMANCE ANALYSIS OF MODIFIED PUMP USED AS TURBINE**", 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 July 2011 to June 2012 under the supervision and guidance of **Mr. M. K. Singhal,** Senior Scientific Officer, **Dr. R. P. Saini**, Associate Professor & Head, 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 or diploma.

Date: June 12, 2012

Place: Roorkee

(ANUJ KUMAR)

CERTIFICATE

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

(Mr. M. K. Singhal)
Senior Scientific Officer ,
Alternate Hydro Energy Centre,
Indian Institute of Technology Roorkee,
Roorkee – 247 677

(Dr. R. P. Saini)
Associate Professor & Head,
Alternate Hydro Energy centre,
Indian Institute of Technology Roorkee,
Roorkee – 247 677

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 **Mr. M. K. Singhal**, Senior Scientific Officer and **Dr. R. P. Saini**, Associate Professor & Head, Alternate Hydro Energy Centre, Indian Institute of Technology Roorkee for their valuable guidance and infilling support for the completion of dissertation work.

I am also grateful to 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.

Last but not the least, I would like to express my humble respect and special thanks to my parents **Mr. Anand Kar Kulshrestha** and **Mrs. Sneh Lata** and others who have directly or indirectly helped me for the completion of dissertation work

Date: June 11, 2012
Place: Roorkee

Anuj Kr.
(ANUJ KUMAR)

ABSTRACT

Energy plays an important role in almost all areas of human and commercial activities. Electricity generation through the non-renewable sources is quite common to meet the demand of the growing population and developing world. But the non-renewable energy sources are not expedient from ecological point of view and also responsible for producing major greenhouse gas (GHG) and promote global warming. These impediments forces to look for other clean and cheap sources for energy generation. The current concern on the global environment has imposed restraints on the production of electricity. The emphasis is put on the development of environmental friendly methods to promote the sustainable social development. It is in these circumstances, that micro hydro power is drawing more attention.

The conventional turbine technologies like Pelton and cross flow turbines that have been implemented in micro hydropower sites having different head and discharge, have to be custom-made and are therefore more expensive also need expert design and precise manufacturing skills for a good performance. This often becomes a bottleneck especially for low capacity projects. An alternative of using well-known approach of '**Pump as Turbine**' can be contemplated and popularized. A large number of theoretical and experimental studies have been reported to predict the performance of centrifugal pumps in turbine mode. However, there is a need to study further this concept in order to get the advantages for sustainable development. Recently attempts to predict performance of PAT have been made using computational fluid dynamics. Some modifications in pump to improve PAT efficiency have also been attempted. Experimental study has also been carried out in order to improve the part load efficiency of pump in the turbine.

Under the present investigation, performance analysis of a commercial centrifugal pump provided with movable guide vanes to improve part load efficiency in turbine mode has been carried out using Computational Fluid Dynamic. Complete model for simulation was prepared using Pro-E 2.0 and mesh was generated using ANSYS 'MESH'. Simulation was carried out for different guide vanes positions using ANSYS 'FLUENT'. Operating characteristics for modified PAT was drawn and compared with available experimental results. A good agreement has been found between the numerical and experimental results.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

Symbol

H	head (m)
Q	discharge (m^3/s)
N	rotational speed (rpm)
N_s	specific speed (m, m^3/s)
h	head ratio
q	flow rate ratio
p	pressure (Pa)
P	power W or kW
T	torsion moment (N-m)
V	velocity of impeller (m/s)
ω	angular velocity (rad/s)
u	peripheral velocity of impeller
g	acceleration due to gravity, (m/s^2)
D	dia of impeller
b	width of impeller
τ	shear stress of shaft material (MN/m^2)
β	blade angle

Abbreviations

kWh	kilowatt hour
PRV	pressure reducing valve
PAT	pump as turbine
CFD	computational fluid dynamics
BEP	best efficiency point
GHG	green house gas

Greek symbols

η	hydraulic efficiency
γ	specific weight

Subscripts

t	turbine
p	pump
n	net

m	mechanical loss
v	volute loss
l	leakage loss
e	kinetic energy loss
i	loss in impeller
1	for inlet
2	for outlet
x	for x-direction
y	for Y-direction
z	for z-direction
f	for flow
w	for whirl
r	for relative

CHAPTER 1

INTRODUCTION AND LITERATURE REVIEW

1.1 GENERAL

Energy plays an important role in almost all areas of human and commercial activities, and it is very important input for those countries that are developing from economic point of view. It is the tool to forge the economic growth of the country. Every sector of Indian economy – industry, agricultural, transport, commercial and domestic, needs input of energy. There has been an ever-increasing need for more and more power generation as energy demand is increasing day by day in all countries of the world. In the true global perspective of the power demand it can be laid with certainty that many of the developing countries of the world are now a days, experiencing the "energy crisis".

Adequate generation of electricity is essential to develop the economy's infrastructure of a country. Electricity generation through the non-renewable sources is quite common to meet the demand of the growing population and developing world thereby pressurizing the energy sources like coal, oil, natural gas, uranium. But the fast depleting nature and increasing prices of petroleum products (coal, oil, gas, etc.) are the major complications in fulfilling the power demands from these sources. In addition to this, the non-renewable energy sources are not expedient from ecological point of view. They are responsible for producing various gases especially 'CO_2' which is a major greenhouse gas (GHG) and promotes global warming. Other problems like acid rain, toxic wastes are also associated with the use of non-renewable energy sources. These impediments forces to look for other clean and cheap sources for energy generation.

So in this present scenario, there is a need to move to other sources of energy so as to face the energy crisis and one of the solutions to this may be the use of abundant renewable energy resources around. Renewable energy would play a major role in energy industry of the twenty-first century and beyond. These alternative energy resources will not only help in reducing greenhouse emissions but may also cater the need of world in the next century. The renewable energy sources are solar power, wind power hydroelectric power, biomass materials, geothermal energy, tidal energy, wave energy and hydrogen as fuel cells, among which hydropower is the second largest renewable source.

The total installed capacity of electricity generation in India as on 31 March 2012 is 199627MW out of which 131353MW (65.79%) through thermal, 38990 MW (19.53%) through hydro, 4,780MW (2.39 %) through nuclear and 24503 MW (12.27%) through other renewable energy source [1]. The overall generation in the country has increased from 771.551 BU during 2009-10 to 811.143 BU during the year 2010-11.The growth rate of Hydro power improved by 9.97% [2]. According to International Energy Agency (IEA), a threefold rise in India's generation capacity is expected by 2020. Among all the renewable energy sources, small hydropower is the most promising and significant source. Table 1.1 shows the growth of electricity generation through thermal, hydro, nuclear since 1950. The figures of the tables show that the hydropower generation is increased substantially in last 60 years.

Table 1.1: Growth of electricity generation [3]

S. No.	Year	Thermal (MW)	Hydro (MW)	Nuclear (MW)	Total (MW)
1.	1950	1153	559	-	1712
2.	1960	2736	1417	-	4653
3.	1970	7906	6383	420	14709
4.	1980	17562	11791	860	30213
5.	1990	43764	18307	1565	63638
6.	2001	73273	25574	2860	101708
7.	2002	76057	26269	2720	105046
8.	2004	80457	29507	2720	112864
9.	2006	89962	33193	3900	127056
10.	2007	90173	33600	3900	127673
11.	2009	96045	36917	4120	137082
12.	2012	131353	38990	4780	175123

India is blessed with immense amount of hydro-electric potential and ranks 5th in terms of exploitable hydro-potential on global scenario. As per assessment made by CEA, Our country is endowed with enormous economically exploitable and viable hydro potential assessed to be about 84,000 MW at 60% load factor (1,48,700 MW installed capacity). In addition, 6781MW in terms of installed capacity from small, mini and micro hydel schemes have also been assessed. Also, 56 sites for pumped storage schemes with an aggregate installed capacity of 94,000 MW have been identified. However, only 15% of the hydroelectric potential has been harnessed so far and 7% is under various stages of development. Thus, 78% of the potential remain without any plan for exploitation [4]. As such, large hydro is also renewable in nature, but it is associated with some of the major problems like, land submergence, Resettlement and Rehabilitation, which ultimately results into vary long gestation period of the Large Hydropower Projects, may be up to 30-40 years. Considering these facts, there is a large scope of development in small and micro hydro power sector in India. So, Ministry of Renewable Energy has been promoting micro and small hydro projects ($\leq$ 25MW) so as to provide energy to remote and hilly areas due to their environment friendly nature and having no problem of large water storage and rehabilitation of population. Also, electricity generation through small hydro is very well for sustainable development.

1.2 SMALL HYDRO POWER

There is a general tendency all over the world to define Small Hydropower by the power output. Different countries follow different norms, the upper limit ranges between 5 to 50 MW, as given in the Table 1.2. Small hydro projects are economically viable and have relatively short gestation period. The major constraints associated with large hydro projects are usually not encountered in small hydro projects.

Table 1.2: Worldwide definition of SHP

S. No.	Country	Capacity (MW)
1.	UK	≤ 5
2.	UNIDO	≤ 10
3.	Sweden	≤ 15
4.	Colombia	≤ 20
5.	Australia	≤ 20
6.	**India**	$\leq$ **25**
7.	China	≤ 25
8.	Philippines	≤ 50
9.	New Zealand	≤ 50

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 [5]. Small hydro power projects are further classified in Table1.3.

Table 1.3: Classification of small hydro power in India [5]

S. No.	Class	Station Capacity in kW
1.	Micro Hydro	Up to 100
2.	Mini Hydro	101 to 2000
3.	Small Hydro	2001 to 2500

1.2.1 Benefits of Small Hydro Power

In addition to hydro power being a renewable source of generating power, unlike wood, coal, oil and natural gas during the combustion of which carbon dioxide, Sulphur dioxide and methane gases are released into the atmosphere –and consequently contributes to the greenhouse effect – the prudent utilization of hydropower leaves the ecosystem largely untouched. Among the sources of alternative renewable energies, hydropower is attractive as other renewable sources like wind and solar are available intermittently.

The biggest advantage of Small Hydro Power (SHP) is that it is 'clean' and renewable source of energy available round the clock [6]. It is free from many issues and controversies that continue to 'hound' large hydro, like the submergence of forests, siltation of reservoirs, rehabilitation and relocation, and seismological threats. Other benefits of small hydro are user-friendliness, low running cost, and short gestation period enabling quicker returns, flexibility of installation and operation in an isolated mode and also in a localized or regional grid system; it solves the low voltage problem in the remote hilly areas and helps in reducing the losses in transmission and distribution etc.

In addition to these obvious benefits, SHP contributes numerous economic benefits as well. It has served to enhance economic development and living standards especially in remote areas with limited or no electricity. On the macro level, rural communities have been able to attract new industries mostly related to agriculture owing to their ability to draw power from SHP stations.

1.2.2 Advantages of Micro Hydro Power

The current concern on the global environment has imposed restraints on the production of electricity. The emphasis is put on the development of environmental friendly methods to promote the sustainable social development. It is in these circumstances, that micro hydro power is drawing more attention. In rural areas, electricity is required for lighting and mechanization of agriculture tasks. Advent of electricity in rural areas is an effective solution of improvement of rural people. The area without energy accessibility comprises islands, hills, forest etc. In this area, energy sources are inadequate and conventional power plants or grid extensions are very difficult to establish. Micro-hydropower projects are the excellent alternative for electricity generation in remote areas. Micro- hydropower projects can be installed on small streams, small rivers, and canals without any recognizable effect on environment to meet the energy needs of small villages in rural areas. In addition to meeting the needs of an area, a properly designed, installed and managed micro-hydro power plant can also contribute significantly towards employment generation, improved living conditions and improved education facilities.

The civil works for micro hydro power stations are simple. They do not need elaborate construction works in reinforced concrete, no expensive powerhouse and highly optimized electromechanical equipment. Operation is simple and maintenance is easy. Under favorable circumstances, micro-hydro represents one of the cheapest methods of electricity generation. There is no need for a long transmission lines because output is consumed near the source itself. In short, Micro hydropower represents one of the cheapest solutions to offer the benefits of electrification in isolated rural communities. Micro hydro power projects use available small water stream of water and do not contribute to environment damage, offering decentralized electrification at a low running cost and with long life.

1.2.3 Disadvantages and Constraints of Micro Hydro Power

Hydro project have an important characteristic that no two potential sites are alike. Topography, flow regime and volume of the river concerned together with the geological condition of the site are variables that make each installation unique. The high capital cost of hydropower development has been a constraint to the development of Micro Hydro Power plants. This is especially true regarding the problems of regulating speed of the generating set. The identification in these schemes requires physical involvement and reconnaissance at micro level of the state organization. Capital cost for such plant is relatively high due to high equipment cost when compared with the small amount of power generation possible which is a constraint in development of such scheme. Equipment cost is high because there are number of different micro hydropower sites having different head and discharge; therefore turbines need to be tailored for particular sites. This process is a costly one as design and fabrication of turbine is complicated. So, this is the reason why micro hydro power development is not attractive. So, by reducing the equipment cost in micro hydropower projects can become more useful and accessible.

1.2.4 Types of Hydro Turbine used in Micro Hydro Scheme

In case of selection of turbines for a micro hydropower, as can be seen from Fig 1.1 the usual alternatives would be either cross flow turbine or Pump as Turbine. The Cross Flow Turbine used in the micro hydro power range suffers from poor efficiency though the construction is simple and can be carried out easily.

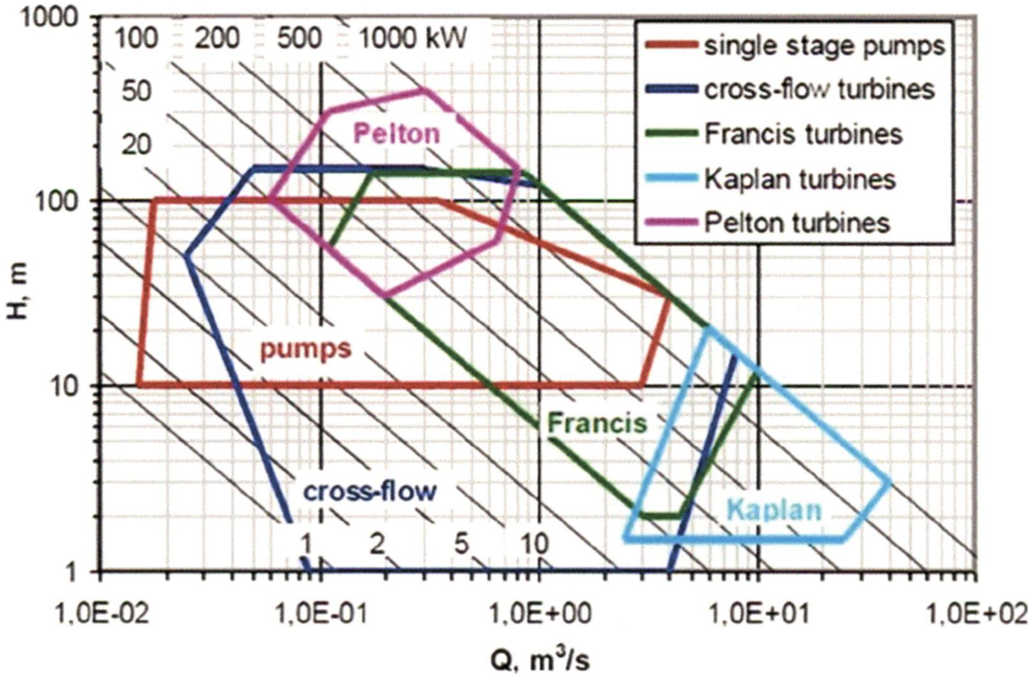

Fig. 1.1 Typical operation range of PAT [7]

One of the easiest ways to reduce the equipment cost is the use of centrifugal pump in reverse mode and can be used as an alternative to conventional hydraulic turbine. Pumps are relatively simple machines with no special designing and are readily available in most developing countries. The basic principle work of hydro-turbines are reversal of pumps, therefore, an alternative solution that can be developed in overcoming problem to get hydro turbines for micro hydro schemes are by using pumps, by flowing water in the reverse direction through in the pumps, as hydro turbines. Hydraulic pumps operated in turbine mode with good efficiency are much less expensive than turbines due to their large-scale production, absence of hydraulic control components. In contrast to a conventional turbine, a reverse running pump turbine has no inlet guide vanes; therefore, the variable discharge characteristics are slightly different to the Francis turbine. However, it has the following advantages: uniform quality, simple construction and durability. A pump operated as turbine also has almost same efficiency as the pump, which is competitive with other turbines type; a small number of parts enable easy maintenance and inspection. Pump manufacturers do not normally provide characteristic curves of their pumps working as turbines. This makes it difficult to select an appropriate pump to run as a turbine for a specific operating condition. From the economical point of view, it is often stated that capital payback period of PATs in the range of 5–500 kW is two years or less. The problem with PAT arises when there are seasonal variations in flow rate. As there are no guide vanes in standard pump units, therefore when the discharge varies or the load varies there is loss of efficiency of PAT resulting in poor part load efficiency of PAT.

1.3 INTRODUCTION TO COMPUTATION FLUID DYNAMICS

Computational fluid dynamics (CFD) is concerned with numerical solution of differential equations governing transport of mass, momentum, and energy in moving fluids. CFD activity emerged and gained prominence with availability of computers in the early 1960s. Today, CFD finds extensive usage in basic and applied research, in design of engineering equipment. CFD does provide us with valuable information for understanding the complicated fluid flows in the turbo machinery and help to optimize the design and to improve the performance of machines.

1.3.1 Industrial Application of CFD in the Design of Hydraulic Turbines

In last two decades, the CFD technology has become an integrated part of the engineering design of hydraulic turbines. There are following advantages of this advanced numerical methodology:

1. The turbo machinery flows can be more accurately predicted. This allows designers to explore more design alternatives, which would otherwise be too time consuming or are outside the range of previous experience.
2. The application of CFD in hydraulic design enables designers to have better control on the flow behavior. Higher turbine loading and flow capacity can be achieved without causing detrimental flow phenomena such as flow separation, cavitations and chocking. In consequence, the size and therefore the cost of the turbine can be reduced.
3. The modern computer visualization of flow fields helps the turbine designer understand the flow in turbine passages better. The intuitive representation of the flow fields makes the potential problems of the design easily identified. This can reduce the hydro dynamic losses and improve the performance of the turbine system.
4. The numerical rigs can reduce the turbine development time and costs. In some circumstances, the numerical simulation is satisfactory and can replace the model experiment, which is costly and time consuming.

1.3.2 Limitations of CFD Methodology

Every engineering design method has its limitations. It is important for the designers to be aware of these limitations, and the source of error and accuracy of the method. Coleman and Stern pointed out the source of error associated with CFD flow analysis can be divided into two broad categories: modeling errors and numerical errors.

The modeling errors category includes errors due to assumptions and approximations in the mathematical representation of physical process and errors due to the incorporation of the empirical data into the model.

The numerical errors include computer round off errors and errors due to the numerical schemes which includes the aspects such as discretization, grid density and orthogonality, artificial dissipation and convergence. The most discussion worthwhile aspect is the grid density i.e. the grid dependency of CFD simulations. It is obvious that the finer the grid the better the prediction and the costs and the time of simulation.

1.3.3 Concept of Computation Fluid Dynamics

CFD is the analysis of the system involving fluid flow, heat transfer and associated phenomena such as chemical reaction by means of computer based simulation. The physical aspect of any fluid flow is governed by three conservation laws i.e. conservation of mass, conservation of momentum and conservation of momentum. But the physical aspect of fluid flow in hydro turbine governed by two fundamental laws those are given below:

- **Conservation of mass:** The mass of fluid is conserved.

$$\frac{\partial \rho}{\partial t} + \nabla \bullet \left(\rho \vec{V}\right) = 0 \tag{1.1}$$

- **Conservation of Momentum:** The rate change of momentum is equal to sum of the forces on a fluid particle, in the same direction (Newton's second law).

$$\rho\left[\frac{\partial \vec{V}}{\partial t} + \vec{V} \bullet \nabla \vec{V}\right] = F_b - \nabla p + \mu \nabla^2 \vec{V} + \frac{\mu}{3}\nabla\left(\nabla \bullet \vec{V}\right) \tag{1.2}$$

1.3.4 CFD Analysis

Basic concepts

Following are the basic concepts of CFD analysis:

a. Problem identification and pre-processing
   i. Define modeling goals.
   ii. Identify the domain which will be model.
   iii. Design and create the grid.
b. Solver execution
   iv. Set up numerical model.
   v. Compute and monitor the solution.
c. Post processing
   vi. Examine the results.
   vii. Consider revisions to the model.

1.3.5 Steps to solve the Problem using CFD Approach

The various steps required to solve the problem using CFD are described below [8]:

i. Creation of Mathematical Model
ii. Choose a Discretization Method

There are many methods, but the most important ones are:

a. Finite Difference Method (FDM)
b. Finite Volume Method (FVM) and
c. Finite Element Method (FEM)
   i. Numerical Grid Generation
   ii. Finite Approximation
   iii. Solution of Algebraic Equations
   iv. Convergence Criteria

1.3.6 Numerical Simulation Methodology

Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms to known quantities [9].

FLUENT provides the following choices of turbulence models:

i. k-ε Models

    a. Standard k-ε model
    b. Renormalization-group (RNG) $k-\varepsilon$ model
    c. Realizable k-ε model

ii. k-w Models

    a. Standard $k-\omega$ model
    b. Shear-stress transport (SST) $k-\omega$ model

iii. v^2-f Model

iv. Reynolds Stress Model (RSM)

v. Detached eddy simulation(DES) model

vi. Large eddy simulation (LES) model

1.4 LITERATURE REVIEW

The concept of electricity generation through reverse running centrifugal pump is not new. Around 80 years ago, the research on this field had been started. A large number of theoretical and experimental studies have been done for prediction of performance of reverse running centrifugal pumps. But no method is reliable for the entire specific speed range and results obtained by these relations had almost ±20% deviation from experimental data [10]. Therefore, still there is a need to explore this area more deeply to harness the advantages of this technology for sustainable development. Most recent attempts to predict performance of PAT have been made using computational fluid dynamics. However, without verifying the CFD results by experimental data, they are not reliable. In order to understand the PAT behavior, performance prediction of pump in turbine mode, modification in pump to improve efficiency of PAT and application of CFD in this area and identify the, the studies carried out by various researchers will be discussed and gaps for present dissertation will be identified.

1.4.1 Theoretical Investigation on PAT

Williams [11] presented three examples of different types of PAT schemes of micro hydro power and advocating the use of induction generator and controller (IGC) design of PAT units for isolated micro-hydropower projects. The study illustrated many advantages by using PAT with induction generator. Single-phase induction motors can be used as stand-alone generators, but there may be problems in achieving excitation and in determining the size and arrangement of the capacitors required. However, the use of three-phase induction motor as a single phase generator is a good approach of providing a single-phase supply. This paper described range of PAT which covered the range of multi-jet Pelton turbines, cross flow turbines and small Francis turbines with the advantages of both practical and cost advantages over other types of turbine for medium head sites.

Derakhshan et al. [12] redesigned shapes of the blades using a gradient based optimization method involving incomplete sensitivities for radial turbo machinery developed by Derakhshan et al. [13] to obtain higher efficiency. The optimization was performed in two steps. The primal optimization results show that the torque, head and hydraulic efficiency was increased by 4.25%, 1.97%, and 2.2%, respectively and for final optimization these values was improved by 2.27%, 1.08%, and 1.17% respectively.

Derakhshan and Nourbakhsh [14] reported a theoretical analysis to calculate best efficiency point of an industrial centrifugal pump running as turbine based on ''area ratio" .In this method, turbine mode hydraulic components of pump were estimated by using geometric and hydraulic characteristics of pump in pumping mode. Hydraulic losses in volute and impeller, mechanical losses related to power losses, volumetric losses were taken into account to calculate the head, discharge and efficiency of PAT at BEP. Theoretical method predicted 1.1%, 4.7%, 5.25% and 2.1% lower discharge number, head number, power number and efficiency than experimental data at BEP, respectively. The derivation for finding out the relations for BEP of PAT was described in details in the paper. The final relation for turbine maximum efficiency is expressed as

$$\eta_t = \frac{P_{nt}}{\gamma\, Q_t H_t} = \frac{\gamma Q_t H_t - P_{vt} - P_{lt} - P_{et} - P_{it} - P_{mt}}{\gamma Q_t H_t} \tag{1.3}$$

Joshi et al. [15] explained the simple stand alone micro-hydro scheme which can be applicable to remote communities. A simple method for the prediction of PAT performance to aid in pump selection was described with help of case study of low head micro-hydro site, consisting of an unregulated PAT directly coupled to SEIG (self-excited induction generator) connected to local loads through inverter-distribution transformer. Using complete pump characteristics with different specific speeds by Swanson [16], PAT to pump operating ratios in a constant head mode were developed. After finding out the operating ratios corresponding to the specific speed of pump, required pump for the project is selected. After this, steady state analysis of a SEIG driven by an unregulated PAT was simulated using Chan's approach [17]. Variation in different quantities i.e. speed frequency, output power and efficiency with load variation was simulated. Also variation in generated voltage with excitation capacitance was done.

