Issue 9 vol 3 ISSN: 2321-8134

IJFEAT

INTERNATIONAL JOURNAL FOR ENGINEERING APPLICATIONS AND TECHNOLOGY

DESIGN AND FLOW ANALYSIS OF RADIAL TURBINE

Vivek D. Vattamwar[1], Swapnil S. Nishane[2], Suraj A. Sewatkar[3]

[1]*Asst. Professor, Mechanical Engineering Department, JCOET, MS, India,* ***vivekvattamwar4@gmail.com***

[2]*Asst. Professor, Mechanical Engineering Department, JCOET, MS, India* ***swapnilnishane9420@gmail.com***

[3]*Asst. Professor, Mechanical Engineering Department, JCOET, MS, India,* ***suraj11345@gmail.com***

Abstract

Hydraulic energy is a clean, safe because it requires water. Hydraulic turbines are the machine which converts hydraulic energy into electricity. Mechanical efficiencies of these turbo machine are quite well that is 95%. However reaching such efficiencies is a difficult task and it require a high engineering effort because hydraulic turbine are usually unique product which must be designed for determined local condition that is head and discharge. For this reason for each component of the machine a specific design is needed. The traditional design process is based on experiments, measurements and model tests which is too much time and money consuming. But for last few decades CFD simulation method is developed which is less time consuming and save money also. CFD simulation is done to study the flow behaviour of fluids inside or outside of structure. It helps to make the necessary changes to improve the design. For the economical design of turbine it is very important to understand the flow characteristics indifferent parts of turbine that is how energy transfer and transformation take place in different parts which help in analyzing their performance in advance before manufacturing them. Instead of detailed analytical treatment & using complicated velocity triangles to study the performance of a given turbine, it is recommended to carry out the CFD analysis for the same. The current work deals with the CFD analysis of radial turbine at various guide vane angles & study of its effect on the turbine performance.

Index Term: *Radial turbine, CFD, Turbine performance.*

--- *** ---

1. INTRODUCTION

1.1. Introduction of Radial Turbine

Reaction turbines are acted on by water, which changes pressure as it moves through the turbine and gives up its energy. They must be encased to contain the water pressure (or suction), or they must be fully submerged in the water flow. Newton's third law describes the transfer of energy for reaction turbines. Most water turbines in use are reaction turbines and are used in low (<30 m or 100 ft) and medium (30–300 m or 100–1,000 ft) head applications. In reaction turbine pressure drop occurs in both fixed and moving blades. It is largely used in dam and large power plants.(15)

The Francis turbine is a type of reaction turbine, a category of turbine in which the working fluid comes to the turbine under immense pressure and the energy is extracted by the turbine blades from the working fluid. A part of the energy is given up by the fluid because of pressure changes occurring in the blades of the turbine, quantified by the expression of degree of reaction, while the remaining part of the energy is extracted by the volute casing of the turbine. At the exit, water acts on the spinning cup-shaped runner features, leaving at low velocity and low swirl with very little kinetic or potential energy left. The turbine's exit tube is shaped to help decelerate the water flow and recover the pressure. A Francis turbine consists of the following main parts:

1.2. Spiral casing: The spiral casing around the runner of the turbine is known as the volute casing or scroll case. All throughout its length, it has numerous openings at regular intervals to allow the working fluid to impound on the blades of the runner. These openings convert the pressure energy of the fluid into momentum energy just before the fluid impound on the blades to maintain a constant flow rate despite the fact that numerous openings have been provided for the fluid to gain entry to the blades, the cross-sectional area of this casing decreases uniformly along the circumference.

1.3 Guide or stay vanes: The guide vanes consist of number of blades that can be adjusted in order to increase or reduce the flow rate through the turbine. The vanes are arranged between two parallel covers normal to the turbine shaft. The primary function of the guide or stay vanes is to convert the pressure energy of the fluid into the momentum energy. It also serves to direct the flow at design angles to the runner blades. The basic Purpose of the stay vanes & guide vanes is to convert a part of pressure energy of the fluid at its entrance to the kinetic energy and then to direct the fluid on to the runner blades at the angle appropriate to the design.

- Moreover, the guide vanes are pivoted and can be turned by a suitable governing mechanism to regulate the flow while the load changes.
- The guide vanes are also known as wicket gates.
- The guide vanes impart a tangential velocity and hence an angular momentum to the water before its entry to the runner.
- The guide vanes are constructed using an optimal aerofoil shape, in order to optimize off-design performance.

1.4 Runner blades: Runner blades are the heart of any turbine as these are the centers where the fluid strikes and the tangential force of the impact causes the shaft of the turbine to rotate and hence electricity is produced. In this part one has to be very careful about the blade angles at inlet and outlet as these are the major parameters affecting the power production.

1.5 Draft tube: The draft tube is a conduit which connects the runner exit to the tail race where the water is being finally discharged from the turbine. The primary function of the draft tube is to reduce the velocity of the discharged water to minimize the loss of kinetic energy at the outlet. This permits the turbine to be set above the tail water without any appreciable drop of available head.(18)

2. MODELLING & SIMULATION

2.1 Modelling of Turbine using CATIA

3-D model is created for CFD analysis by using CATIA V-5. The procedure for the creation for model is explained here.

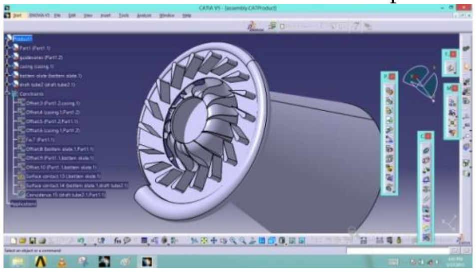

Fig-1: Assembly of Radial Turbine

The assembly model is created in the CATIA V 5 in assembly design workbench.

Start---- mechanical design ---- assembly design

Assembly is formed using the 4 parts namely

Runner, Guide vanes , Casing , Draft tube

Before assembly design we will create a separate part in part design.

2.2) Runner

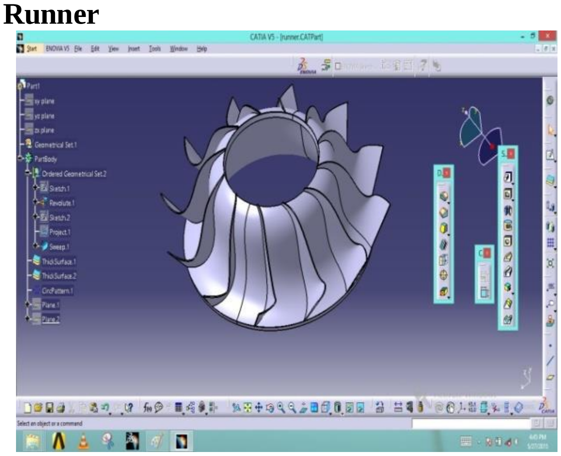

Fig-2: C/S View of Runner

The runner is mounted on a shaft and the blades are fixed on the runner at equal distances. The vanes are so shaped that the water reacting with them will pass through them thereby passing their pressure energy to make it rotate the runner.

2.3. Guide vane

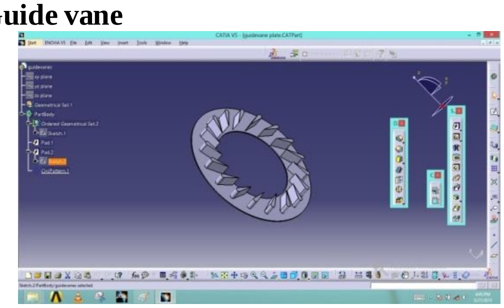

Fig-3: C/S Guide View

At inlet the water flows into the wheel at the centre and then glides through radially provided fixed guide vanes and then flows over the moving vanes. The function of the guide vanes is to direct or guide the water into the moving vanes in the correct direction and also regulate the amount of water striking the vanes.