Derakhshan and Nourbakhsh [18] derived a new method to predict the BEP of a PAT based on pump's hydraulic specifications. Some correlations for low specific speed pumps (N_s< 60) were developed with same specific speeds and having different impeller's diameters. The value of h and q were predicted by this method which are in good coincidence with experimental data but more experimental data will improve accuracy of this method. Authors have also been discussed a comparison between the predicted values of h and q by different methods for finding out the BEP of PAT. But no single method was in closely coincidence with the experimental data throughout the whole range of specific speeds. A procedure was presented in this study to select a proper centrifugal PAT for a small hydro-site (only valid for specific speed≤ 150 (turbine)).

Ramos and Borga [19] carried out steady and transient regimes analysis based on suter parameters. This study intends to be a pragmatic tool for better understanding about the use of pumps in turbine mode. This analysis gave a more economic solution to recover part of dissipated energy, in point of view of reduced dimensions imposed for turbo-machine equipment. The study illustrated an advantageous solution of pumps in drinking water or sewage systems for recovering power. After analysis, it is concluded that the use of pumps as turbines can obtain a maximum relative efficiency up to 80%, depending on the type of the runner

Williams [20] presented a study on comparison of eight different PAT prediction methods [21-28]. The test results on 35 pumps of various types and sizes were used for the comparison. It is concluded after comparison that no method provided an accurate prediction for the all pumps. However, the method proposed by Sharma [26] falls somehow in acceptable limits. This study advocates for the testing of pump in order to be certain of its turbine characteristics before installation.

Ramos and Borga [29] described the use of pump as turbines as good alternative to dissipation of excess flow energy that, in normal conditions, would be lost. Analysis of steady state conditions was done based on suter parameters to prove that pump can be used in industrial process and production of renewable energy. Finally it was concluded that whatever the type of the motor or alternator adopted, PAT is a unconventional solution to energy production.

Mariano Arriaga [30] reported the status of Pico-hydro development in Lao PDR. This paper illustrated the concept of PAT as a viable technical, economical alternative and long-term reproducible system for communities where Pico-hydro propeller turbines were not sufficient and proper turbines were significantly more expensive. A 2 kW_{el} project in Xiagnabouli province was proposed as a high quality and cost-effective solution for rural electrification of community which was installed, commissioned, and maintained by local staff and villagers. Feasibility study, design and analysis of this project have also been discussed after considering power generation alternatives, sizing, asynchronous motor simulation, civil works, cost estimation, and social aspects.

Singh and Nestmann [31] investigated an optimization routine for the turbine-mode operation of radial flow centrifugal pumps to improve the accuracy, robustness and reliability of prediction and selection models for PATs, especially in the low specific speed range. Block diagram of the consolidated model for pumps as turbines with optimization routine is shown in Fig 1.2. The optimization routine has also been evaluated experimentally for three pumps and significant improvement in the accuracy of the turbine predictions with the errors falling within the ±4 % acceptance bands in full load zone. Study recommended giving special emphasis to the accurate predictions in no-load operating region of the PATs and validating the efficiency characteristics on a calibrated test rig, for realizing the complete value of the optimization routine.

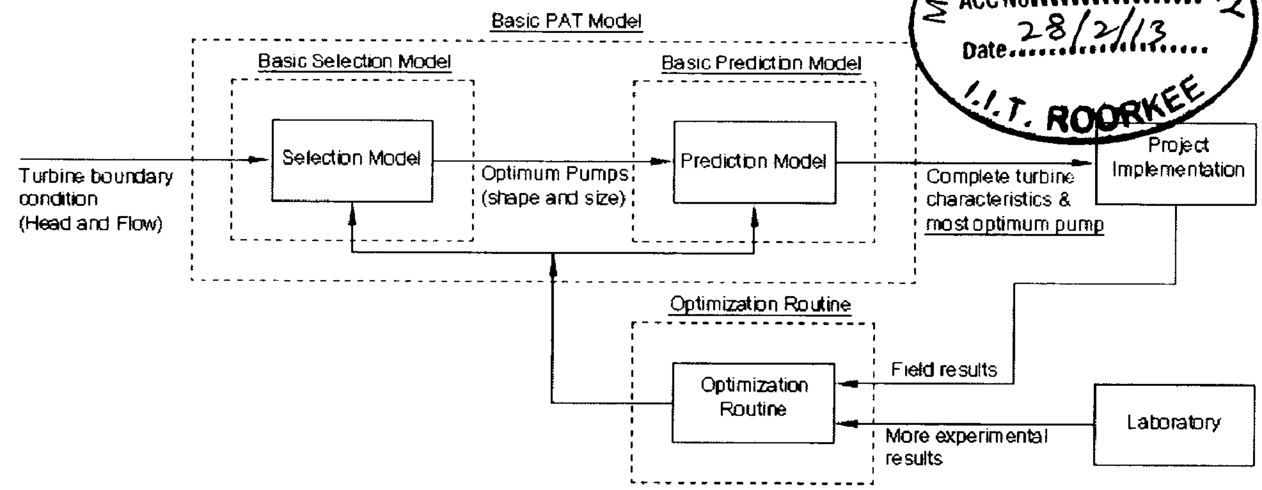

Fig.1.2 Block diagram of the consolidated model for pumps as turbines with optimization routine [31]

Raja and Piazza [32] carried out a study on the use of centrifugal pump in reverse mode as hydraulic power recovery turbine for sea water reverse osmosis (RO) systems. The plant size, power, cost per kWh and payout periods were taken into consideration for the cost analysis and it was found that by using of centrifugal pumps as HPRT, overall plant efficiency was increased as power consumption per unit volume of the produced water was reduced.

1.4.2 Experimental Investigation

Derakhshan and Nourbakhsh [14] carried out experimental investigation on a pump (specific speed 23.5) used in turbine mode. After measuring all parameters, PAT head, discharge, output power and efficiency were obtained and complete characteristic curves of PAT were achieved. A first-order uncertainty analysis is performed using constant odds combination method, based on a 95% confidence level. The uncertainty of head, flow rate, power and efficiency are ±5.5%, ±3.4%, ±5.1% and ±5.5%, respectively.

Fernandez et al. [33] described the behavior of a centrifugal pump in turbine mode at several rotating speeds with the help of experimental investigation in a hydraulic set-up. The results showed that the turbine characteristics can be predicted to some extent from the pump characteristics and the performance curves has also been obtained at a constant rotational speed and constant head. The constant speed curves showed that the net head was increased in pump mode and at low flow rates; the pump provides an inverse working area with power consumption and head dissipation. An increase of speed with the head was shown in constant head performance curves. They also studied the forces actuating on the impeller and found that forces measured in turbine mode are smaller than the forces measured in pump mode.

Joshi et al. [34] studied a simple approach to predict PAT performance to aid in pump selection, especially high specific speed pump, with a case study of a micro-hydro site producing 25kW electric power from 5.5 meters of gross head. Author formulated a relationship between pump and turbine specific speeds to assist in pump selection for a particular hydro site.

$$\frac{N_{st}}{N_{sp}} = \frac{N \times \left(\sqrt{P}/H^{5/4}\right)_t}{N \times \left(\sqrt{Q}/H^{3/4}\right)_p} \tag{1.4}$$

By using BEP values of pump and the above relation, the relationship between the PAT specific speed and pump specific speed was derived for pump selection purposes for equal rotation speeds.

Derakhshan and Nourbakhsh [18] presented some correlations to predict the best efficiency point of a pump working as a turbine, based on pump hydraulic characteristics using experimental data. Four industrial centrifugal pumps with specific speeds from 14 to 56 ($m,m^3/s$) were tested experimentally. At the same rotational speeds, a centrifugal pump in turbine mode needs higher flow rate and head than in pumping mode. Pumps with higher specific speeds have lower ratios of head and discharge. But variations of power ratio were not proportional to variations of pump's specific speed.

Suarda et al. [35] carried out experimental analysis by using two small pumps: diffuser-pump and volute-pump; to determine performances of pumps as turbines with output-powers and their efficiencies. The experiment showed that the efficiency of the volute-pump operating as turbine is slightly better than or at least equal to the pump efficiency and also slightly better than diffuser pumps. It was concluded that Centrifugal volute-pumps as hydro turbines were potential alternative solution and widely available in the market ranging from small to big size.

Nautiyal et al. [36] carried out an experimental investigation of a centrifugal pump having specific speed 18, to study its performance characteristics in pump and turbine mode operation. Experiment showed a satisfactory operation of a centrifugal pump as turbine without any mechanical and technical problem. It was found the pump operates at higher head and discharge values in turbine mode as compared to pump operation. However, the best efficiency in turbine mode was found 8.53% lower than best efficiency in pump mode. The experimental results of tested pump and other pumps studied by some previous researchers were used to develop new correlations to obtain turbine mode characteristics of pump from pump mode characteristics by using its best efficiency and specific speed in pump mode. Values obtained from the derived correlations were compared with experimental results and results of other methods which show very low deviation. The term 'χ' gives relation between best efficiency and specific speed in pump mode. The developed correlation for head and discharge ratio are as

$$\chi = \frac{\eta_p - 0.212}{ln(N_{sp})} \tag{1.5}$$

$$q = 30.303\chi - 3.424 \tag{1.6}$$

$$h = 41.667\chi - 5.042 \tag{1.7}$$

Singh and Nestmann [37] carried out experimental investigation to find out the effects of impeller rounding on a combination of radial flow and mixed flow PATs and to accurately characterize these effects with respect to internal hydraulic variables over the complete operating region of the PAT (part-load, BEP and overload). After application of developed model, Experimental results showed that system loss coefficient has reduced drastically for all the impeller rounded PATs in all the three operating regions and the exit relative flow angle has increased marginally for most of the PATs. Due to impeller rounding, positive impact on the overall efficiency in all operating region was found in range of 1–3%. But this model was not useful to explain the effect of impeller rounding on the two external PATs.

Singh and Nestmann [31] carried out experiment study for three pumps with specific speeds of 18.2 rpm, 19.7 rpm and 44.7 rpm to evaluate the optimization routine. Based on experimental results, significant improvement in the accuracy of the turbine predictions was found and for all the three pumps, errors were found within the ±4% acceptance bands in the full load operating region. But for no-load region, the errors were in the range of ±10–20% (except lower specific speed) which could still be viewed as an inadequacy. In order to make the model more stringent, it was proposed to reduce the acceptance criteria further to ±2% tolerance bands of the prediction errors. The paper also recommended extensive use of the optimization routine in PATs projects and developing a database of accurate field results to improve the routine further.

1.4.3 CFD Analysis of PAT

Derakhshan and Nourbakhsh [14] carried out simulation of centrifugal pump having specific speed of 23.5, in direct and reverse mode by CFD. For the verification of numerical results, simulated pump was tested as a turbine experimentally. CFD results were in good coincidence with experimental data for pump mode not only at best efficiency point but also in part-load and over-load zones. But for turbine mode, CFD results were not in acceptable coincidence as turbine head and power were lower than experimental data at same discharge. This study shows that CFD fails in the turbine boundary but the future works on CFD application can be improved.

Derakhshan et al. [12] carried out 3D flow simulation on impeller blades redesigned after the optimization process. The multiblock structured grids on the blades were prepared. Mass flow rate, velocity direction, turbulence kinetic energy (k), and turbulent dissipation (ε) were imposed at the inlet boundary, while at outlet boundary condition static pressure was prescribed. After that a periodic boundary condition was also applied between two blades. The study showed differences of less than 1% for efficiency and head in simulation results.

Natanasabapathi et al. [38] describes the investigation of Pump as Turbine using numerical approach. The blade profile was created using BladeGen software and surfaces of the blade were transferred to Pro-Engineer through IGES format. The domain of complete runner was created in Pro-engineer and transferred to CFX-5.6 Build for meshing and result analysis. The Head drop across the turbine was matching with the experimental results while there was a deviation in the efficiency computed from CFD at discharges away from BEP due to considerable amount of error at the Frozen rotor interface between casing and runner. In order to reduce the interface loss, two rings of structured grid of two-element thickness were introduced in between casing and runner. After that results were encouraging and the error was considerably reduced. So, it was suggested that use of a structured griding near the interface is the solution of unrealistic results like gain of total pressure where in reality loss is expected.

Rawal and Kashirsagar [39] carried out numerical analysis on axial pump in turbine mode results were compared with the experimental data. Satisfactory similarity between the experimental and numerical results was found for best efficiency point. Authors also stated that numerical model was quite helpful in investigation of various parameters like internal hydraulic losses i.e. losses in the impeller, the draft tube and the casing and flow pattern which could not be measured experimentally.

Nautiyal et al. [40] carried out a study on the application of CFD and its limitations for PAT using cases of previous researchers ([14], [38], [41]). Study reported that CFD analysis was an effective design tool for predicting the performance of centrifugal pumps in turbine mode and to identify the losses in turbo machine components like draft tube, impeller, casing, But there was still deviation in results of experiment and CFD of turbine mode operation of pump. It was suggested that by using finer mesh, numerical methods and turbulences models, differences in results of experimental study and CFD can be minimized.

Barrio et al. [42] carried out a numerical investigation on the unsteady flow in commercial centrifugal pump operating in direct and reverse mode with the help of commercial code FLUENT which was used to solve the full unsteady Reynolds averaged Navier-Stokes equations together with the standard k- ε model plus standard wall functions to simulate turbulent effects. Results of numerical simulation were in good agreement with the experimental results. This numerical model was used to estimate the radial load on the impeller, as a function of flow rate, for both modes of operation by means of a full integration of the instantaneous pressure and shear stress distribution on all the impeller surfaces in each of the time steps. The study revealed that in the reverse mode, the flow only matched the geometry of the impeller at nominal conditions while developing re-circulating regions of fluid at low flow rates (near the pressure side of the blades) and high flow rates (near the suction side). Study showed that total radial load (steady and unsteady components) in turbine mode was lower than the maximum total load in pump mode for operating below turbine rated conditions and total thrust on impeller would be higher than maximum load for higher turbine flow rates.

Fecarotta et al. [43] investigated minimum complexity of the CFD calculation mesh, in order to perform faster and reliable simulations. The study described the CFD analysis as reliable tool to better understand the interaction between the hydro mechanical equipment and the flow behavior in spite of difficult calculation. CFD calculations were carried out to predict the turbine behavior under different flow conditions (steady and transient) and the performance curves for both modes have been obtained. The analysis showed that transient calculations are good for the simulation of the real behavior of the machine as it includes interactions between the rotor, the stator and the hydrodynamic conditions.

Sedlar et al. [44] carried out CFD analysis of flow in the middle stage of the radial flow multistage pump having specific speed 23, operating in turbine regime. The ANSYS CFX commercial CFD system was used to solve the fully unsteady three-dimensional Reynolds-averaged Navier-Stokes equations together with the Menter's SST model of turbulence. For the estimation of the pump (or turbine) head and hydraulic efficiency as well as for the visualization of flow phenomena in the middle stage, two complete stages were analyzed with fully unsteady simulation to avoid the influence of boundary conditions and the position of rotor and stator parts on the results. The calculated numerical results were compared with the experimental results. Analysis showed that the hydraulic efficiency of the multistage pump in the reverse mode can be quite high, even without any expensive corrections of the manufactured parts. Full scale impeller in the reverse mode provided the flow regimes close to the optimal rate of flow without large separations while optimal rate of flow was shifted to left with reduced impeller diameter and hydraulic efficiency will be decreased with this modification.

1.4.4 Modification of Pump running in Turbine Mode

Derakhshan et al. [12] redesigned shapes of the blades using a gradient based optimization method involving incomplete sensitivities for radial turbo machinery developed by Derakhshan et al. [14] to obtain higher efficiency and optimized impeller was modified by rounding of leading edges and hub/shroud inlet edges in turbine mode . After each modification, a new impeller was manufactured and tested. Experimental results showed improvement of efficiency in all flow rates of part load and overload zones. An increase of -2.2%, +9.4%, +14.8%, and +2.9% for discharge, head, power and efficiency respectively was found for optimized impeller and blade behavior was improved for a wider operating range. But the head was increased slightly more than numerical optimization data. Rounding of optimized impeller improved these values to +5.5%, 11.5%, 36.1%, and 5.5% for flow rate, head, power, and efficiency respectively. Author concluded that by doing easy modifications on the impeller of the Pump used in turbine mode, its maximum efficiency was increased.

In order to prevent excessive turbulence for efficiency consideration, Suarda et al. [45] provided the modification on impeller by grinding the inlet ends of the impeller tips of centrifugal volute type pump to a bullet-nose shape. After modification, this pump was tested experimentally in turbine mode at the maximum head of the pump, 13 meter, and at various capacities. The results showed that the output-power and efficiency of the modified centrifugal volute-pump in turbine mode was slightly better than before modified one. But the volute-pump as turbine for bigger size of pump was recommended to be modified their impeller tips for better performance because in smaller pumps the effect of modification on their performance was not significant and a lot hard work for modification was required.

Singh [46] discussed various possibilities of modifying the pump geometry to improve the performance of a given pump in turbine mode. The study showed that impeller rounding was the most beneficial modification out of different geometric modifications. In the study, impeller rounding on eight different centrifugal pumps was carried out. An qualitative understanding of impeller rounding effects with respect to the internal hydraulics was presented. The study showed that non-modified and impeller rounded characteristics were very close to each other and the predicted curve was offset by a margin of over 3–4%. The study concluded that it was significant to have accurate prediction models for different pump shapes for hydraulic optimization due to impeller rounding.

Pandey [47] carried out experimental study to predict the performance characteristics of modified centrifugal pump. The modification at runner vane shape was provided to reduce the shock losses at inlet and the arrangement for movable guide vane was made to improve part load efficiency. Study showed that peak efficiency was improved by 4.5% and part load efficiency has also improved.

Saini and Ahmad [48] carried out experiments on pumps in turbine mode having specific speed range from 10 to 300 and developed a monogram as shown in Fig 1.3; to aid in the selection of a pump to be used as turbine for a particular site. This method reduced the calculation effort involved in the selection as was proposed by early investigators.

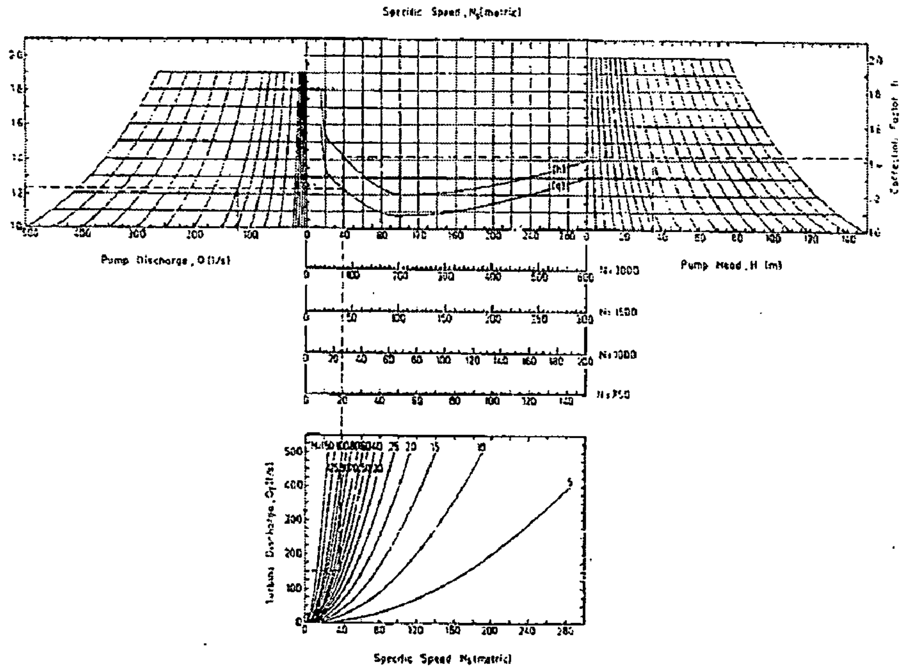

Fig. 1.3 Monogram for selection of PAT [48]

1.5 OBJECTIVE OF DISSERTATION

Literature survey discussed above shows that that different aspects of PAT has been reviewed by different researchers till now such as its performance, selection, testing etc. However, due to some disadvantage, PAT has not become popular. The main disadvantage of PAT is its part load efficiency which is found to be very poor. There is no flow control mechanism available in pump as with conventional turbine. During the variation in seasonal flow, PAT work under poor part load efficiency. In order to improve the part load efficiency, attempts have been made for modification in pumps in earlier studies. Recently an attempt was made to modify a pump by providing with the guide vanes. The modified pump in turbine mode was tested experimentally. Objective of present study is to carry out CFD analysis for a modified pump used in turbine mode with the following objectives and compare the results with available experimental results:

i. To identify the system and operating parameter for modified pump used as turbine.
ii. To develop the model for CFD analysis.

iii. To analyze the performance of modified Pump as Turbine under different condition.

iv. To validate the simulation results with experimental results available [47]

1.6 ORGANIZATION OF DISSERTATION WORK

Chapter 1 presents an introduction about the energy scenario, importance of the micro hydropower in the present energy context. It also describes the objective of the present study. A summary of work carried out by different authors, their objectives and conclusions are presented in this chapter. It also describes the objective of the present study.

Chapter 2 consists of brief discussion about PAT which includes historical background, concept of PAT, advantages, disadvantages and applications of PAT. Prediction of PAT performance and selection of appropriate PAT for given site condition have been discussed.

Chapter 3 gives the identified parameters of selected centrifugal pump for the CFD analysis. Details of designing parameters of modified pump used as turbine for computational study have been discussed.

Chapter 4 details the modelling of each part of modified PAT i.e. casing, impeller, draft tube, guide vanes. Modelling of complete assembly of modified PAT and meshing for complete set up have been discussed. Detailed procedures adopted for the simulation are explained in this chapter.

Chapter 5 details the results obtained from the computational study of modified PAT. The comparison of CFD results with the experimental study has also been discussed.

Chapter 6 presents the conclusion of present work and recommendations for future work.

CHAPTER 2

PUMP AS TURBINE

2.1 GENERAL

Pumps are designed as fluid movers. However, pumps have also been used for many years to generate electrical power or to provide rotating shaft energy. For example, a centrifugal pump can serve as a prime mover to drive a fan, another pump, a compressor or a generator where high pressure liquid is available as a source of water. In application of pumps used as a fluid mover, the liquid enters the suction part at low pressure, absorbs shaft works in the impeller, and leaves the discharge nozzle at higher pressure. When a pump is used as a turbine, the liquid enters at high pressure, drives the impeller in reserve, and leaves at a lower pressure. A basic difference between conventional pumps and hydraulic turbines is that pumps do not have flow control device. Pumps are usually designed for a specific total dynamic head and discharge requirement. Thus, PAT are best suited to constant flow applications.

Because pumps are designed as fluid movers, they may be less efficient as hydraulic turbines than equipment expressly designed for that purpose. However, using pump as a turbine can be cost effective in many situations. Among the more obvious reasons to use a pump as turbine are easy and cheap installation and readily available [49].

The second problem associated with the use of pump as turbine is the difficulty of finding the turbine head and discharge, which are different than the rated pump head and flow, and are needed to select the correct pump for particular site parameters.

There is no question as to whether a pump can operate as a power-generating unit. The concerns are more related to equipment application, efficiencies, operating range, cavitation characteristics, costs, etc. This chapter will discuss these and other factors.

2.2 HISTORICAL BACKGROUND OF PUMP AS TURBINE

When pumps were first used in turbine mode is not clear but the use of pumps as turbines has been a research topic for over 70 years. Its journey began when Thoma [50] and his engineers accidentally found that pumps operated very efficiently in turbine mode, when they were trying to evaluate the complete characteristics of pumps.

The turbine mode operation became an important research question to many manufacturers pumps were prone to abnormal operating conditions. Knapp (1937) later published the complete pump characteristics for few pump designs based on experimental investigation.

In the 1950s and 1960s, the concept of pumped storage power plants was evolved mainly in developed countries to manage the peak power requirements. 'Pumps as Turbines' hence found one of its important applications and this sustained the research to a certain extent. However the capacity of such plants was in the order of 50 MW to 100 MW and the numbers of them were few. In later years, chemical industries were the area for the application of PAT for energy recovery. These manufacturers have developed empirical methods for determining performance in the normal turbine mode by reference to the known performance of the unit as a pump [51].

In certain chemical processes it was necessary to dissipate the energy of high-pressure fluids through small pipe lengths. Instead of simply throttling, PATs were installed to recovery some energy. Even in water supply networks identical applications of this technology were found.

2.3 CONCEPT OF PAT

A liquid pump is very similar to a hydraulic turbine except that it operates in reverse mode. Mechanical work, usually supplied by an electric motor or other prime mover, is used to add energy to the fluid being handled. The mechanism for transferring the energy is increasing the velocity of fluid as it passes through the impeller flow passages. After passing through impeller, the fluid is moving at a high velocity so that the energy is in the form of kinetic energy. A diffuser is used to convert a portion of this kinetic energy to pressure head by increasing the flow pressure area. A hydraulic turbine is used to extract energy from the fluid. A pressure head of fluid is partially converted to kinetic energy by allowing the fluid to flow through a volute (spiral casing) which acts as a nozzle to convert some of the pressure energy in to velocity. The turbine runner extracts kinetic energy from the fluid and by letting the fluid accelerate in the runner passage so that the reaction from acceleration produces torque on the shaft. Fig 2.1 and 2.2 show pumps as fluid mover and hydraulic turbine respectively.

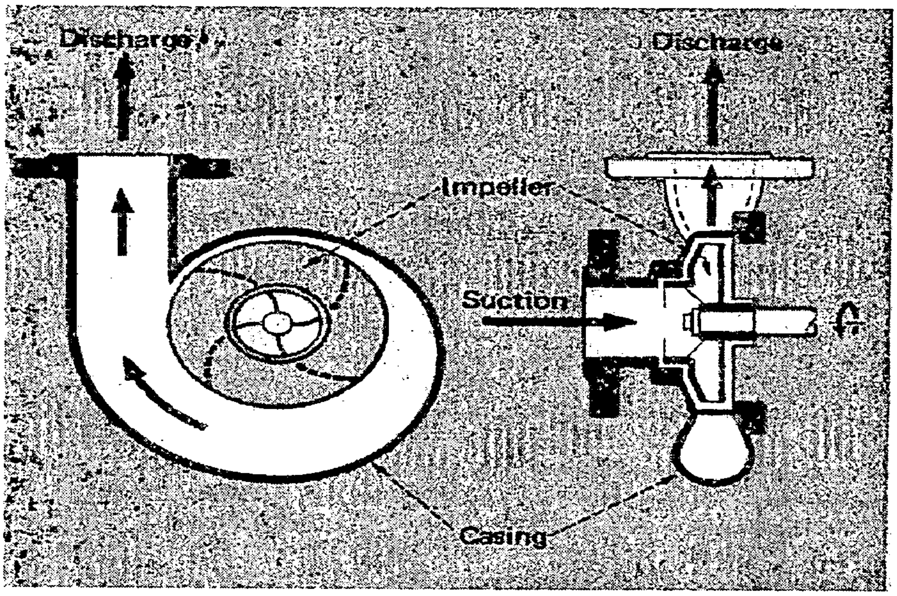

Fig. 2.1 Radial flow pump as fluid mover [52]

The performance of a pump will have different best efficiency point (BEP) flow parameters when operating as a turbine. This is true because the energy for losses due to friction, etc. must be derived from the flowing fluid in a turbine. In a pump, the energy losses are included in the mechanical energy supplied to the pump drive shaft and are not transmitted to the fluid. Therefore, for a machine operating at a particular speed, the flow and head will be less when pumping than in the turbine mode [53].

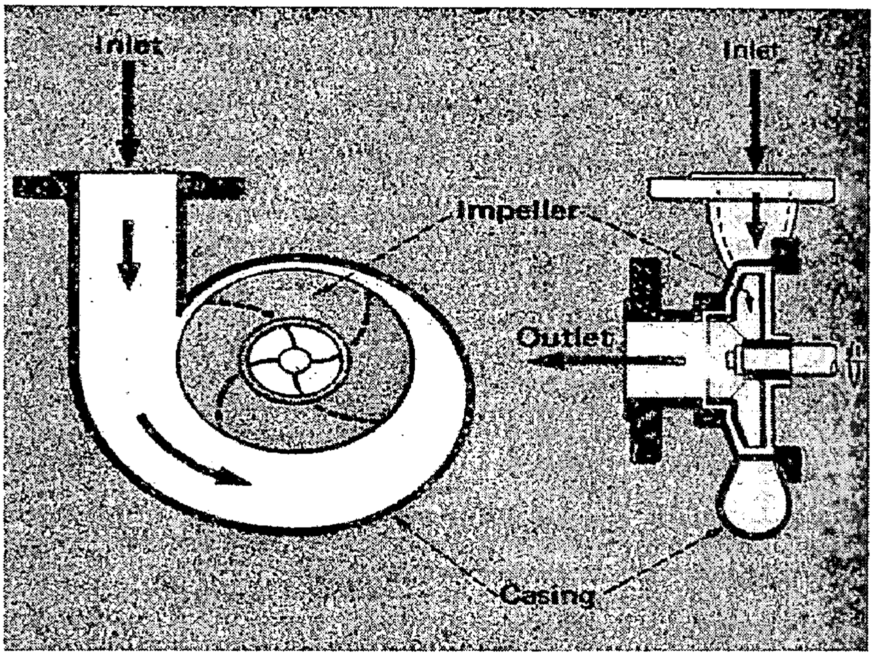

Fig. 2.2 Radial flow pump as hydraulic turbine [52]

The concept of PAT lies in the fact that apart from not having a flow control device, centrifugal pumps are physically and hydraulically similar to Francis turbines. Only the difference lies in their function. Centrifugal pump converts the mechanical energy of impeller into pressure energy and some kinetic energy of water whereas Francis turbine converts pressure energy and kinetic energy of water into mechanical energy of runner. Therefore if a centrifugal pump is operated in reverse mode, i.e. if the water outlet section of a centrifugal pump is used for entering water at high pressure and if the water inlet section of a pump is used for exiting water, it can function as a Francis turbine. The water, at high pressure entering in the centrifugal pump passes through the impeller and rotates the impeller and exits from the eye of the impeller.

2.3.1 Why use Pump as Turbine [54]

Standard pump units when operated in reverse mode have a number of advantages over conventional turbines for micro- hydro power generation. Pumps are mass produced, and as a result, have advantages for micro- hydro compared with the purpose-made turbines. There are several reasons for using pumps in place of turbine. Most of these reasons relate to money, either in reduced capital investments or in reduced time required for construction and maintenance. The main advantages are as follows:

- Available for a wide range of heads and flows
- Available in large number of standard sizes
- Low cost
- Spare parts such as seals and bearings are easily available. Thousands of pump dealers, suppliers, and service organization are now in existence. Maintenance and repair service for pumps can usually be performed by local shops, thereby drastically reducing transportation cost and down time
- Easy installation- uses standard pipe fittings
- Integral pump and motor can be used as a turbine and generator set
- Short delivery time as pump of popular size and types are stocked by dealers and can be delivered in 3 to 5 days.

There are several practical benefits of using a direct drive pump as turbine i.e. one in which the pump shaft is connected directly to generator. One of the advantages of using a PAT instead of a conventional turbine is opportunity to avoid a belt drive .however, in some circumstances there are advantages to fitting a belt drive to a PAT. The advantages of using a direct drive arrangement are:

- Less maintenance- no need to adjust belt tension or replace belt
- There are fewer losses in drive, which saves up to 5% of output power:
- Longer bearing life – no sideways forces on bearing
- PAT and generator come as one unit so installation becomes easy.
- Lower cost – no pulleys and smaller base plate.

2.3.2 Disadvantages of Pump as Turbine

The main limitation is that the range of flow rates over which a particular unit can operate is much less than that of a conventional turbine. The main disadvantages of PAT are as follows:

- The major problem associated with the PAT is the difficulty of finding the turbine head and discharge, which are different than the rated pump head and flow, and are needed to select the correct pump for a particular site.
- PAT has lower efficiency at part load. A conventional turbine has an effective hydraulic control (adjustable guide vanes, nozzles or runner blades) to adjust the machine to the available flow or the required output
- Unlike turbines, pumps have no hydraulic control device, therefore a control valve must be incorporated in the penstock line (additional costs) to start and stop the PAT. If the valve is used to accommodate the seasonal variations of flow, the hydraulic losses of the installation will increase sharply.

Limitations of using direct drive pump are [54]:

- Limited choice of generators available for a particular PAT.
- Turbine speed is fixed to speed of generator – thus reducing the range of flow rates when matching the PAT performance to the site condition.

2.3.3 Applications for Pump as Turbine

Common applications for pump as turbine are given below:

i. **Village scheme, mainly for household lighting**

In many village schemes in developing countries, where the main electric load is evening lighting, a PAT is suitable. During the daytime the generator can also be used to power equipment that will benefit the economy of the village e.g. a circular saw, crop dryer or sewing machines. In this type of scheme PAT will be designed to run using the flow available at the driest time of the year [55].

ii. **Electricity for remote farms**

PAT has been successfully used for supplying electricity to remote farms in UK. Since the output of generator is limited, some care has to be taken in switching on application, not to overload the system. Spare power, not used in general appliances, can be used for background heating [55].

iii. **Battery charging and other intermittent load applications**

A PAT scheme could be used to provide power for battery charging more locally. In this case, it is not essential that the power is available continuously, and it is possible to run a PAT intermittently. Other applications for an intermittently operating PAT are for running refrigerators for ice making or vaccine storage, or for running electrically heated crop dryers [55].

iv. **Water pumping**

PAT can also be used for pumping water for domestic use, where a farm or village is situated above the main stream level. In this case the PAT is connected directly to a high head centrifugal or positive displacement pump which pumps a small quantity of water to a high head [55].

2.4 PREDICTION OF PAT PERFORMANCE

The performance in turbine mode of a pump will be different from performance of pump mode. For the ideal condition of machine design parameters, head and flow remains same in both pump and turbine mode. However losses will be different in turbine mode due to following:

2.4.1 Geometry of the Pump

At the entrance and exit of the impeller velocity triangles can be drawn by knowing the blade angles at inlet and exit. Head can be determined by the inlet velocity triangle. For the optimum condition of operation in pump mode is reached at a flow which corresponds to whirl free inlet condition. Due to circulation loss the impeller out let velocity of fluid is slightly deflected and leaves the Impeller not at the exact blade angle assume for infinite number of blade but at a fluid angle β_2. This reduces the generated pump head. When pump operates in reverse mode, its performance is determined by the inlet velocity triangle. The circulation loss occurs at inner periphery of impeller is practically negligible due to smaller diameter. The friction less performance in turbine mode corresponds to the ideal Euler's condition and head and flow at best efficiency point will be greater than in pump mode.

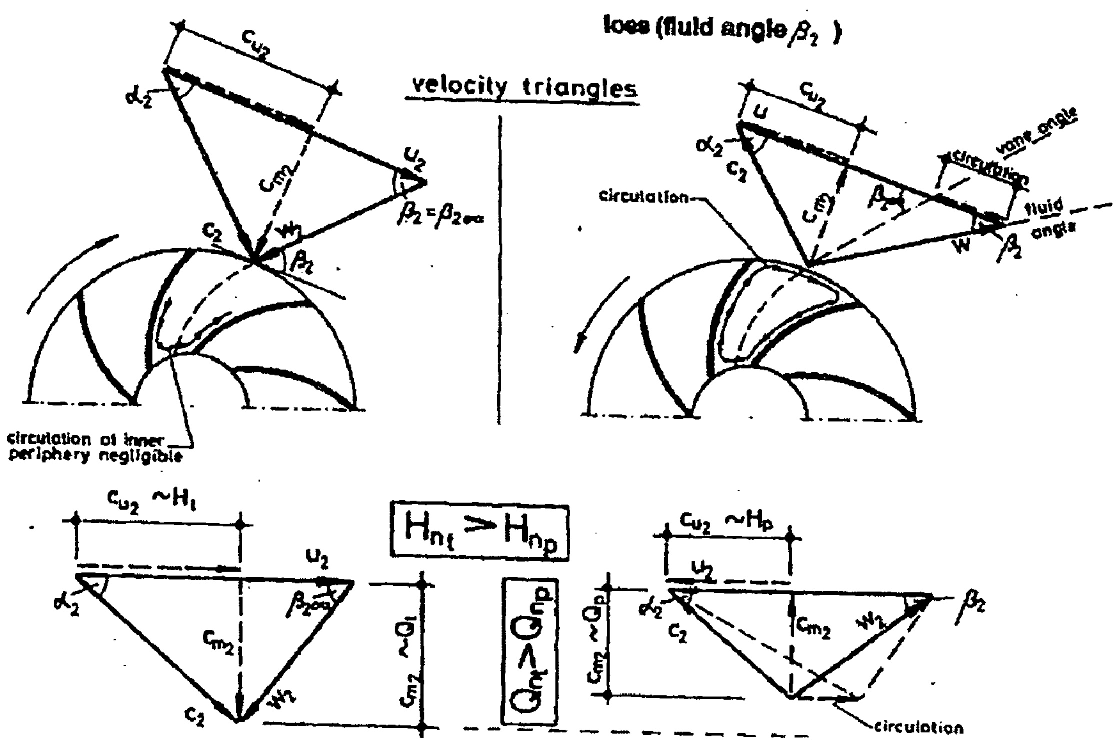

Fig. 2.3 Performance difference between pump and turbine mode including circulation losses [56]

2.4.2 Hydraulic Losses

When fluid passes through the impeller, it is subjected to friction and shock losses. Due to these losses ideal energy transfer from the impeller to fluid as expressed by the Euler's equation is not achieved. Total head generated by the pump is always lower than the Ideal head. This reduction of head is called as hydraulic efficiency of pump. It reverse mode, PAT operates at optimum flow conditions, an increased pressure must act on the PAT. Therefore friction and shock losses must added to the ideal head given by the Euler's equation. Other losses in pump consist of leakage of fluid from high pressure side to low pressure side reducing the total flow pumped. These losses are called volumetric loss and efficiency corresponding to these losses are called volumetric efficiency. Similar losses are appears in turbine mode. Fig 2.4 shows performance difference between pump and turbine mode including hydraulic losses.

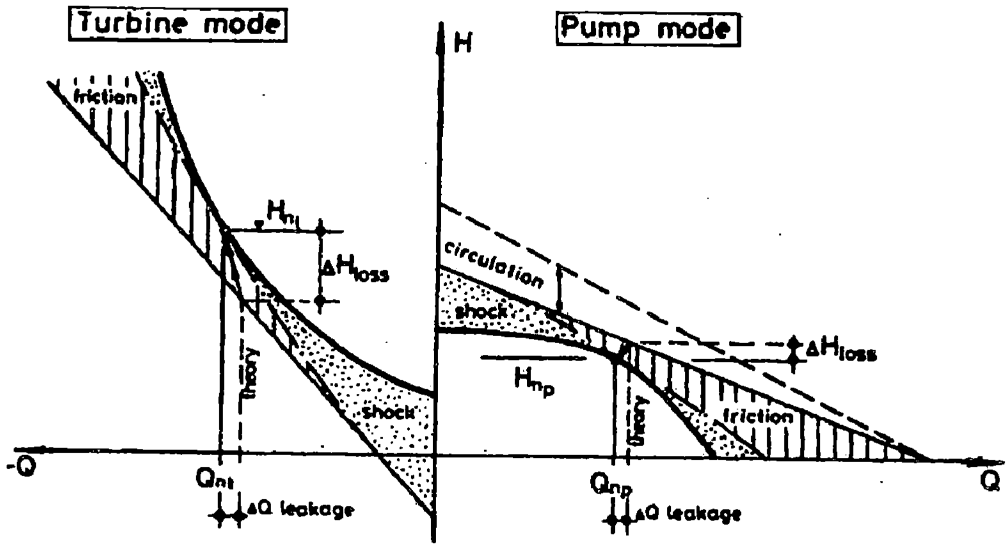

Fig. 2.4 Performance difference between pump and turbine mode including hydraulic losses [56]

In Fig. 2.5, the differing velocity triangles are shown for the best efficiency points in pumping mode and in turbine mode of operation. At the outer diameter (location 2), the absolute flow angle, α_3, is approximately the same for both modes of operation in respect to the shape of the volute. While the flow in pumping mode leaves the impeller with a relative flow angle, β_{3p}, which is smaller than the blade angle, β_2, (effect of angle deviation), the relative flow angle in turbine mode should be as close as possible to the blade angle to avoid shock losses, i.e. $\beta_2 = \beta_{3t}$ .To accomplish this, the radial velocity c_{3r} must be higher in turbine mode than in pumping mode; thus also the flow rate.

At the inner diameter (location *1*), the absolute flow angle, α_0, is approximately the same (90°) for pumping and turbine mode from the condition of no pre-rotation and no-swirl outflow, respectively. Pump impellers are normally designed to have a blade angle, β_1, which is somewhat larger than the relative flow angle, β_{0P}, at best efficiency condition to improve the suction behavior. In turbine mode, the relative flow angle, β_{0T}, is larger than the blade angle, β_1, caused by the effect of angle deviation at outlet. On the other hand, the flow rate in turbine mode should not be much higher than the flow rate in pumping mode, because a swirl will occur at the outlet, which will decrease the efficiency [56].

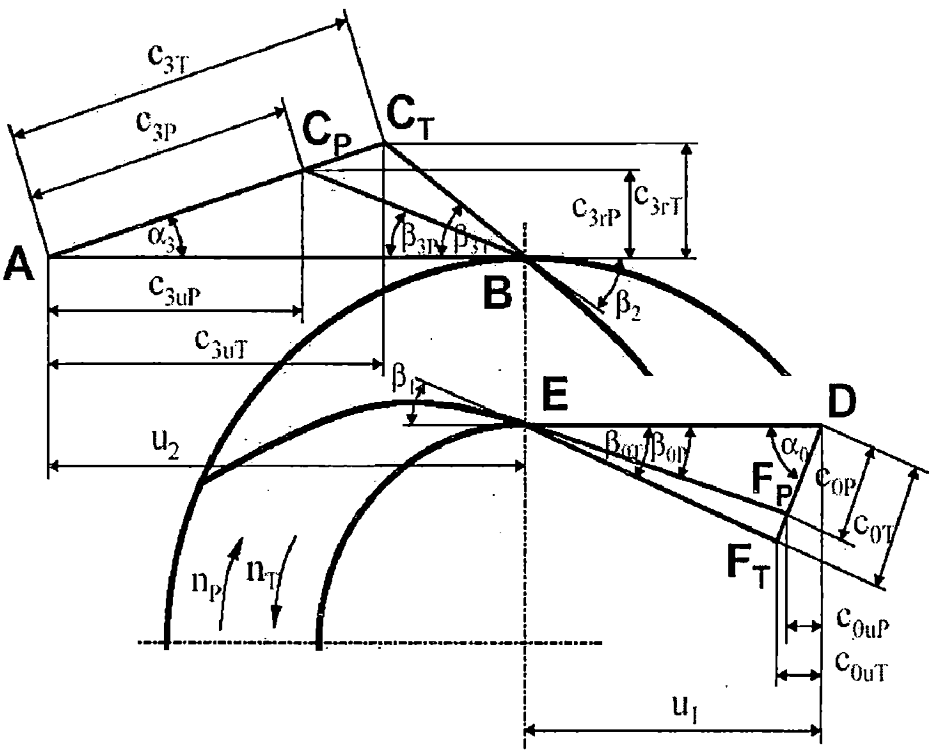

Fig. 2.5 Velocity triangles for best efficiency point in pump & turbine mode [56]

From these reasons, a pump in reverse mode has its best efficiency point at higher flow rates compared to the pumping mode. This can also explain why the efficiency remains nearly constant or – especially for low specific speed pumps – is even higher in the turbine mode than in the pumping mode. The relative influence of the secondary losses (leakage and disc friction losses) on the total efficiency is lower due to the higher shaft power in turbine mode [56].

2.5 SELECTION OF PUMP AS TURBINE

In the most common situation, there is a liquid stream with fixed head and flow rate, and an application that requires a fixed rpm; these are the turbine design conditions. We want to select a pump with a turbine BEP at these conditions. With performance curves, turbine design conditions will be converted to pump design conditions. Then, the pump is selected from a manufacturer's catalog.

The most common error in pump selection is using the turbine design conditions in choosing a pump from a catalog. Because catalog performance curves describe pump duty, not turbine duty, the result is an oversized unit that fails to work properly.

Instead of performance curves, we can use a manufacturer's conversion factors that relate turbine BEP performance with pump BEP performance. Variation of conversion factors for discharge and head depends on specific speed of pump.

2.5.1 Method for Selection of Pump as Turbine

The following procedure [57] may be adopted for selection of a pump to be used as turbine for a given site data:

i. Arrive at the values of rated head (H_t in m) and rated discharge (Q_t in m^3/s) from a given site data.

ii. The above data yields the value of available hydraulic power $P_t = QgH$ in kW.

   Keeping this in view, select the value of speed of generator and also the speed of pump.

iii. Find the value of specific speed for turbine data by using the following formula:

$$N_{st} = \frac{N_t \sqrt{Q_t}}{H_t^{3/4}} \tag{2.1}$$

iv. Examine whether it is desirable to use double suction pump. This should be done wherever possible because shaft of double suction pump experiences negligible thrust. Neglecting the effect of efficiency and assuming $\eta_t = \eta_p$ we get:

   - $N_{sp} = N_{st}$ for single suction pump (2.2)

- $N_{sp} = \frac{N_{st}}{\sqrt{2}}$ for double suction pump (2.3)

v. Find the value of conversion factors from test theoretical curves, as shown in Fig. 2.6, Fig. 2.7 & Fig. 2.8 and test curves, as shown in Fig. 2.9 and find the values of Q_p and H_p.

vi. Find from pump catalogues whether pump for chosen values of Q_p, H_p and η_p is available. If not select suitable pump speed for which a pump is available to match desired Q_p and H_p. Find new specific speed and check the values of conversion factors for the same. With little trial and error, it will be possible to select a pump.

vii. In all cases recheck all your values for selected pump and find out the output power expected.

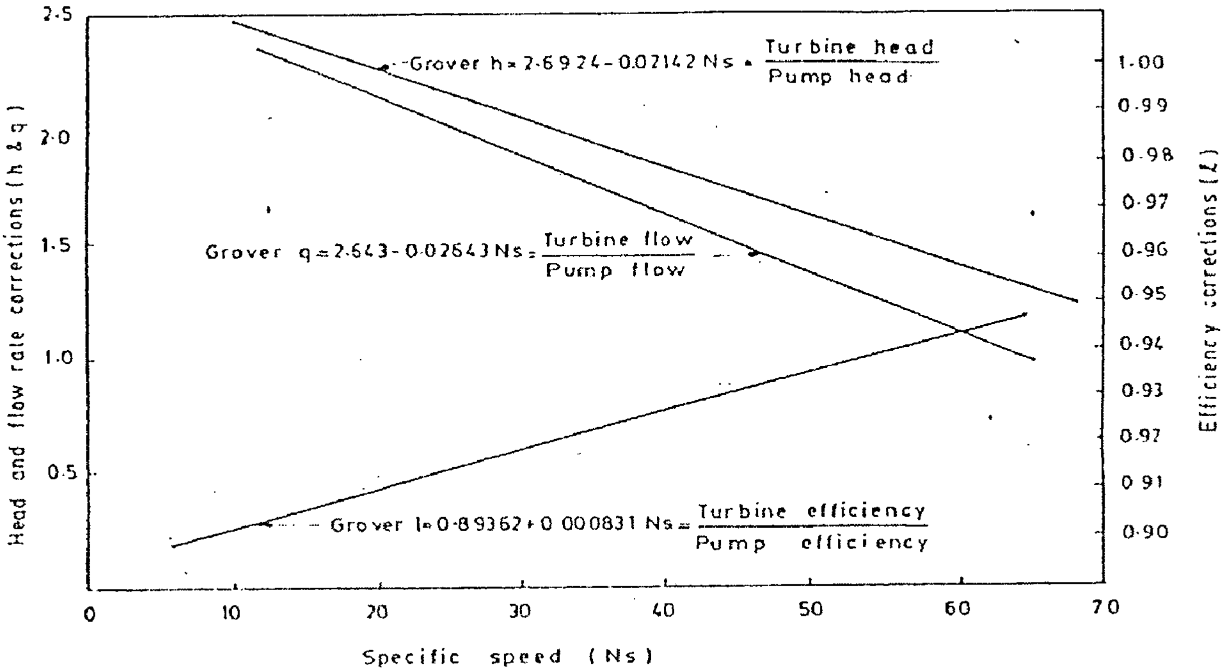

Fig. 2.6 Conversion coefficients for turbine pump parameter transformations as a function of specific speed [57]

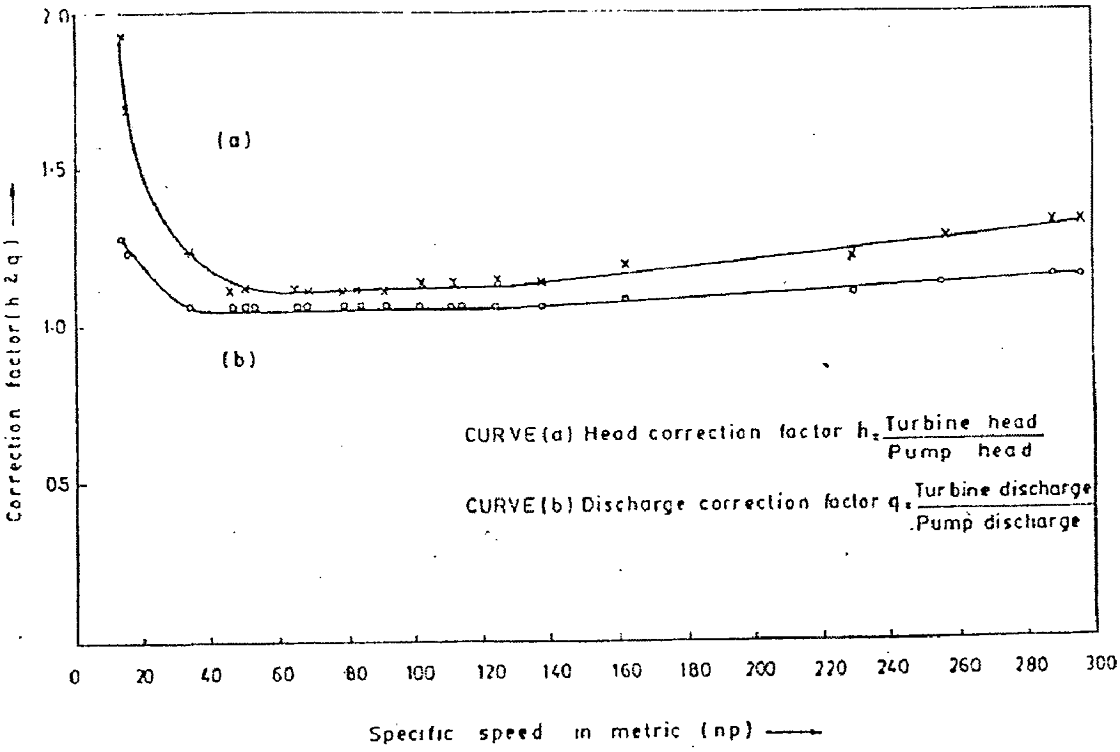

Fig. 2.7 Head and discharge correction factor by Stepanoff equations [57]

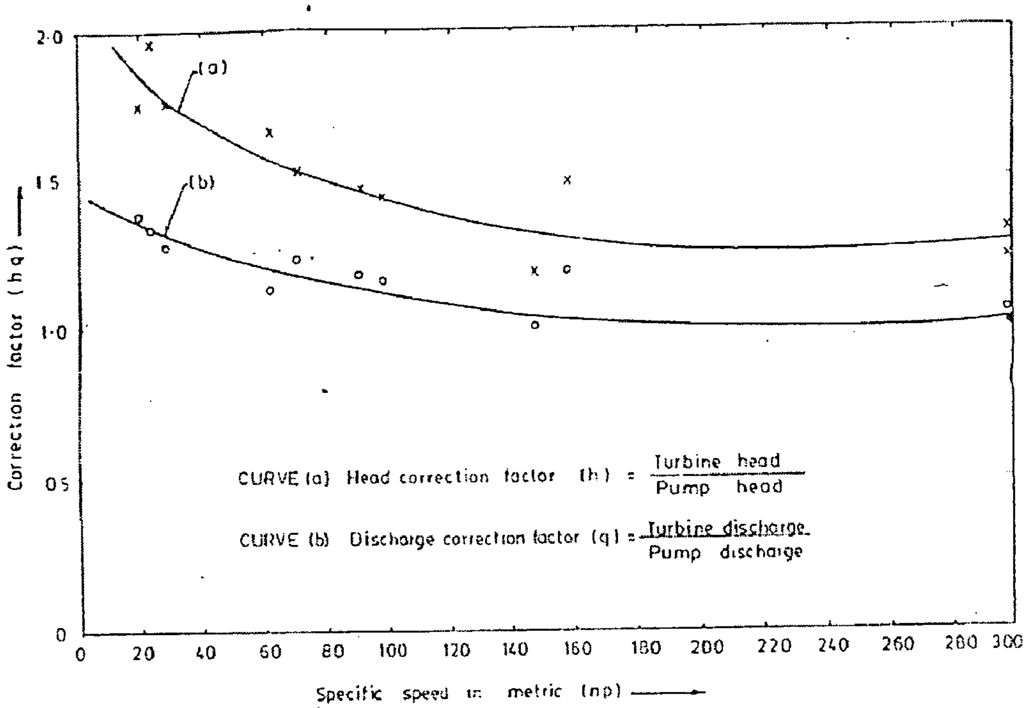

Fig. 2.8 Head and discharge correction factor theoretically calculated [57]

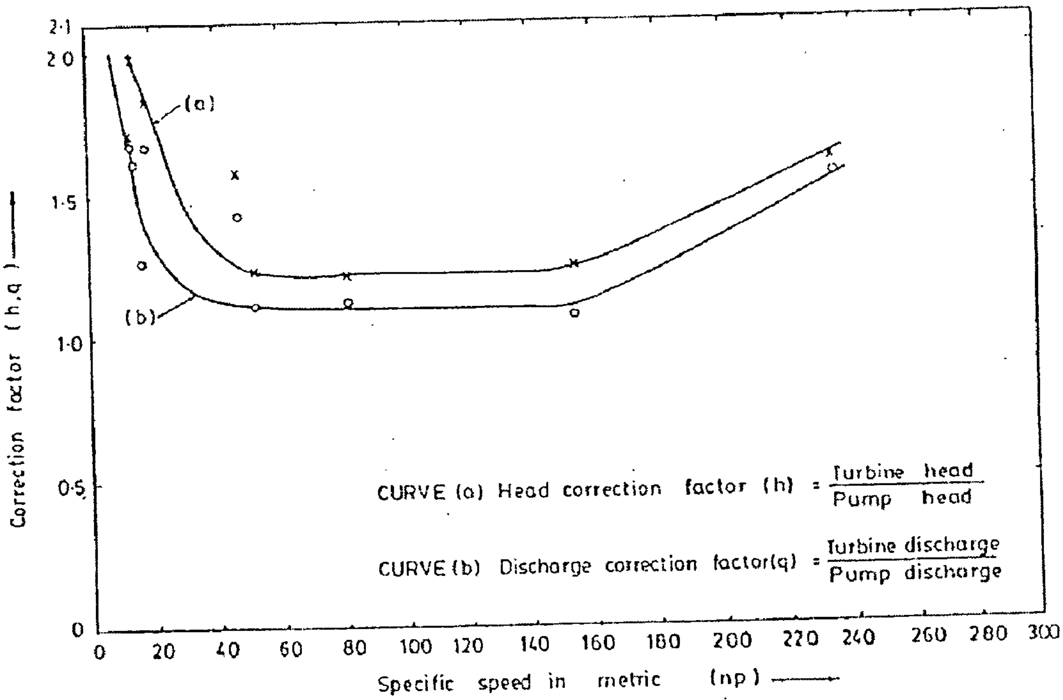

Fig. 2.9 Head and discharge correction factor from test data [57]

CHAPTER 3

IDENTIFICATION OF DESIGNING PARAMETERS OF PUMP

3.1 GENERAL

Application of commercial available centrifugal pump in turbine mode has been reported for micro hydro power development. The major problem with the pump as turbine is its poor part load efficiency. In order to improve the part load efficiency, movable guide vanes are proposed to be fitted around the pump impeller of centrifugal pump. This chapter describe in detail the steps involved in identifying the system and operating parameter for modified pump used as turbine used for modeling and investigation purpose.

3.2 SELECTION OF CENTRIFUGAL PUMP USED IN TURBINE MODE

To carry out the proposed work, it is required to identify the system (centrifugal pump) and its operating and designing parameters used for modeling purpose. A centrifugal pump NW10E of mixed flow type made of Kirloskar Brothers Ltd., Dewas is selected to carried out the present work. This Pump was used in experimental study to predict the performance characteristics of modified centrifugal pump [47]. In that experimental study this centrifugal pump was modified provided with guide vanes to improve part load efficiency. The salient features of selected centrifugal pump are given in Table.3.1

Table 3.1: Pump specifications

S.No.	Particulars	Detail
1.	Make	Kirloskar, Dewas
2	Type	NW10E
3.	Size	125 mm ×125 mm
4.	Impeller Diameter	260 mm
5.	Head	21.5 m
6.	Capacity	42.5 lps
7.	BHP	15.7 HP (11.71 kW)
8.	RPM	1500 rpm
9.	Peak efficiency	78 %
10.	No. of impeller vanes	7 Nos.

3.3 STUDY OF DESIGN PARAMETERS OF THE SELECTED PUMP

Based on available specifications, detail dimensioning of pump can be carried out. For the present study, there are following components for which designing parameters are required.

a. Casing
b. Impeller
c. Draft tube
d. Guide vanes

3.3.1 Casing

The casing of centrifugal pump under consideration is volute type which surrounds the impeller. It is of spiral type in which Cross sectional area increases from inlet to outlet of pump (in case of turbine area of flow reduces). Standard IS: 12800 (Part-I) is used to calculate major dimension of spiral casing [59]. Casing of pump is shown in Fig 3.1.

Fig. 3.1 Casing of pump

As outlet diameter of impeller is 260 mm, dimensions of spiral casing are shown in Fig 3.2. All the dimension of spiral casing corresponding to specific speed of 30.97 were calculated and are given as:

A	=	260 ×	0.49	=	127.4 mm
B	=	260 ×	0.88	=	229 mm
C	=	260 ×	0.71	=	184 mm
D	=	260 ×	0.81	=	210 mm
E	=	260 ×	0.58	=	151 mm
F	=	260 ×	0.91	=	237 mm
G	=	260 ×	0.76	=	197 mm
H	=	260 ×	0.65	=	170 mm
I	=	260×	0.85	=	220 mm
J	=	260×	1.37	=	356 mm

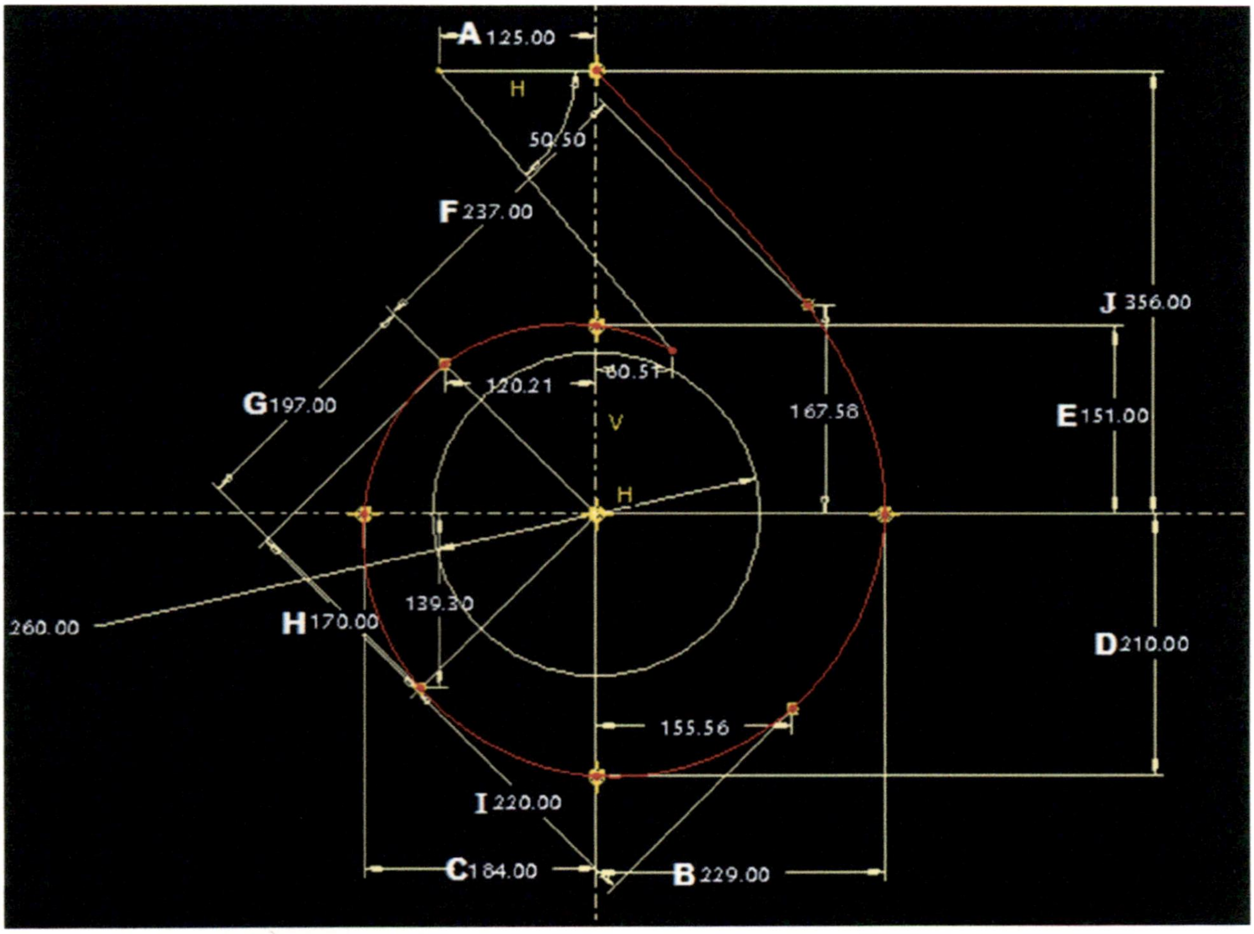

Fig.3.2 Dimensioning of casing

3.3.2 Impeller

Impeller rotating part of centrifugal pump consists of a series of back ward curved vanes. It is made of bronze for small capacity and for the large capacity it is made of cast steel or stainless cast steel. Impeller of pump is shown in Fig 3.3.

Fig. 3.3 Impeller of pump

Impeller is designed on the basis of design flow rate, pump head and pump specific speed. The design data for design of impeller is as follow:

- Flow rate Q= 42.5 lps = 0.0425 m^3/s
- Pump head H = 21.5 m
- Pump speed N= 1500 RPM
- Peak efficiency of pump η =78%

Referring the study of Khin Cho Thin et al.[58], design of the impeller of centrifugal pump will be done. The specific speed of pump is calculated.

$$\text{Specific speed } N_s = 3.65 \times n \frac{\sqrt{Q}}{H^{3/4}}$$

$$N_s = 3.65 \times 1500 \times \frac{\sqrt{0.0425}}{21.5^{3/4}}$$

$$= 113.04$$

$$\text{Inlet diameter of impeller} D_1 = k_1 K_0 \sqrt[3]{\frac{Q}{N}}$$

where k_1 =1.1~ 1.15and K_0=4

$$D_1 = 1.1 \times 4.5 \times \sqrt[3]{\frac{0.0425}{1500}}$$

$$D_1 = 0.150 \text{ m} = 150 \text{ mm}$$

$$\text{Outlet diameter of impeller} D_2 = 19.2 \left(\frac{N_s}{100}\right)^{1/6} \frac{\sqrt{2gH}}{N}$$

$$D_2 = 0.268 \text{ m}$$

= 268 mm = 260 mm (as per actual diameter)

D_0 is the eye diameter of impeller $D_0 = K_0 \sqrt[3]{\frac{Q}{N}}$

$$= 4.5 \times \sqrt[3]{\frac{0.0425}{1500}}$$

$$D_0 = 0.137 \text{ m} = 137 \text{ mm}$$

Shaft diameter at hub section $d_0 = \sqrt[3]{\frac{T}{0.2\,\tau}}$

Where T is torsion moment and given by

$$T = 9.65 \frac{P}{N}$$

Where $P = \alpha_1 .\ 9.81\ Q\ H / \eta$

= 1.5 *9.81* 0.0425*21.5 /0.78

= 17.238 kW

$T = 9.65 \times 17.238 \times 1000/1500$

= 110.89 N-m

And taking $\tau = 80\ \text{MN/m}^2$

Therefore, $$d_0 = \sqrt[3]{\frac{110.89}{0.2 \times 80 \times 10^6}}$$

$$d_0 = 0.0190 \text{ m} = 19 \text{ mm}$$

Hub diameter $D_{bt} = 1.5 \times 19 = 28.5$ mm

Hub length $L_{bt} = 2 \times 19 = 38$ mm

Inlet width of impeller $b_1 = R_0/2$

where $R_0 = D_0/2 = 137/2 = 68.5$ mm

$$b_1 = 68.5/2$$

$$= 34.25 \text{ mm}$$

Outlet width of impeller $b_2 = 0.78 \times \left(\frac{n_s}{100}\right)^{1/2} \left(\frac{Q}{n}\right)^{1/3}$

$$= 0.78 \times \left(\frac{113.04}{100}\right)^{1/2} \left(\frac{0.0425}{1500}\right)^{1/3}$$

$$b_2 = 0.02527 \text{ m}$$

$$= 25 \text{ mm}$$

Major dimensions of impeller are shown in Fig 3.4.

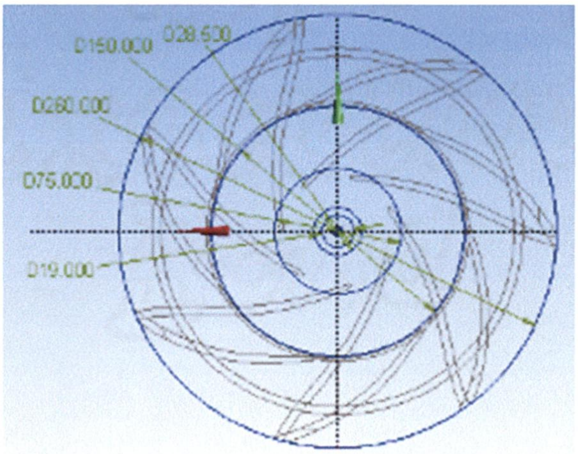

Fig. 3.4 Dimensioning of impeller

Velocity triangles at inlet and outlet of impeller are shown in Fig 3.5.

Inlet blade angle of impeller is:

$$\tan\beta_1 = V_1/u_1$$

u_1 can be calculated as :

$$u_1 = \frac{\pi D_1 N}{60} = \frac{\pi \times 0.150 \times 1500}{60}$$

$$= 11.78 \text{ m/s}$$

Here, we know $V_1 = V_{f1}$

And we also know, $Q = \pi D_1 b_1 . V_{f1}$

Hence,

$$V_{f1} = \frac{Q}{\pi D_1 b_1}$$

$$= \frac{0.0425}{\pi \times 0.150 \times 0.03425}$$

$$= 2.58 \text{ m/s}$$

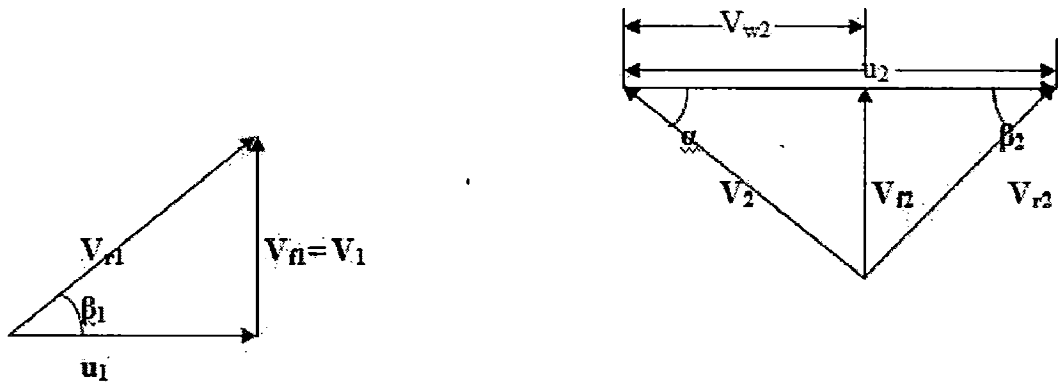

Fig. 3.5 Velocity triangle at inlet and outlet of impeller

So,

$$\tan\beta_1 = V_1/u_1$$

$$= 2.58/11.78$$

Inlet angle of impeller, $\beta_1 = 12.35°$

Outlet angle of impeller is assumed as 37°

All the worked out dimensions of impeller is given in Table 3.2.

Table 3.2: Dimensioning of impeller

S. No.	Details	Dimensions
1.	Inlet diameter of impeller	150 mm
2.	Outlet diameter of impeller	260 mm
3.	Eye diameter of impeller	137 mm
4.	Shaft diameter at hub section	19 mm
5.	Hub diameter	28.5 mm
6.	Inlet width of impeller	34.25 mm
7.	Outlet width of impeller	25 mm
8.	Inlet angle of impeller, β_1	12.35°
9.	Outlet angle of impeller	37°

3.3.3 Draft Tube

Draft tube is an important part for turbine operation and its effect cannot be neglected as it can recuperate the kinetic energy of water coming out of runner. A design detail of draft tube is necessary to evaluate the performance of pump in reverse mode. Draft tube is designed as per IS: 12800. Dimensions of draft tube are shown in Fig 3.6.

$$\text{Height of draft tube } 'H_1' = 3.6D_2$$

$$= 3.66\times260$$

$$= 952.5 \text{ mm}$$

$$\text{Diameter of the draft tube at exit end } D_3 = 1.17\ D_2$$

$$D_3 = 306 \text{ mm}$$

All the dimensions of draft tube are given in Table 3.3.

Table 3.3: Dimensioning of draft tube

S.No.	Details	Dimensions (mm)
1.	Diameter at inlet of draft tube	150 mm
2.	Height of draft tube up to exit end	952.5 mm
3.	Diameter of the draft tube at exit end	306 mm

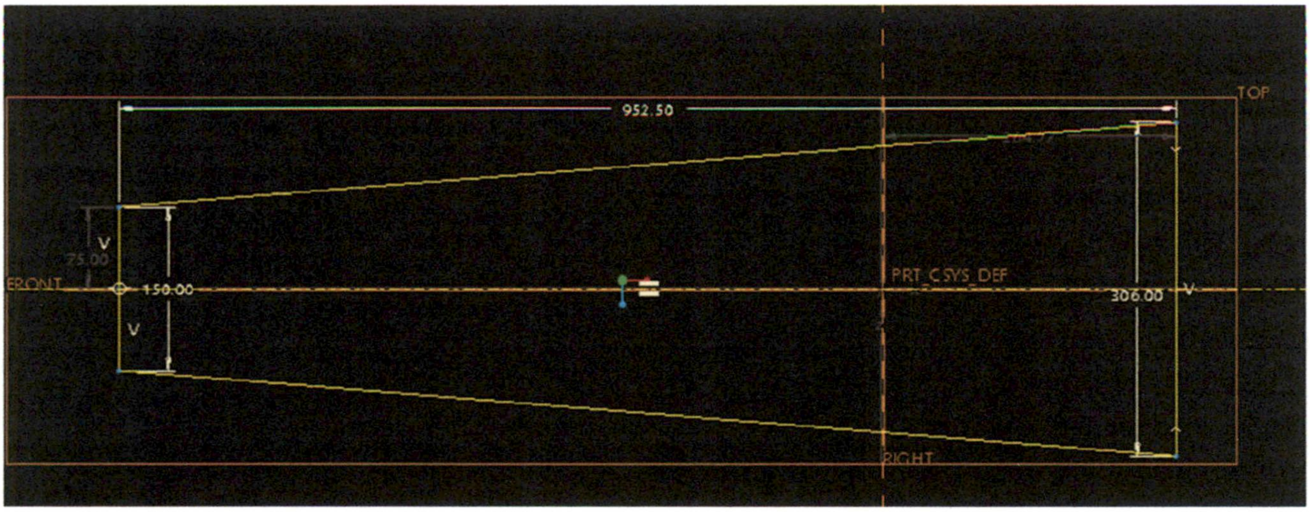

Fig. 3.6 Dimensions of draft tube

3.3.4 Guide Vanes

As there is no flow control mechanism in conventional pump, so the fluctuation in the flow results in poor part load efficiency. In order to use PAT with its maximum efficiency over a wide range of varying discharge, movable guide vanes can be used. Guide vanes have cross section known as aerofoil section made of cast steel which allows the water to pass over them with minimum losses and without forming eddies. Guide vanes are mounted around the periphery of impeller in casing, as shown in Fig 3.7.

Fig. 3.7 Guide vanes around the periphery of impeller

The diameter along which the guide vanes are to be provided should be greater than impeller diameter because the guide vanes need some space for their movement. Since pump impeller diameter is equal to 260 mm, therefore the diameter of circle along which guide vanes are to be provided, D_G is taken as 300 mm. As the number of slots (vanes) in pump impeller are seven. Therefore, maximum number of guide vane should be seven. But since there is less space for movement of 7^{th} guide vane , therefore 6 guide vanes are to be provided at their respective location and the 7^{th} location is left as it is. By providing the guide vanes the area of casing is slightly restricted at entry, which results more flow available at the end of casing. Since pump impeller width is equal to 25 mm therefore guide vane width is taken as 25 mm. Details of guide vane used for modeling which is taken from earlier experiment study, are as follows:

- Length of guide vane = 80 mm
- Width of guide vane = 25 mm
- No. of guide vanes = 6

CHAPTER 4

MODELING, MESHING AND NUMERICAL SIMULATION

4.1 GENERAL

In this chapter, investigation of the flow field in modified pump in turbine mode using computational fluid dynamics (CFD) methodology is discussed. The basic elements of the modified pump used in turbine mode are casing, impeller, guide vanes and draft tube. The computational modeling and meshing of calculation domain is first step in CFD analysis. For modeling purpose, design details of centrifugal pump have been carried out in last chapter. The complete process of computational modeling and meshing of pump configuration i.e. the impeller, casing and the draft tube and guide vanes is described in following paragraphs. After meshing, flow simulations are carried out for different flow rates and operating characteristics of modified PAT is obtained and compared with available with available experimental results. The salient features of centrifugal pump are given in Table. 4.1.

Table 4.1: Salient features of centrifugal pump

S.No.	PARTICULARS	DIMENSIONS
1.	No. of impeller vanes	7 Nos.
2.	Diameter of impeller eye	150 mm
3.	Outer diameter of impeller	260 mm
4.	Blade thickness	6 mm
5.	Width of impeller at outlet	25 mm
6.	Vane angle at inlet	12.5^0
7.	Vane angle at outlet	37^0
8.	Rated Head	21.5 m
9.	Rated Discharge	42.5 lps
10.	Rated pump efficiency	78 %
11.	Rated Speed	1500 rpm

4.2 CREATING GEOMETRY

The geometries are modeled using Pro-engineer solid modeler except impeller blades.

4.2.1 Creating Model of Impeller

Impeller of pump is both side covered and contain seven radial curved blades. The detail of impeller of centrifugal pump has been calculated for modeling. Creation of Impeller model is carried in two steps: creation of blade profile and creation of complete runner domain. Blade profile is created using BladeGen software of ANSYS 14.0. Steps for creation of impeller are given below:

- Select radial impeller for centrifugal pump then a dialog box as shown in Fig 4.1 will open.
- Input for blade profile will be given. After adjusting vane angle at inlet and outlet, Meridional Profile of blade is shown in Fig 4.2

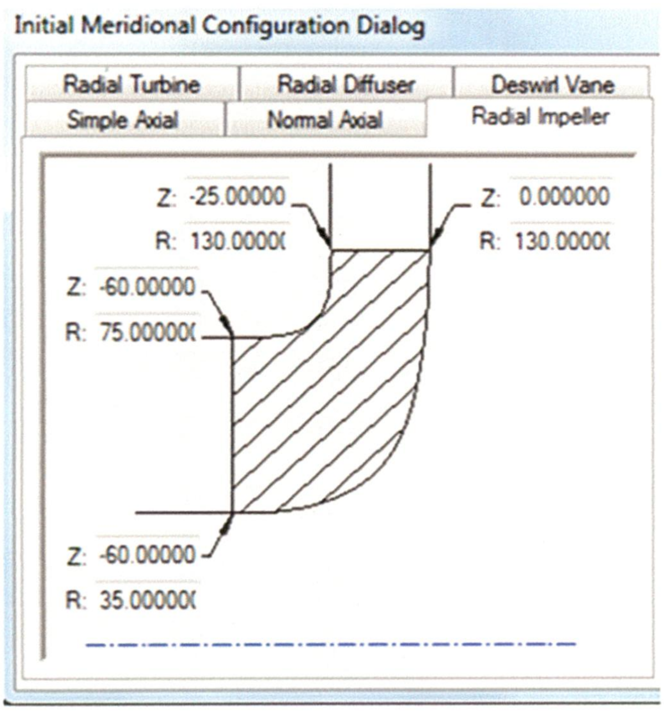

Fig. 4.1 Initial meridional configuration dialog

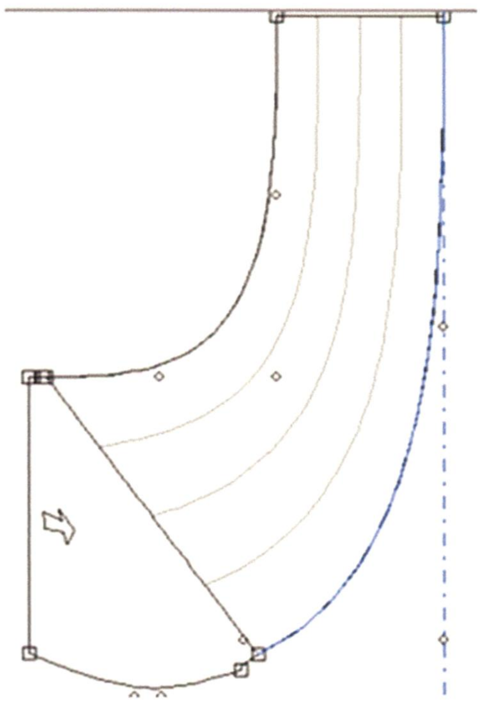

Fig. 4.2 Meridional profile of blade

- This blade profile is transferred to Geometry.
- This blade profile will be opened in Geometry and save IGES format of blade profile and open in Pro-Engineer as shown in Fig 4.3

- After this, all the domain of complete runner will be created in Pro-E by using simple commands like extrude and revolve. Complete model of Impeller in Pro-E is shown in Fig 4.4 and Fig 4.5.

Fig. 4.3 IGES format of blade profile in Pro-E

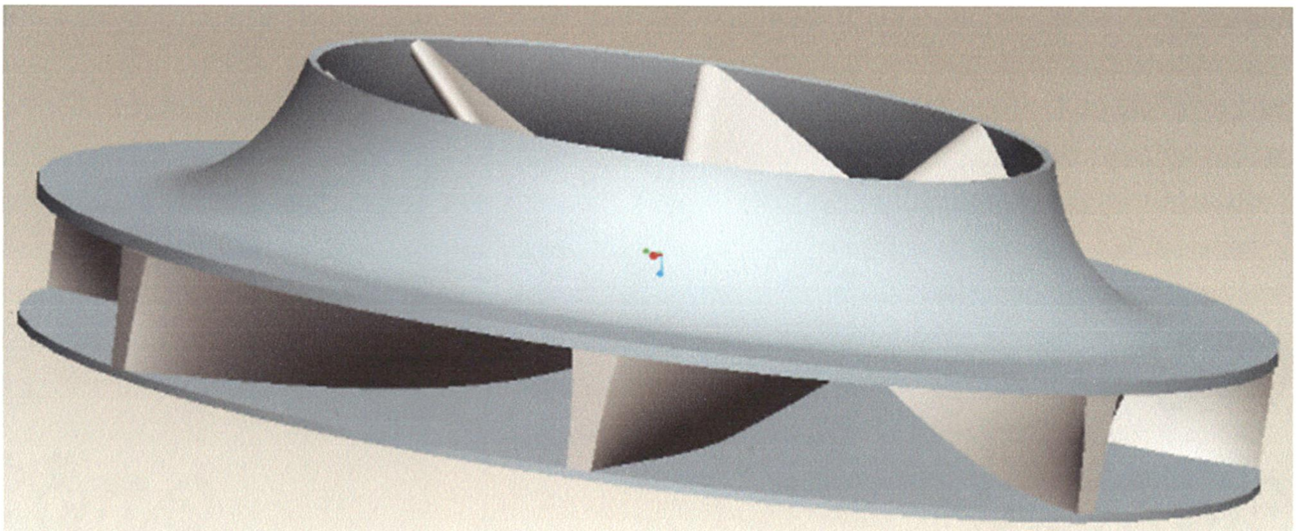

Fig. 4.4 Impeller of centrifugal pump

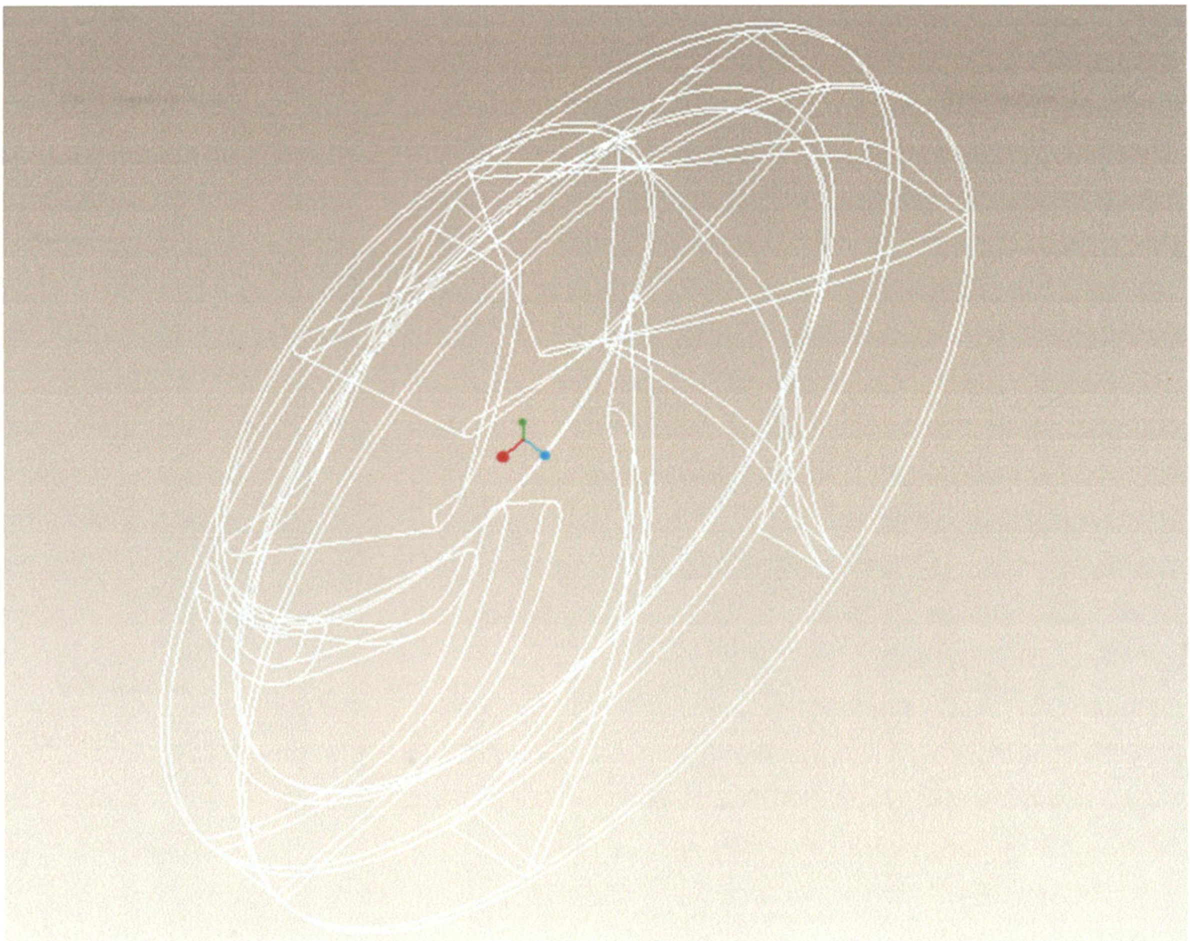

Fig. 4.5 Wireframe of complete model of impeller in Pro-E

4.2.2 Creating Model of Casing

The casing, which is one of the complex geometry in pumps or turbines was also created in Pro-Engineer in a simplest way so as to get a better mesh as the quality of mesh depend on the complexity of the geometry. Casing is divided in eight sections for modeling. Modelling of casing is done by using swept blend command. Trajectory for swept blend is shown in Fig 4.6 and complete model of casing is shown in Fig 4.7.

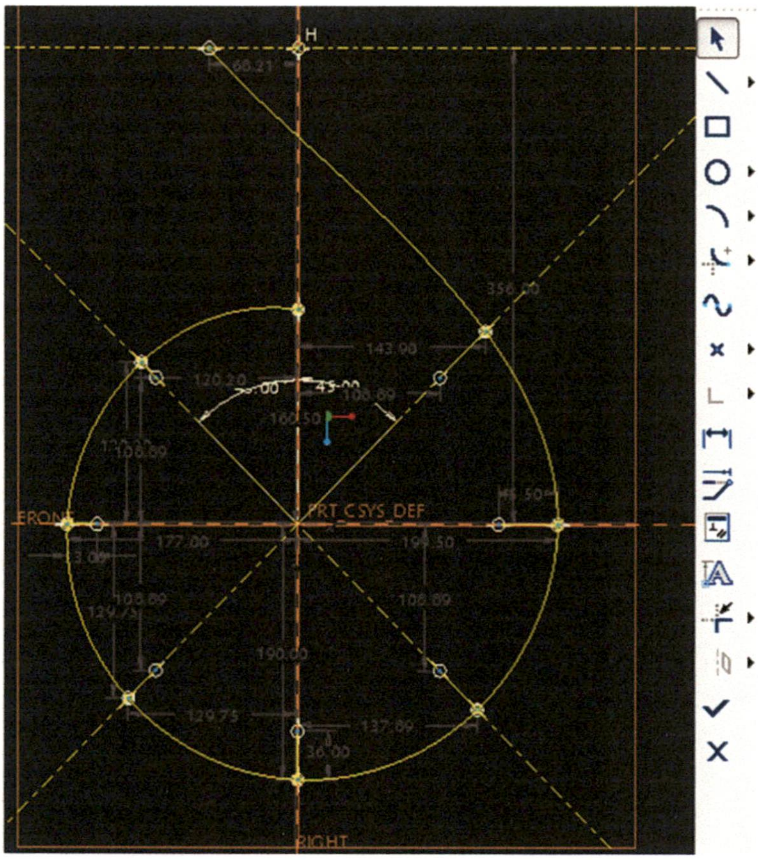

Fig. 4.6 Trajectory for casing modeling

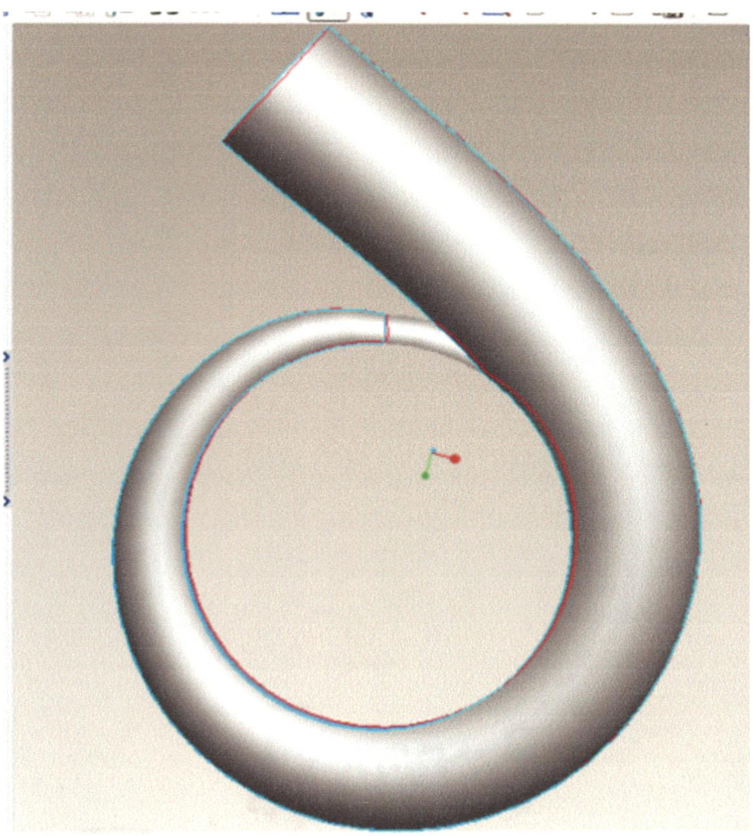

Fig. 4.7 Model of casing

4.2.3 Creating Model of Draft Tube

Required dimension for draft tube modeling has been calculated according to IS-12800 Standard. Draft tube is relatively simpler in construction and model of draft tube is created using Pro-E. Sketch and model of draft tube are shown in Fig 4.8 and 4.9 respectively.

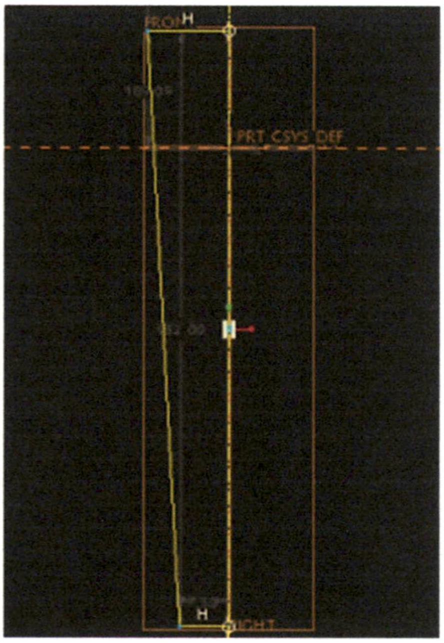

Fig. 4.8 Sketching for the modelling of draft tube

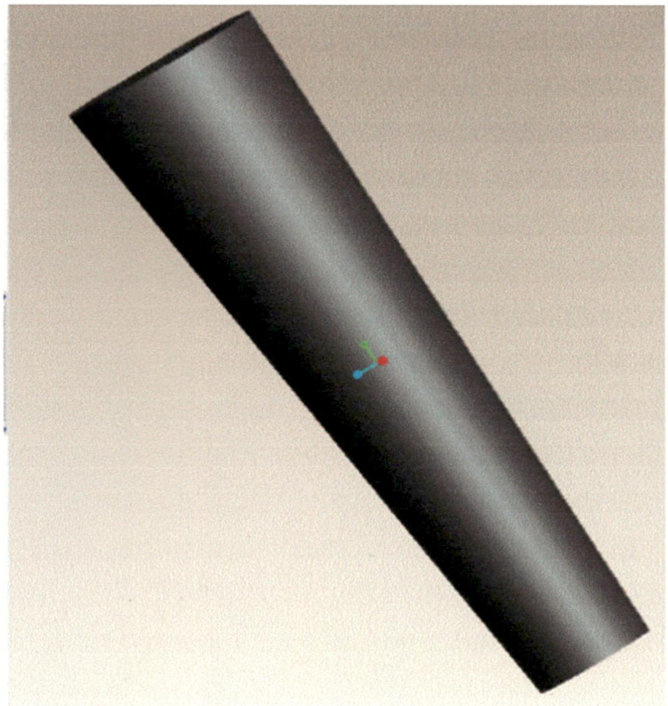

Fig. 4.9 Model of draft tube

4.2.4 Creating Model of Guide Vanes

The major problem of PAT is its poor part load efficiency which can be improved to a greater up to extend by providing the guide vanes around the impeller which can be moved according to the flow or load condition and thus a constant efficiency can be maintained over a wide range of discharge. Modeling of guide vanes is done by using Pro-E. Wireframe and model of guide vane are shown in Fig 4.10, 4.11 and 4.12 respectively.

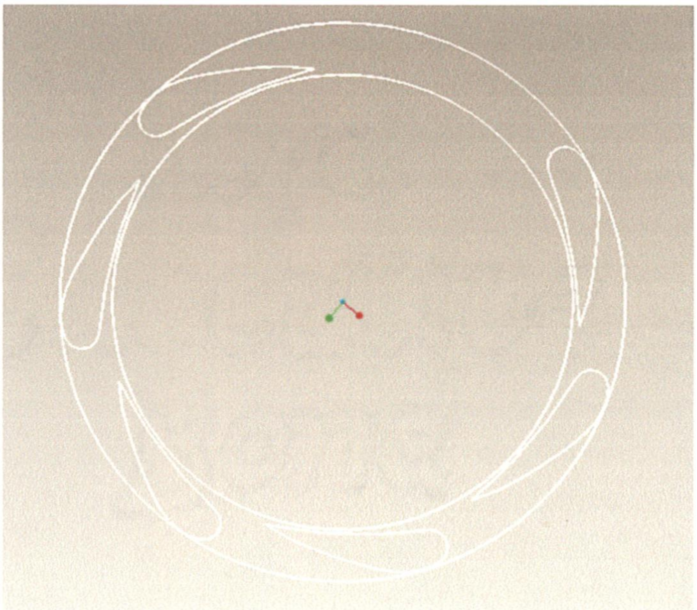

Fig. 4.10 Wireframe of guide vanes model at 50° (design condition)

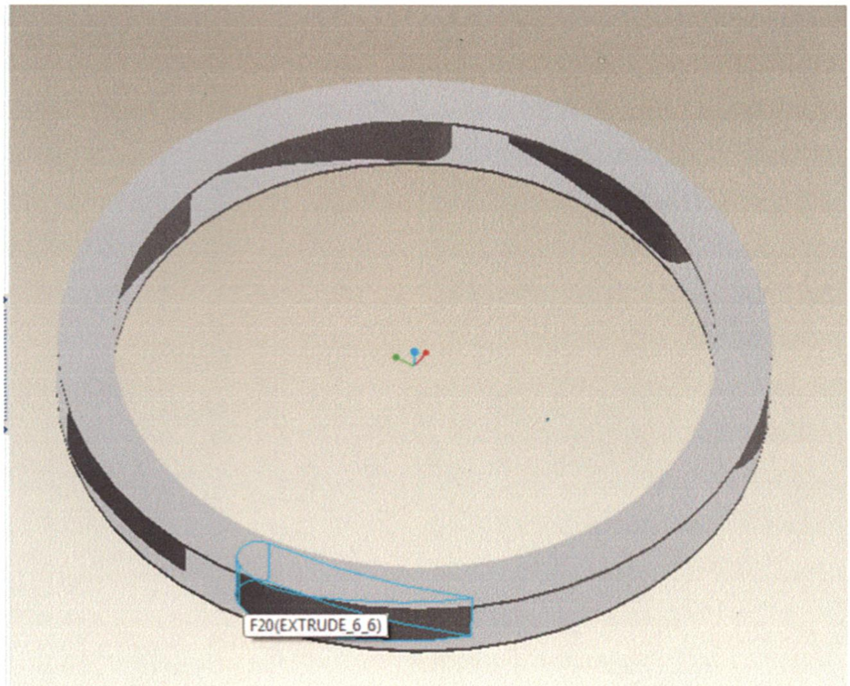

Fig. 4.11 Modeling of guide vanes at design condition

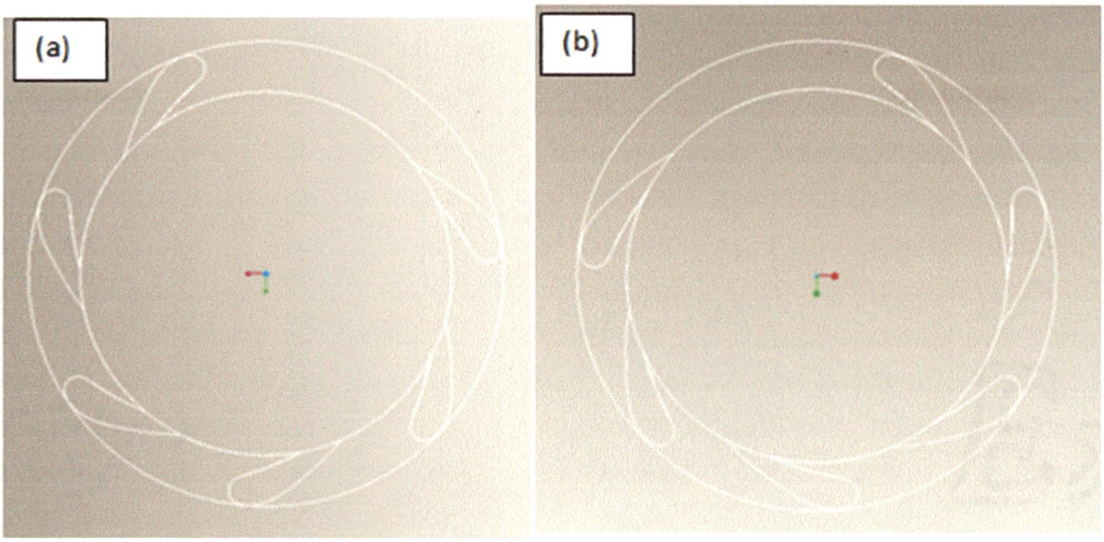

Fig. 4.12 (a) Wireframe of guide vanes model at 30°

(b)Wireframe of guide vanes model at 38°

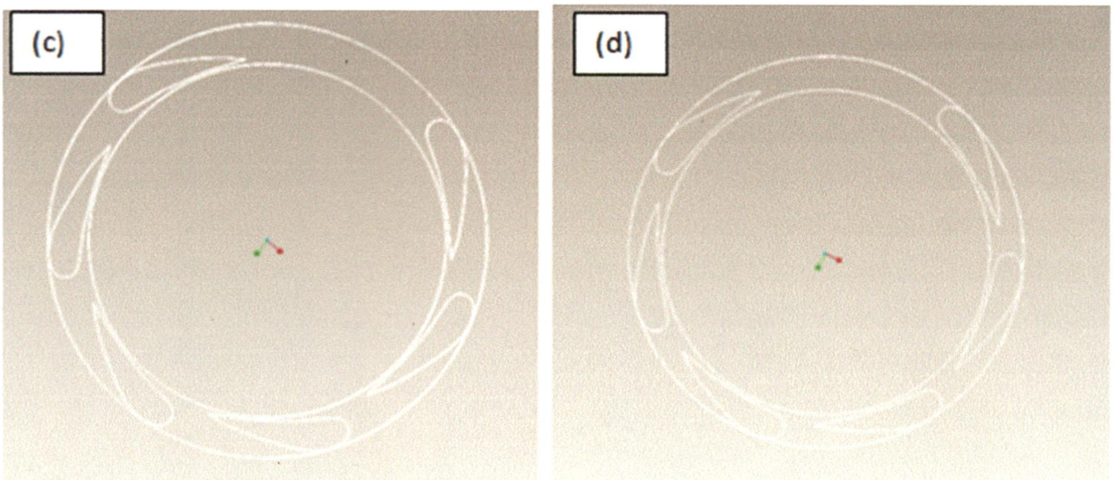

Fig. 4.12 (c) Wireframe of guide vanes model at 45°

(d)Wireframe of guide vanes model at 55°

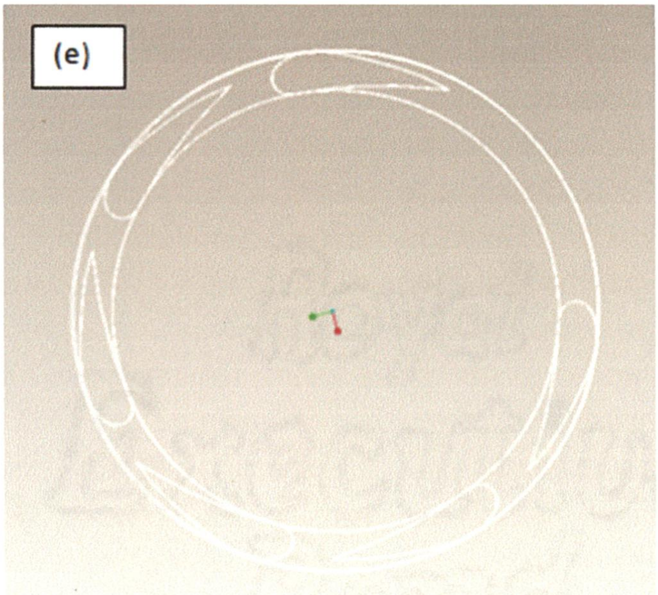

Fig. 4.12 (e) Wireframe of guide vanes model at 60°

4.2.5 Assembly of Modified Pump used in Turbine Mode

IGES format of all component like impeller, casing, draft tube and guide vanes are saved in pro-e. These all components are imported in ANSYS Geometry module one by one. Step for assembly are as follow:

i. First of all, casing is imported in ANSYS workbench
ii. After clicking on Generate button, casing is displayed in graphics window
iii. Then another part of pump is imported
iv. After this go to 'Operation' and select "Add Frozen" click on generate. Another part will be displayed in graphics window.
v. Repeat the above step 2 and 3 for next two parts, thus the assembly of modified PAT will be displayed in graphics window as shown in Fig 4.13 and Fig 4.14

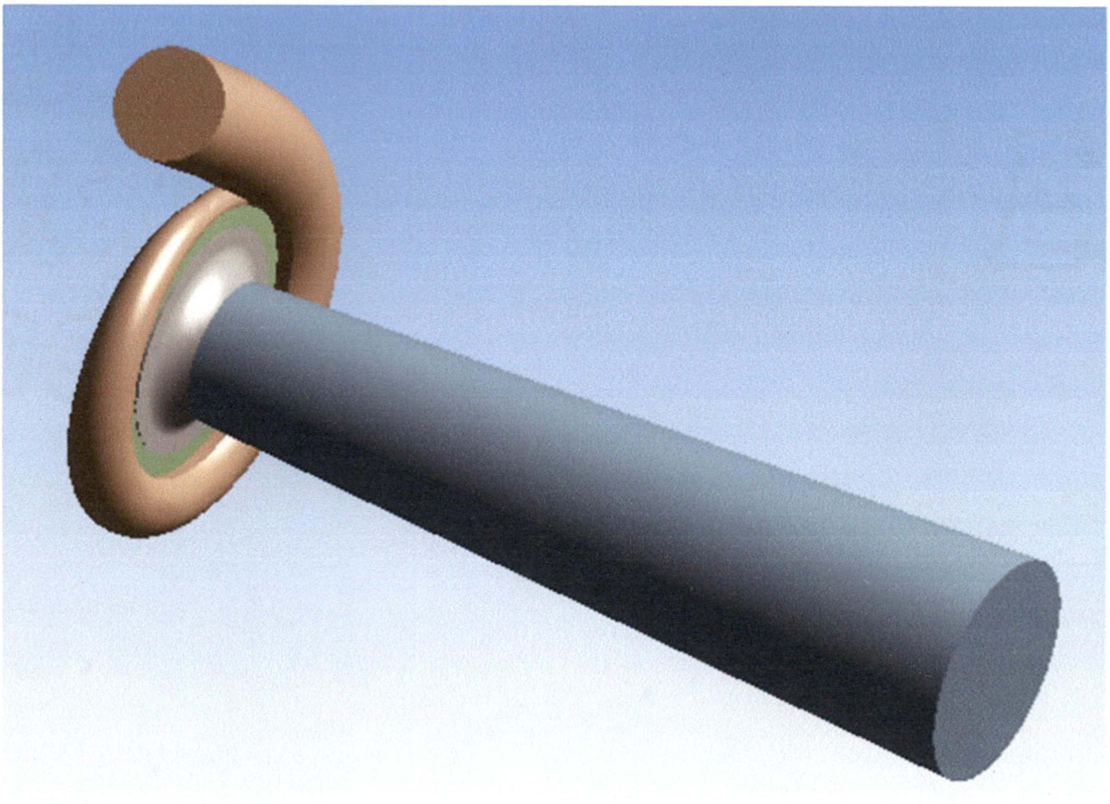

Fig. 4.13 Computational model of modified pump as turbine

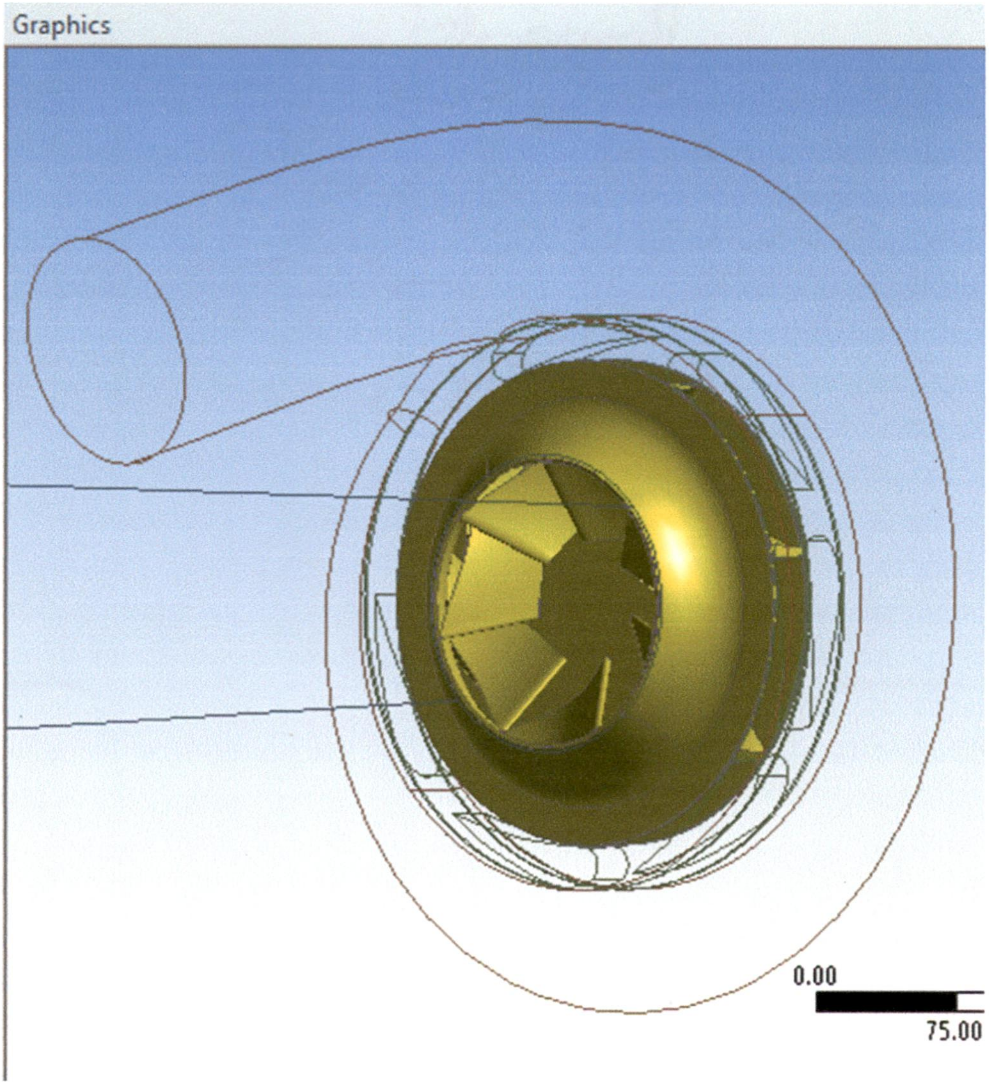

Fig. 4.14 Wireframe view of assembly of modified PAT (highlighted impeller)

4.3 MESH GENERATION

One of the most important and time consuming task in the CFD simulation process is the generation of mesh. As the geometry is complex, unstructured mesh consists of triangular and tetrahedral element is used. "MESH" is used for meshing the complex geometry of impeller, casing, draft tube and guide vanes. Complete assembly is imported in Geometry part of 'MESH' module of ANSYS.

4.3.1 Specifying Zone Types

Zone-type specifications define the physical and operational characteristics of the model at its boundaries and within specific regions of its domain. There are two classes of zone-type specifications:

- Continuum types
- Boundary types

Before generating the mesh, it is required to specify the zone types in 'design modeler' and 'Mesh' .Following zone types are specified for the present case:

Continuum types

Continuum-type specifications, such as FLUID or SOLID are defined in 'design modeler' as shown in Fig 4.15. The characteristics of the model within specified regions of its domain are defined. Following Fluid zones are specified.

a. Casing volume: Fluid
b. Impeller volume : Fluid
c. Draft tube volume : Fluid
d. Guide vanes volume : solid

Now this assembly is edited in 'Mesh'. In 'Mesh', Boundary-type specifications only inlet and outlet and connections (Mesh Interface) for simulation are defined as shown Fig 4.16.

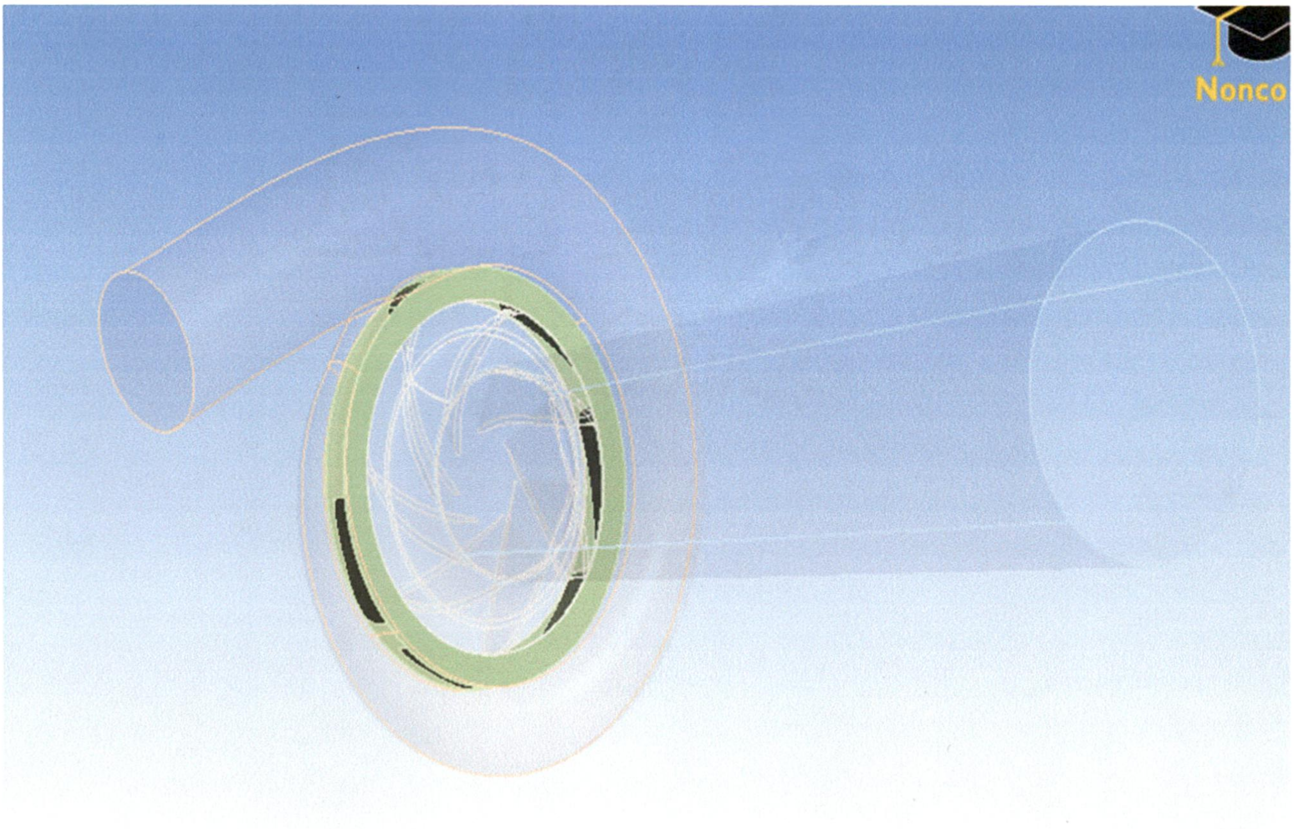

Fig. 4.15 Complete assembly of modified PAT after defining continuum types specifications

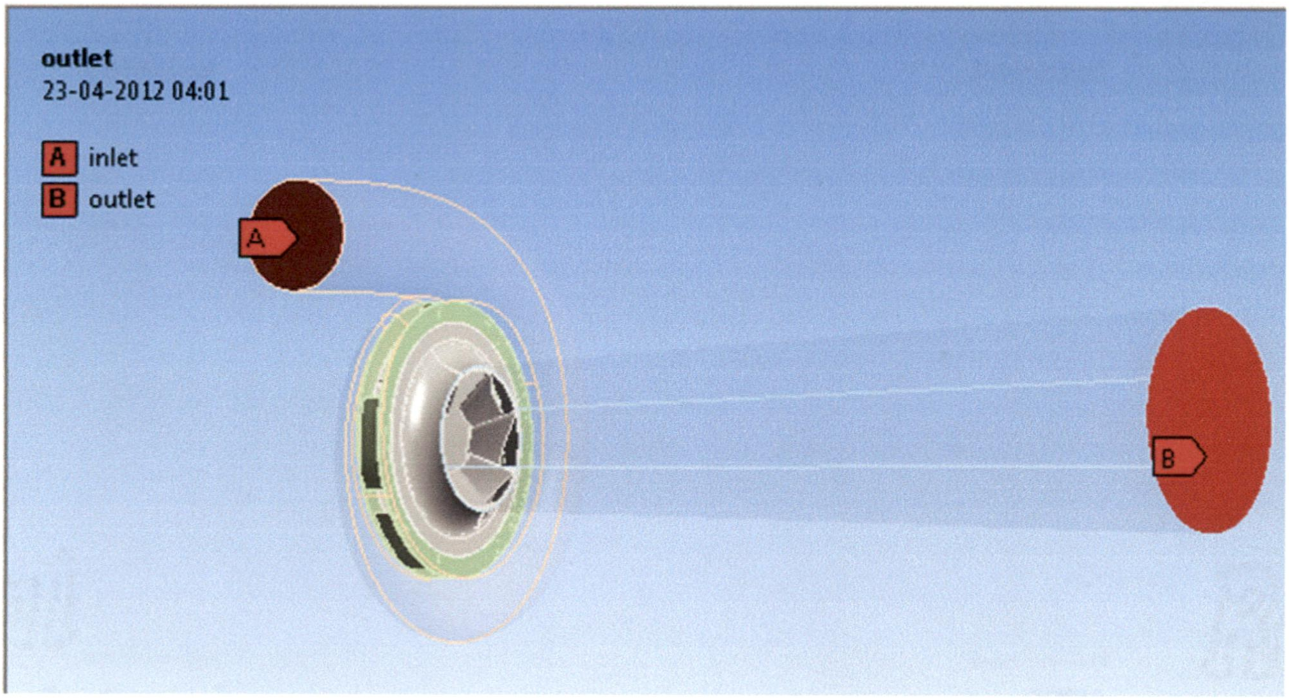

Fig. 4.16 Defining of inlet and outlet for modified PAT

Now, physics preference is defined as CFD and fluent solver is specified. Mesh is generated for each component. Details of unstructured tetrahedral meshing are given below for each component:

i. **Casing**

Unstructured Tetrahedral scheme has been used for the meshing for casing. Casing consists of 7180 nodes and 33880 elements. Meshing of casing is of fine quality.

ii. **Impeller**

Same meshing scheme is used for the meshing of impeller. Impeller consists of finer grid with total meshing node 1108795 and elements 5269492.

iii. **Guide Vanes**

Guide vane consists of 502077 nodes and 2449110 elements by using Unstructured Tetrahedral scheme.

iv. **Draft Tube**

Total 11373 nodes and 10250 elements have been generated from the tetrahedral meshing of draft tube.

Meshing for complete assembly is shown in Fig 4.17

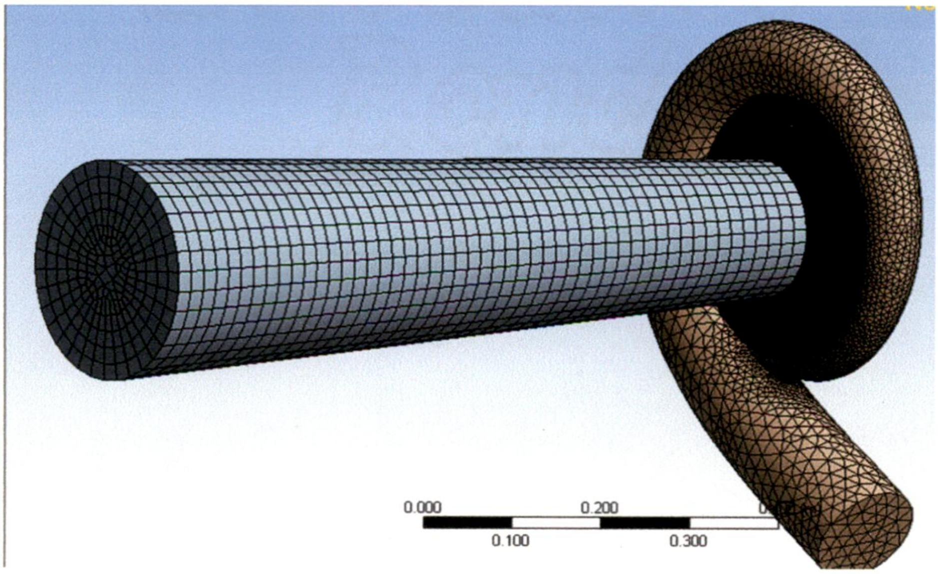

Fig. 4.17 Computational mesh for modified pump as turbine setup

4.4 EXPORT THE MESH FILE

After completion of modeling, meshing and specifying boundary zones in PRO-E and ANSYS MESH respectively, the mesh is exported to 'FLUENT Solver' for simulation. Pre- Steps are as follows for exporting the mesh file to FLUENT.

i. Update the project in 'MESH'.
ii. Edit project in 'Setup' of FLUENT cell, FLUENT Launcher will open.
iii. After click 'OK' on FLUENT Launcher, mesh file is read. Mesh of complete assembly will be displayed in graphics window of FLUENT as shown in Fig 4.18.

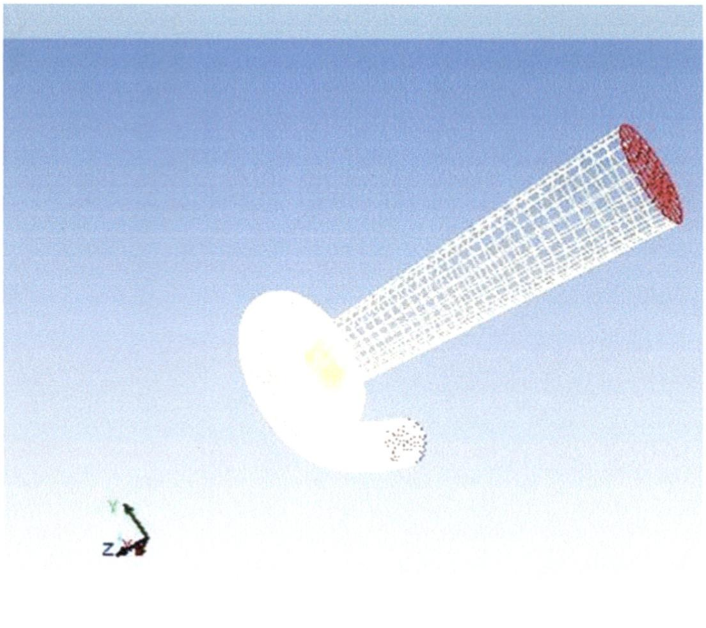

Fig. 4.18 Displaying the complete set up of modified PAT in FLUENT

4.5 NUMERICAL SIMULATION IN FLUENT

This article describes the 3-dimensional simulation of flow in modified pump as turbine. A commercial 3-dimensional Navier-Stokes code called "FLUENT" with different turbulence models is used to simulate the flow. The "FLUENT" Moving Reference Frame (MRF)-model is used to consider the Runner in rotating reference frame and other components in stationary reference frame. In the calculation, Finite Volume method (FVM) is used for the discretization of governing equations. The following assumptions were made during the simulation of modified PAT.

i. Steady state flow
ii. Incompressible fluid
iii. Constant fluid properties
iv. Single phase flow
v. No leakage in turbine
vi. Hydraulically smooth surfaces of all components.

Steps for numerical simulation of pump as turbine are as described below:

STEP 1: Mesh check

Check the mesh files in fluent. It is very important to check the quality of the resulting mesh, It is also important to verify that all the elements have positive area/volume, otherwise simulation is not possible.

STEP 2: Model and solver

Select segregated solver for the flow analysis of pump in reverse mode. Select the standard k-ε model among the available model in FLUENT. This model is robust and converges fast, which is usually better suited for its popularity.

Governing equations and Turbulence model

The simplest "complete models" of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained by using physical reasoning and bears little resemblance to its mathematically exact counterpart. The standard k-ε model is valid only for fully turbulent flows. The assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible.

It is assumed in the present investigation that the flow can be treated as an incompressible fluid with constant physical properties. Furthermore, the flow is considered steady relative to moving reference system, the physical problem can be described by the Reynolds-averaged Navier-Stokes equations and the mass continuity equations, which can be expressed in general Cartesian form as:

$$\frac{\partial U_j}{\partial x_j} = 0. \tag{4.1}$$

$$\rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] - \frac{\partial}{\partial x_j}(\rho \overline{u_i u_j}) \tag{4.2}$$

The subscripts i and j can take values of 1, 2 and 3 in order to take into account the three dimensions in which the fluid revolves. These averaged equations contain the Reynolds stress terms $\overline{u_i u_j}$, which characterize the influence of the turbulence on the mean flow field. This additional term must be modeled in order to solve these equations. Hence, the quality of the numerical results will depend on the choice of the turbulence model, which is a set of additional equations.

The standard $k - \varepsilon$ model is the most used formulation in engineering calculations. It provides a linear relationship between the Reynolds stresses and the mean rate of strain by using the eddy-viscosity hypothesis. Hence, the Reynolds stress terms can be expressed as:

$$\overline{u_i u_j} = -v_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) + \frac{2}{3}\kappa\delta_{ij} \tag{4.3}$$

Where δ_{ij} is unity for $i=j$ and zero otherwise. The turbulent viscosity v_t is directly related to the turbulent kinetic energy k and the dissipation rate :

$$v_t = \frac{\mu_t}{\rho} = C\mu\frac{\kappa^2}{\varepsilon} \tag{4.4}$$

The two extra equations introduced by the $k - \varepsilon$ model take the following form:

$$U_j \frac{\partial \varepsilon}{\partial x_j} = \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{\varepsilon 1}\frac{\varepsilon}{\kappa}P_\kappa - C_{\varepsilon 2}\frac{\varepsilon^2}{\kappa} \tag{4.5}$$

$$U_j \frac{\partial \kappa}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \kappa}{\partial x_j} \right] + P_k - \varepsilon \tag{4.6}$$

Where the production of turbulence kinetic energy P_k can be written as:

$$P_k = v_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \tag{4.7}$$

The model constants $C_\mu, C_{\varepsilon 1}, C_{\varepsilon 2}, \sigma_k, \sigma_\varepsilon$ are determined experimentally and are assigned standard values of 0.09, 1.44, 1.92, 1.0 and 1.3 respectively.

STEP 3: Material property

This step requires selecting the type of working fluid, which is taken into consideration, for flow analysis. Water-liquid for fluid zone and steel for solid zone have been taken as working fluid and defined its property. Table 4.2 shows the properties of material used for simulation.

Table 4.2: Properties of material

S. No.	Material Name	Properties and their value	
1.	Water	Density	998.2 (kg/m^3)
		Specific heat	4182 (J/kg-k)
		Thermal conductivity	0.6 (w/m-k)
		Viscosity	0.001003 (kg/m-s)
2.	Steel	Density	8030 (kg/m^3)
		Specific heat	502.48 (J/kg-k)
		Thermal conductivity	16.27 (w/m-k)

STEP 4: Cell zone conditions

Cell zone condition defines the moving frame and stationary frame i.e. define the motion for the moving component of the modified PAT and stationary components of complete set up. Cell zone conditions are defined for following parts:

- Impeller: define moving frame (frame motion and mesh motion), give the rotation to impeller 157.07 rad/s. rotation axis direction is Z- direction in sub pad of cell zone condition.
- Casing: define mesh motion and provide translation velocity in X and Y -direction according to mass flow at inlet as the area at inlet is fixed.
- Draft tube: define mesh motion and give translation velocity in Z-direction.
- Guide vane: guide vanes are stationary; no motion is given to it.

STEP 5: Boundary conditions

The boundary conditions are very critical part in numerical simulation, because the parameters specified in the boundary conditions governs the results of differential equation of flow problem. The boundary conditions used in this problem are as follows:

1. Inlet boundary: Mass flow inlet with flow normal to the inlet boundary, which is inlet of volute casing in centrifugal pump. Range for mass flow inlet for the simulation is 25 to 53 lps.
2. Outlet boundary: At the outlet of calculation domain, the static pressure is specified.
   a. i.e. $P_{out} \geq P_{atm}$

Pressure at draft tube out let is taken as 17000 Pa for simulation.

3. Value of different turbulent parameters like turbulence intensity was taken as 3% and turbulence viscosity ratio has been taken as 140 for both inlet and outlet boundary.

STEP 6: Solution control and Solution method

Various factors need to define the turbulence model. We can take the default values of these relaxation factors as shown in "FLUENT" window or can take the more correct value if available. Different factors which need to be specified to solve the differential equation of flow and momentum. These factors are discussed here:

i. Under relaxation factor

a) Pressure = .40

b) Density = 1

c) Body force = 0.2

d) Momentum = 0.4

e) Turbulence kinetic energy = 0.6

f) Turbulence dissipation rate = 0.6

g) Turbulent viscosity = 1

ii Choose the second order discritization scheme for governing equations.

iii The simple scheme is used for the pressure velocity coupling

iv The momentum equation is solved by using second order upwind scheme.

v Presto scheme is used for the pressure correction.

STEP 7: Initialization, Iteration and Solutions

Before calculating the solution, initialization of solution will be done. Fluent performs number of iterations to converge the solution. Solution convergence depends on many factors like no of mesh elements type of meshing etc. In the present study convergence take place more than 1000 of iterations. After convergence save the CASE – DATA file. Different results have been observed from the flow simulation of modified PAT and will be discussed in chapter 5.

CHAPTER 5

RESULTS AND DISCUSSION

5.1 GENERAL

The numerically simulated flow field for 3-D of the modified PAT provided with guide vanes is analyzed to investigate the performance ofpump as turbine. Efficiency of PAT for different guide vane position with standard k-ε turbulence model is calculated and operating characteristics are drawn and presentedin this chapter-5. Further numerical results are compared with available experimental results.

5.2 NUMERICAL SIMULATION OF FLOW FIELD IN PUMP AS TURBINE

The performance of modified PAT can be well understood by numerical simulation of flow field inside the modified pump as turbine. The simulations are made at constant speed of impeller at 1500 rpm and at different values position of guide vanes for different discharge. Range of parameters which have been considered for simulation, are given in Table 5.1 for the pump used as turbine after modification.

Table 5.1: Range of parameters considered for the simulation

S. No.	Parameters	Range for modified PAT
1.	Head (m)	6.0-25
2.	Discharge (lps)	22.00-53.00
3.	Pump speed (rpm)	1500 rpm

To carry out the flow simulation on modified PAT, set of boundary conditions has been specified i.e. mass flow inlet at casing inlet and pressure outlet at draft tube outlet.

Overall efficiency of modified PAT is calculated based on the fundamental equation, i.e. ratio of output power from the turbine to input power supplied to the turbine [60].

$$\eta = \frac{T\omega}{Q(p_1 - p_2)} \tag{5.1}$$

In above equation, T is the net torque acting on the runner (N-m), $\omega$is the angular speech (radian), Q is discharge through PAT (m^3/s), $p_1$is total pressure at the inlet to the casing (Pa) and $p_2$is the total pressure at the exit of draft tube (Pa). The net torque acting on the runner is a resultant of pressure and viscous moments and is calculated by taking surface integral of cross product of stress tensor and radius vector.

$$T = \left(\int \left(\vec{r} \times (\bar{\bar{\tau}} \times \hat{n})\right) d_s\right) . \hat{a} \tag{5.2}$$

Where S represents the surfaces comprising all rotating parts; τ is the total stress tensor.$\hat{n}$is unit vector normal to the surface, r is the position vector, $\hat{a}$ is a unit vector parallel to the axis of rotation.

5.3 PRESSURE CONTOURS AND VELOCITY CONTOURS AT DESIGN CONDITION

Variation of pressure and velocity for a particular mass flow rate (Q= 48lpsdesign condition) at inlet and pressure at draft tube outlet in different parts inside the modified PAT provided with guide vanes are shown by different colors. With the help of pressure contours generated in 'FLUENT', Low pressure zone inside the modified pump as turbine can be determined which are useful to identify cavitation zones inside the modified PAT. To avoid the cavitation inside the modified PAT, pressure at any point within the PAT component should be higher than vapor pressure (10.302×10^4 N/ m^2). The variations of pressure distribution inside the different component are shown in Fig. 5.1 to Fig. 5.5.The

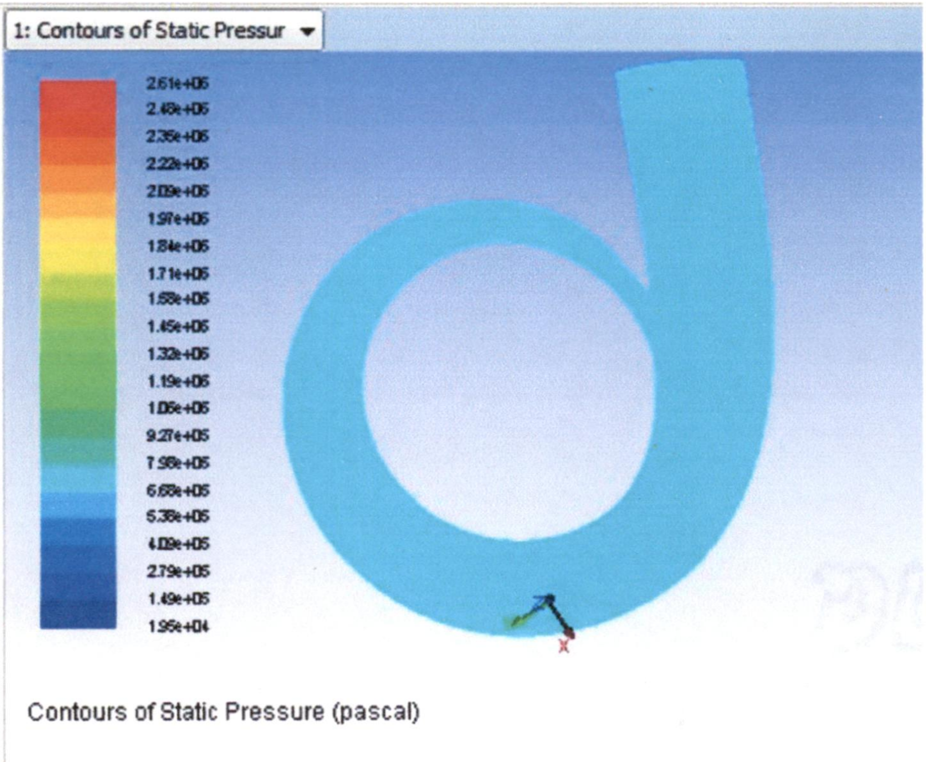

Fig. 5.1 Variations of pressure distribution inside the casing

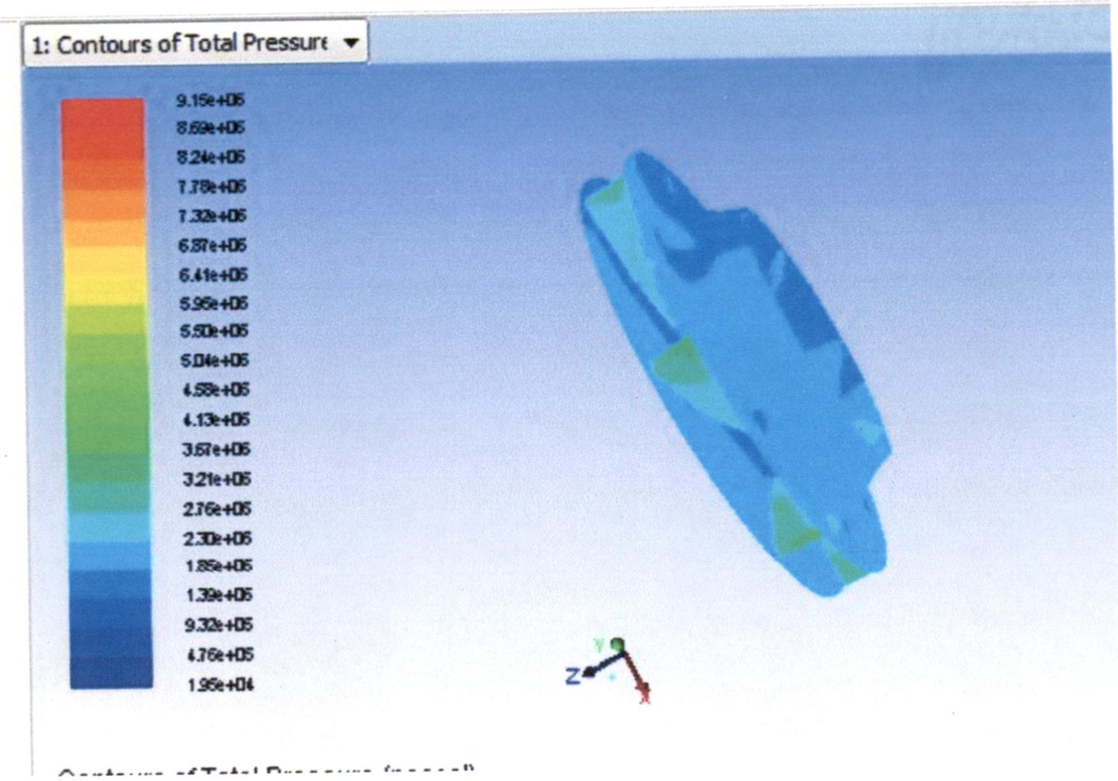

Fig.5.2Variations of total pressure distribution inside the impeller

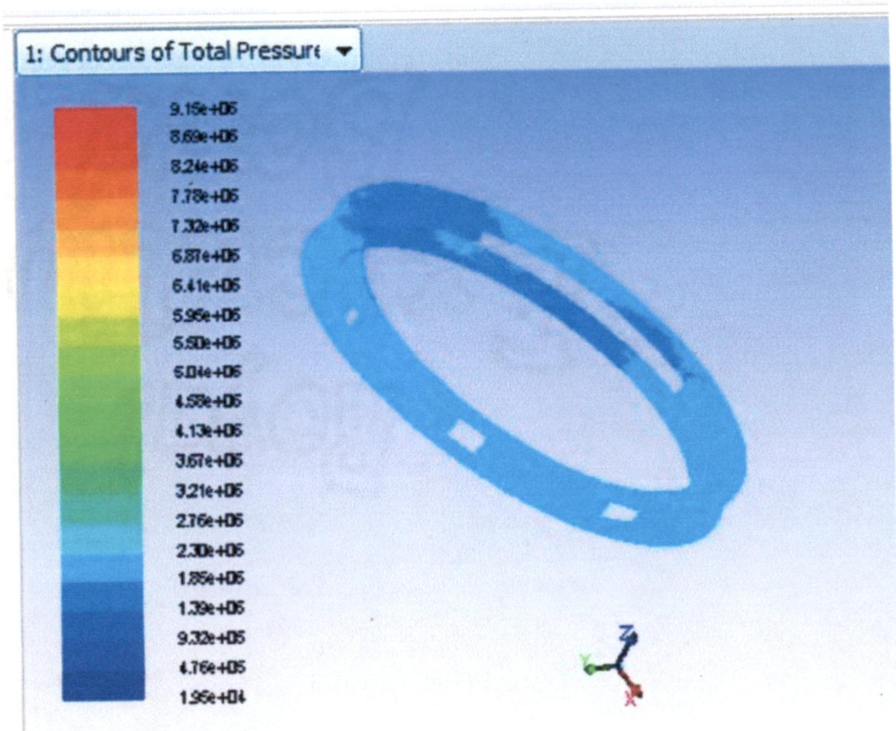

Fig. 5.3Variations of total pressure distribution inside the guide vanes

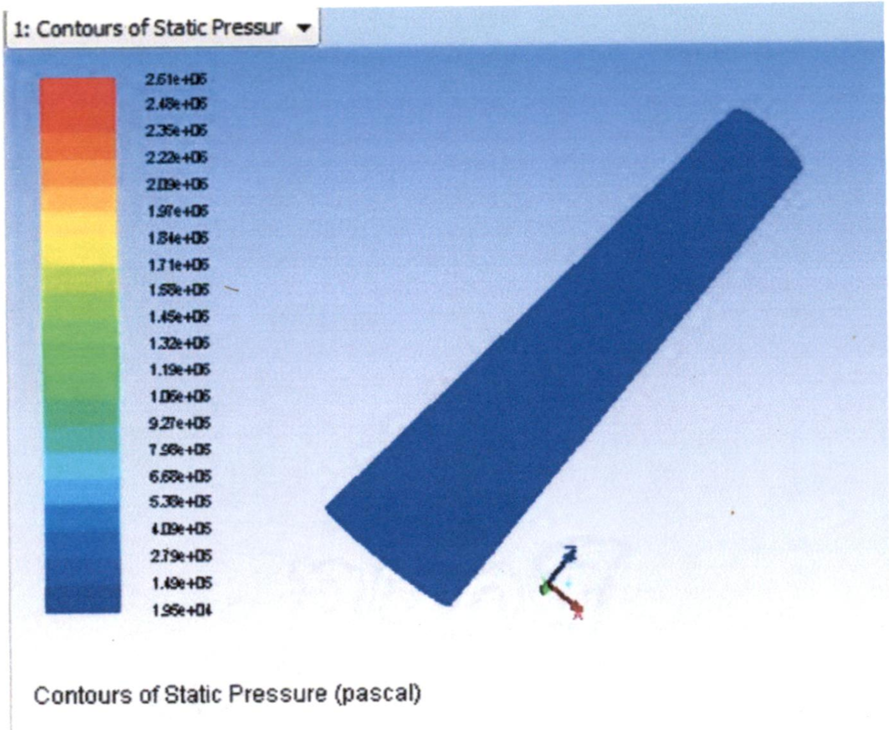

Fig. 5.4Variations of total pressure distribution inside the draft tube

Velocityvector field analysis is helpful to determine Zones of boundary layer separation which leads to eddy formation, and causes dissipation of energy. Velocityof casing, impeller, guide vanes and draft tube are shown in Fig. 5.5 to Fig. 5.8.

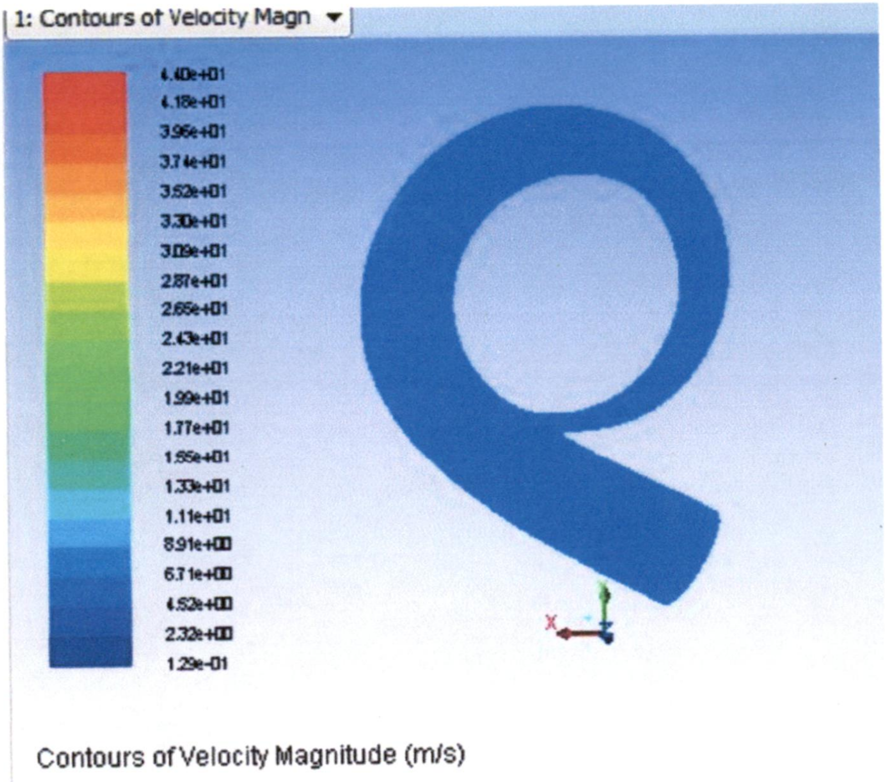

Fig. 5.5 Velocity in casing

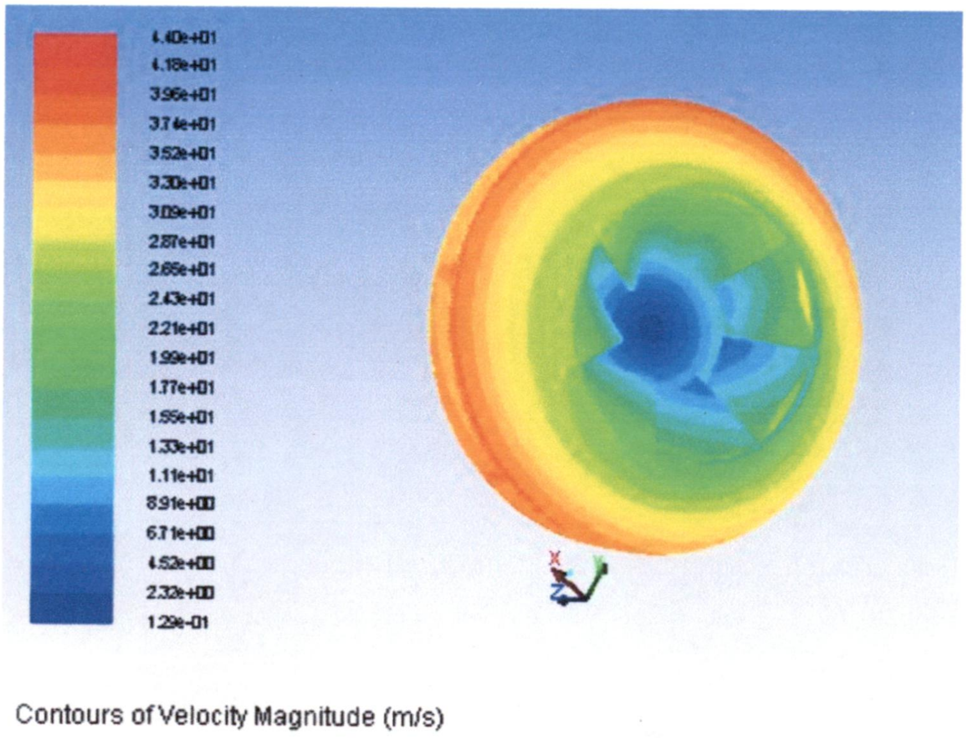

Fig. 5.6 Variation of velocity in impeller

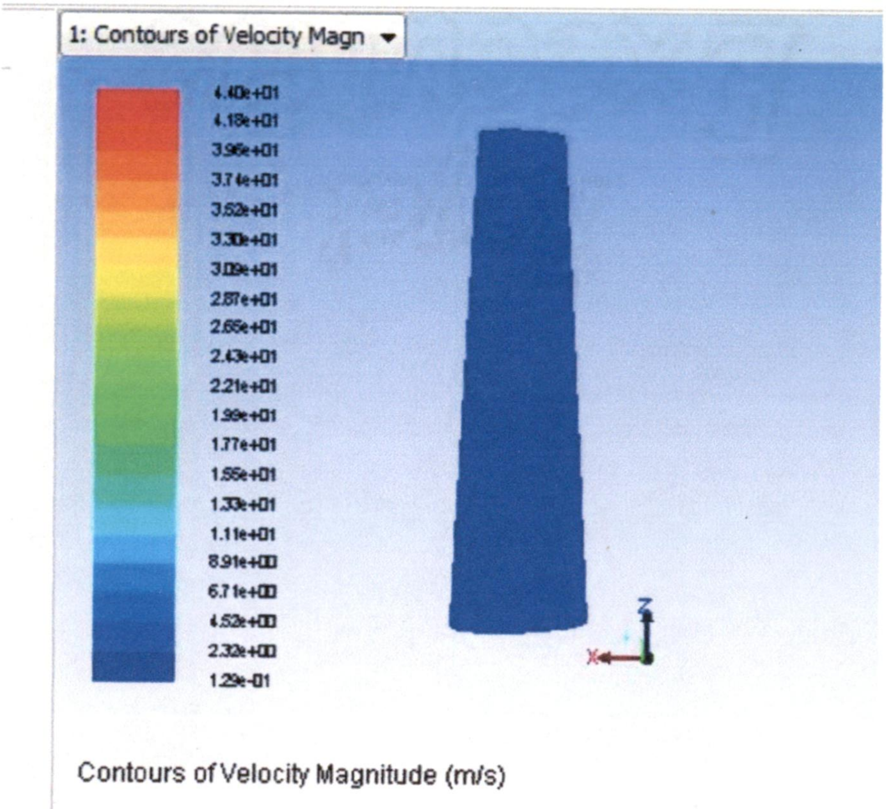

Fig. 5.7 Variation of velocity in draft tube

Calculation of efficiency for design condition

- **Input for simulation**

  Given mass flow at inlet Q= 46lps= 0.046 m^3/s

Pressure at draft tube outlet p_2 = 17000 Pa

Rotational speed ω = 157.07 rad/s

- **Output parameters**

Pressure at inlet of casing p_1 = 332804.9 Pa

Torque component

Tx= -1.75 N-m

Ty= -5.23 N-m

Tz= 57.96 N-m

Calculating Total torque T (N-m) = 58.22 N-m

From equation 5.1, efficiency of modified PAT = 0.63 = 63 %

The above steps are carried out for the other flow rates by changing the guide vane position and inlet pressure combinations and the efficiency was calculated.

5.4 RESULTS

5.4.1 Input Parameters

Mass flow rate (Q) was defined at the casing inlet and total pressure (p_2) considering draft tube submergence was defined at draft tube outlet. PATimpeller was defined in moving reference frame with rotational speed (N= 1500 rpm) and casing and draft tube along with guide vane were considered in stationary reference frame. The values of input parametersare given in Table 5.2.

Table 5.2: Input parameters

S.No.	Guide vane position	Discharge , Q (m^3/s)	Angular speed, ω (rad/sec)	Pressure outlet, p_2 (Pa)
1.	30 degree	0.030	157.07	17000
2.	38 degree	0.040	157.07	17000
3.	45 degree	0.043	157.07	17000
4.	50 degree	0.046	157.07	17000
5.	55 degree	0.048	157.07	17000
6.	60 degree	0.051	157.07	17000

5.4.2 Output Parameters

Based on the boundary conditions applied in the input parameters, the mass and momentum conservation equations were solved iteratively and various output parameters were generated. The head acting on the turbine (H) is calculated based on the total pressure acting on the turbine and torque (T) acting on the turbine is calculated based on the total moment acting on the rotating runner which is a resultant of pressure and viscous moments. The values of output parameters obtained from FLUENT are given in Table 5.3.

Table 5.3:Output parameters generated by FLUENT

S.No.	Pressure at casing inlet p_1 (Pa)	Torque components			Torque, T (N-m)	Efficiency
		T_x (N-m)	T_y(N-m)	T_z(N-m)		
1.	353406.4	-2.44	-5.29	23.71	24.41	0.38
2.	345214.7	-5.21	-8.13	48.93	49.87	0.59
3.	333199.2	-2.24	-6.51	53.71	54.14	0.63
4.	332804.9	-1.75	-5.23	57.96	58.22	0.63
5.	327390.2	-3.60	-4.10	53.37	53.64	0.56
6.	298361.4	-5.26	-10.53	47.35	48.79	0.53

5.4.3 Curve for Efficiency versus Flow of PAT with Guide Vanes

Efficiency has been calculated for different mass flow rate through PAT at different angles of movable guide vanes. Fig 5.8 shows the operating characteristics Curve for modified PAT provided withmovable guide vane.It has been observed from the analysis that after providing the movable guide vanes the part load efficiency of PAT has been improved for 43-46 lps discharge.

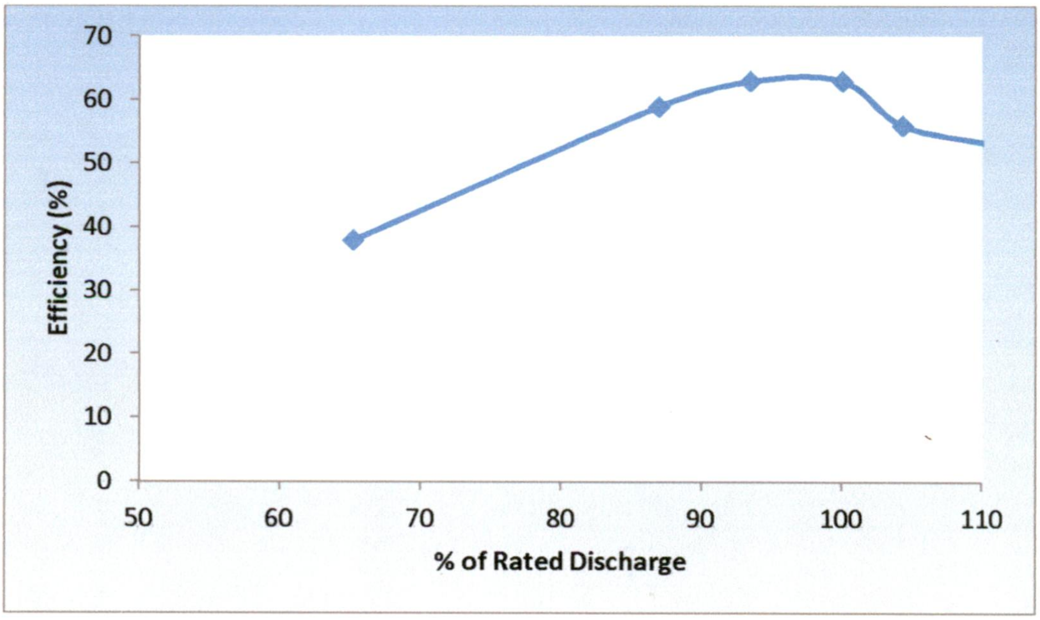

Fig. 5.8 Operating characteristics curve for modified PAT provided with movable guide vanes

5.4.4 Comparison of CFD results with Experimental results

Pandey [47] carried out experimental investigation on PAT provided with guide vane around the impeller. Table 5.4 shows the experimental results of previous study. An attempt has been made to compare the CFD results with these available experimental results. Fig 5.9 shows the comparison between experimental and CFD results.

Table 5.4: Available experimental results [47]

S. No.	Flow (lps)	Power input (kW)	Power output (kW)	Efficiency (p.u.)
1	22.16	1.45	0	0
2	23.37	1.56	0.2	0.11
3	29.71	2.2	0.8	0.4
4	33.05	3.05	1.3	0.51
5	39.32	6.01	3.2	0.63
6	42.94	6.99	3.9	0.65
7	46	9.12	5	0.65
8	47.6	9.61	5.1	0.64
9	48.9	10.15	5.2	0.62
10	49.26	11.14	5.3	0.6
11	51	11.74	5.5	0.57

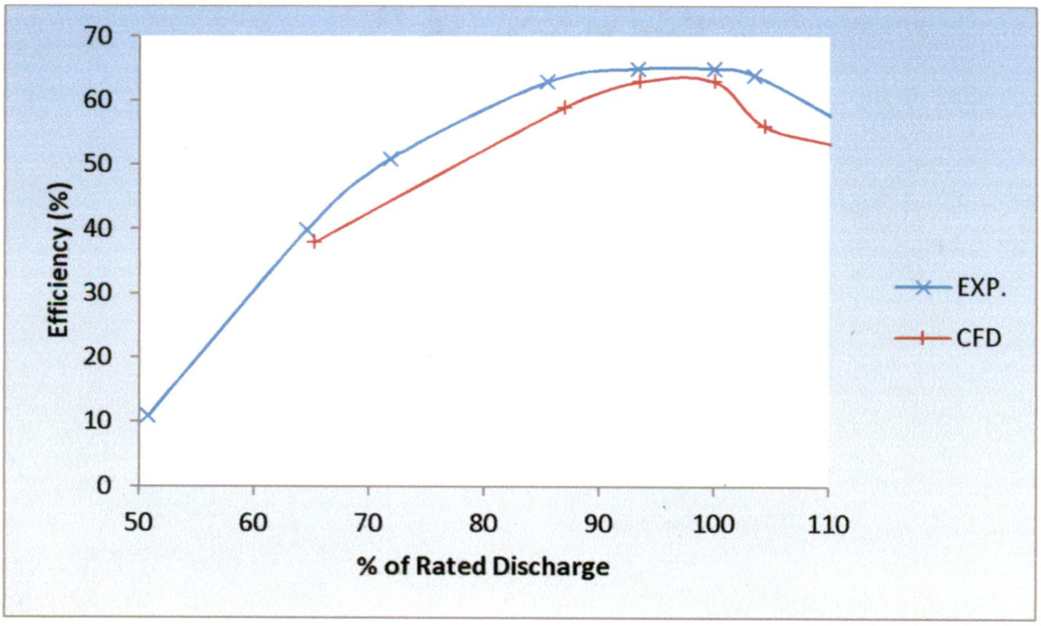

Fig. 5.9 Comparison of experimental and CFD results for modified PAT provided with movable guide vanes

Results obtain from the CFD analysis of Pump as Turbine (PAT) shows satisfactory agreement with the available experimental results. The average percentage deviation of 6.42 was observed. Reason for deviation in results can be errors in geometry creation due to complex geometry of impeller and casing and also due to quality of mesh in impeller. Thus, it can be concluded that the truncation error, round-off error, modeling error, poor quality mesh and some ideal assumptions like steady state flow, neglecting friction losses, boundary layer effect etc. are the causes of deviation between CFD results.

CHAPTER 6

CONCLUSIONS AND RECOMMENDATIONS

6.1 CONCLUSIONS

It is well known that turbines used at such micro hydro sites can easily be replaced by PAT with a number of advantages except the disadvantage of having poor part load efficiency due to absence of flow regulating mechanism in it. Therefore in order to eliminate PAT's disadvantage the centrifugal pump has been modified by providing movable guide vanes around its impeller. In this dissertation, an attempt has been made to analyze the performance of modified PAT provided with movable guide vanes using CFD technique. Following conclusions has been made from the present work of dissertation:

i. A centrifugal pump of mixed flow type having specifications as; head: 21.5 m, discharge: 42.5 lps, speed: 1500 rpm has been selected for the present investigation.

ii. Detailed dimensioning of impeller and casing for modeling purpose has been carried out. Since Draft tube is an important part for turbine operation, so the design parameters for draft tube have also been computed. Guide vanes design parameters are taken from the previous study [47].

iii. Modeling for modified PAT has been done using the software PRO-E 2.0.

iv. Flow simulation on the modified PAT has been carried out using the software ANSYS 'FLUENT'. Results for different flow rates at different guide vanes angles have been obtained.

v. The operating range of pump as turbine has been obtained at maximum part load efficiency around 63% after modifications.

vi. Operating characteristics curve have been drawn from CFD results and are compared with experimental results. It has been observed that numerical results are found to be lower than experimental results. This may be due to truncation error, round-off error, modeling error and some ideal assumptions.

6.2 RECOMMENDATIONS

As regards the future scope, the work can be directed in the following areas:

i. To test the performance of axial flow pump as turbine with adjustable flow control mechanisms

ii. To carry out CFD analysis on reverse operation of centrifugal pump with a finer mesh quality so that results can be matched to a greater extent and more accurate design analysis can be carried out

iii. To carry out techno-economic analysis between a cross flow turbine and a modified PAT based on their part load efficiency and peak efficiency

[1]. < http://www.cea.nic.in/reports/monthly/executive_rep/mar12/1-2.pdf > accessed on 25th April 2012.

[2]. www.powermin.nic.in, Ministry of power, Govt. of India accessed on 25th April 2012

[3]. Nautiyal H, Varun, Kumar Anoop, and Yadav S., "Reverse running pumps analytical, experimental and computational study: A review", Renewable and Sustainable Energy Reviews; Volume 14, Issue 7, pp. 2059–2067, 2010

[4]. National Policy For Hydro Power Development

[5]. http://www.mnre.gov.in/prog-smallhydro.htm – Ministry of New and Renewable Energy, Govt of India, accessed on 25th April 2012

[6]. Mali Sudhir, Kshirsagar JT, and Rana YS., "Kirloskar Brothers in Small Hydro", International conference on Small Hydropower-Hydro Sri Lanka; 2007

[7]. Engeda A., Strate P., and Rautenberg M., " Auswahl von Kreiselpumpen als Turbinen Pumpentagung", Sektion A6, Fachgemeinschaft Pumpen im VDMA, Frankfurt/Main, 1988

[8]. Ferziger, J.H. and Peric, M., "Computation Methods for Fluid Dynamics" Springer Private Limited, New Delhi, 3rd edition (Revised 2004)

[9]. Manual "FLUENT 6.3.26 Software and GAMBIT 6.3 Software." FLUENT Documentation, 2005

[10]. Williams A., "Pumps as Turbines Users Guide". International Technology Publications, London, ISBN 1-85339-285-5, 1995

[11]. Williams AA., "Pump as turbines for low cost micro hydro power", Renewable Energy; Volume 9, Issue 1–4, 1227–34, 1996

[12]. Derakhshan S., Nourbakhsh A., and Mohammadi B., "Efficiency improvement of centrifugal reverse pumps". ASME Journal of Fluids Engineering; Volume 131, pp. 21103–7, 2009

[13]. Derakhshan S., Mohammadi B., and Nourbakhsh A., "Incomplete sensitivities for 3D radial turbomachinery blade optimization" Computers & Fluids; Volume 37, Issue 10 , pp. 1354–63, 2008

[14]. Derakhshan S, and Nourbakhsh A., "Theoretical, numerical and experimental investigation of centrifugal pumps in reverse operation". Experimental Thermal and Fluid Science; Volume 32, Issue 8, pp. 1620-7, 2008

[15]. Joshi S., Holloway AGL., Chang L., and Kojabadi HM., "Development of a standalone micro-hydro system using pump as turbine technology for low head sites in remote areas", 20th international power system conference; 2005

[16]. Swanson MM., "Complete characteristics circle diagram for turbo machinery". Transactions of ASME; Volume 75, pp. 816–26, 1953

[17]. Chan TF., "Self excited induction generators driven by regulated and unregulated turbines" IEEE Transactions on Energy Conversion; Volume 11, 1996

[18]. Derakhshan S., and Nourbakhsh A., "Experimental study of characteristic curves of centrifugal pumps working as turbines in different specific speeds" Experimental Thermal and Fluid Science; Volume 32, Issue 3, pp. 800–7, 2008

[19]. Ramos H., and Borga A., "Pump yielding power", Dam Engineering; Volume 10, pp. 197–217, 2000

[20]. Williams AA., "The turbine performance of centrifugal pumps: a comparison of prediction methods" Proceedings of the IMechE Part A: Journal of Power and Energy; Volume 208, pp. 59–66, 1994

[21]. Schmiedl E., "Serien-Kreiselpumpen im Turbinenbetrieb. Pumpentagung" Karlsruhe, 1988

[22]. Grover KM., "Conversion of pumps to turbines", GSA Inter Corp.; New York, 1980

[23]. Lewinsky Keslitz HP., "Pumpen als Turbinen fur Kleinkraftwerke. Wasserwirtschaft"; Volume 77, No. 10, pp. 531–7, 1987

[24]. Hancock JW., "Centrifugal Pump or Water Turbine", Pipe Line News; Volume 25, Issue 7, 1963

[25]. Childs SM., "Convert pumps to turbines and recover HP", Hydro Carbon Processing and Petroleum Refiner; Volume 41, No. 10, pp. 173–4, 1962

[26]. Sharma K., "Small hydroelectric project-use of centrifugal pumps as turbines", Technical report, Kirloskar Electric Co. Bangalore, India, 1985

[27]. Stepanoff AJ., "Centrifugal and axial flow pumps". John Wiley, New York, 1957.

[28]. Alatorre-Frenk C., "Cost minimization in micro hydro systems using pumps-as-turbines", PhDME Thesis; University of Warwick, UK; 1994

[29]. Ramos H., and Borga A., "Pumps as turbines: an unconventional solution to energy production", Urban Water; Volume 1, Issue 3, pp. 261–3, 1999

[30]. Arriaga M., "Pump as turbine – A pico-hydro alternative in Lao People's Democratic Republic", Renewable Energy; Volume 35, Issue 5, pp. 1109–1115, 2010

[31]. Singh P., and Nestmann F., "Internal hydraulic analysis of impeller rounding in centrifugal pumps as turbines", Experimental Thermal and Fluid Science; Volume 35, Issue 1, pp. 121–134, 2011

[32]. Raja WA. and Piazza RW., "Reverse running centrifugal pumps as hydraulic power recovery turbines for sea water reverse osmosis systems", Desalination; Volume 38, pp. 123–34, 1981

[33]. Fernandez J., Blanco E., Parrondo J., Stickland MT., and Scanlon TJ., "Performance of a centrifugal pump running in inverse mode", Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy; Volume 218, No. 4, pp. 265-271, 2004

[34]. Joshi S., Holloway AGL., and Chang L., "Selecting a high specific speed pump for low head hydro-electric power generation", IEEE Transactions; pp. 603–606, 2005

[35]. Suarda M., Suarnadwipa N., and Adnyana WB., "Assessment Performance of Pumps as Hydro-Turbines", International Conference on Energy and Environment; 2006 (ICEE 2006)

[36]. Nautiyal H., Varun, Kumar Anoop, and Yadav S., "Experimental Investigation of Centrifugal Pump Working as Turbine for Small Hydropower Systems", Energy Science and Technology; Volume 1, Issue 1, pp. 79-86, 2011

[37]. Singh P., and Nestmann F., "An optimization routine on a prediction and selection model for the turbine operation of centrifugal pumps", Experimental Thermal and Fluid Science; Volume 34, Issue 2, pp. 152–164, 2010

[38]. Natanasabapathi SR., and Kshirsagar JT., "Pump as turbine—an experience with CFX- 5.6", Corporate Research and Eng. Division, Kirloskar Bros. Ltd.; 2004 :< www.ansys.com/event/proceedings/2004/PAPERS/127.PDF>

[39]. Rawal S., and Kshirsagar JT., "Numerical simulation on a pump operating in a turbine mode", Kirloskar Brothers India Limited, In: Proceedings of the 23rd International Pump Users Symposium; 2007

[40]. Nautiyal H., Varun, Kumar Anoop, and Yadav S., "CFD Analysis on Pumps Working as Turbines", HYDRO NEPAL; Issue 6, pp. 35-37, 2010

[41]. Rodrigues A., Singh P., Williams A., Nestmann F., and Lai E., "Hydraulic analysis of a pump as a turbine with CFD and experimental data". In: IMechE seminar computational fluid dynamics for fluid machinery; 2003

[42]. Barrio R., Fernandez J., Parrondo J., and Blanco E., "Performance prediction of a centrifugal pump working in direct and reverse mode using Computational Fluid

Dynamics" International Conference on Renewable Energies and Power Quality (ICREPQ'10), Granada, Spain, March, 2010

[43]. Fecarotta O., Carravetta A., and Ramos HM., "CFD and Comparisons for a Pump As Turbine: Mesh Reliability And Performance Concerns", International Journal of Energy and Environment; Volume 2, Issue 1, pp.39-48, 2011

[44]. Sedlar M., Soukal J., and Komarek M., "CFD Analysis of Middle Stage of Multistage Pump operating in Turbine regime", Engineering MECHANICS; Vol. 16, No. 6, pp. 413–421, 2009

[45]. Suarda M., Suarnadwipa N., Adnyana WB., "Experimental work on the modification of impeller tips of a centrifugal pump as a turbine", 2nd joint International Conference on Sustainable Energy and Environment; 2006

[46]. Singh P., "Optimization of the Internal Hydraulic and of System Design in Pumps as turbines with Field Implementation and Evaluation", PhD Thesis, University of Karlsruhe, Germany, 2005

[47]. Pandey Rahul, "Performance analysis of modified pump used as turbine", M.Tech. Dissertation; AHEC, IIT Roorkee, 2008

[48]. Saini, R.P., and Ahmad N., "Method of Selection of pump used in turbine mode" First International conference on renewable energy – small hydro; Hyderabad, India; 3-7 February 1997,

[49]. Chappell J.R, Hickman W.W., and Seegmiller D. S., "Pumps as Turbine- Experience Profile" Acc. 02, IDO-10109, EG & G Idaho Inc., pp. 5- 6, ., September 1982

[50]. Kittredge, C.P., and Thoma, D., "Centrifugal Pumps Operated under Abnormal Conditions", Power; pp. 881-884, 1931

[51]. Chappell J.R, Hickman W.W., and Seegmiller D. S., "Historical Background- Pumps as Turbines Experience Profile", Acc. 02, IDO-10109, EG & G Idaho Inc., pp. 3-4, September 1982

[52]. Buse F., "Using Centrifugal Pumps as Hydraulic Turbines- Pumps as Turbines Experience Profile", Acc. 02, IDO-10109, EG & G Idaho Inc., pp. C-1-1 to C-1-5, September1982

[53]. Chappell J.R, Hickman W.W., and Seegmiller D.S., "Pump As Turbine (Hydraulic) Pumps as turbines Experience Profile", Acc. 02, IDO-10109, EG & G Idaho Inc., pp. 56, September 1982

[54]. Arthur Williams "Pumps As Turbine, A User 's Guide" ITDG Publication London; Second edition, ISBN 1-85339-285-5, pp. 2-3, 1995

[55]. Arthur Williams "Pumps As Turbine, A User's Guide" ITDG Publication London; Second edition, ISBN l-85339-285-5, pp. 1, 1995

[56]. Tamm A., Braten A., Stoffel B., and Ludwig G., "Analysis of a standard pump in reverse operation using CFD", 20th IAHR-Symposium; Charlotte North Carolina USA, paper No. PD-05, 2000

[57]. Saini, R.P., "Selection Procedure of Centrifugal Pump to Use as Turbine for Small Hydro Power Stations", National Seminar on Small Hydro Power Projects; CIT, Coimbatore, paper no.303, 1998

[58]. Thin KC., Khaing MM., and Aye KM., "Design and Performance Analysis of Centrifugal Pump", World Academy of Science, Engineering and Technology; Volume 46, pp. 422-429, 2008

[59]. IS: 12800 Part-I: "Guidelines For Selection Of Turbines, Preliminary Dimensioning and Layout of Surface Hydro-Electric Powerhouses (PART 1 MEDIUM AND LARGE POWER)", 1993

[60]. Jain S., Saini RP., & Kumar A., "CFD approach for prediction of efficiency of Francis turbine" IGHEM ; AHEC, IIT Roorkee , 2010,

1. Kumar Anuj, Saini R.P., Singhal M.K., "CFD Analysis for Performance of Modified Pump used as Turbine" National Conference on Emerging Trends in Mechanical Engineering, ITM University, Gurgaon, India, pp. 450- 459, 1st June 2012
2. Kumar Anuj, Saini R.P., Singhal M.K., "A Review on Pump used as Turbine" Water and Energy International Journal (CBIP) - Communicated