2.4 . Casing

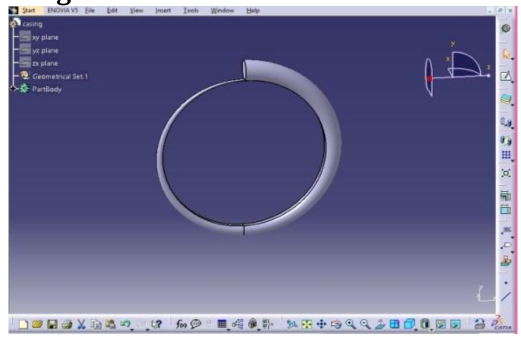

Fig-4: C/S View of Casing

This is a tube of decreasing cross-sectional area with the axis of the tube being of geometric shape of volute or a spiral. The water first fills the casing and then enters the guide vanes from all sides radially inwards. The decreasing cross-sectional area helps the velocity of the entering water from all sides being kept equal. The geometric shape helps the entering water avoiding or preventing the creation of eddies

2.5) Draft tube

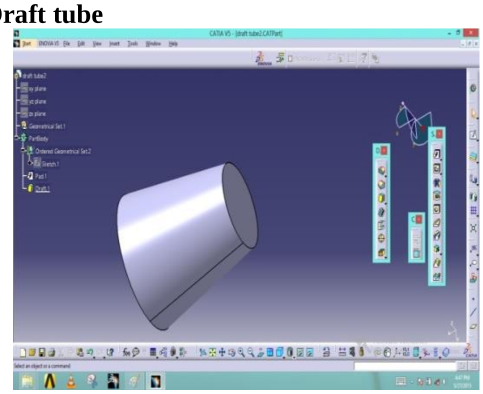

Fig-5: Draft tube of Turbine

This is a divergent tube fixed at the end of the outlet of the turbine and the other end is submerged under the water level in the tail race. The water after working on the turbine, transfers the pressure energy there by losing all its pressure and falling below atmospheric pressure. The draft tube accepts this water at the upper end and increases its pressure as the water flows through the tube and increases more than atmospheric pressure before it reaches the tailrace.

3. CFD ANALYSIS OF TURBINE

3.1) CFD Analysis for Guide Vane Angle of 10°

Pressure Contours:

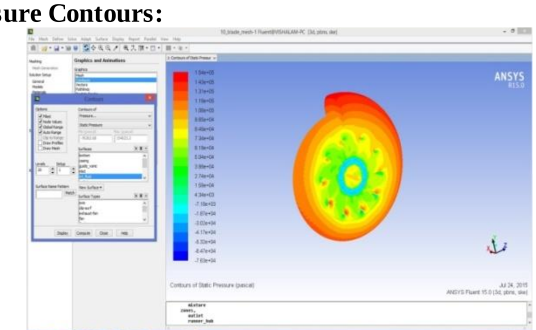

**Fig-6: CFD Analysis for Guide Vane Angle of 10°
Pressure Contours**

Velocity Contours:

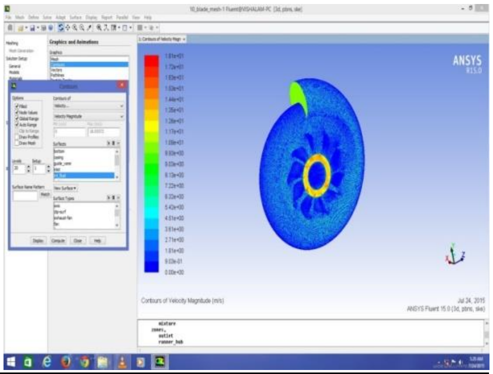

Fig-7: CFD Analysis for Guide Vane Angle of 10°
Velocity Contours

From the velocity contours the maximum flow velocity is i.e. $V_{f1} = 18.1$ m/s

$V_1 = V_{f1}/\sin(\alpha)$

$V_1 = 104.23$ m/s

$V_{w1} = V_1 \cos(\alpha)$

$= 102.64$ m/s

$\tan(\phi) = V_{f2} / u_2$

$\Phi = 85^o$

$V_{f1}/ = V_{f2}$

$u_2 = 1.58$ m/s

$u_2 = \frac{\pi D_2 N}{60}$

$1.58 = \frac{\pi * 0.311 * N}{60}$

N=97.07 rpm

$u_1 = \frac{\pi D_1 N}{60}$

$u_1 = \frac{\pi * 0.5 * 97.07}{60}$

$u_1 = 2.54$ m/s

Power developed (P) = $\rho * Q * Vw_1 * u_1$

P = 902.04 KW

Simulation Error

% error = (Analytical – CFD) /Analytical

% error = (1355.66 – 902.04) / 1355.66

% error = 33.46 %

3.2. CFD Analysis for Guide Vane Angle of 15°

Pressure Contours:

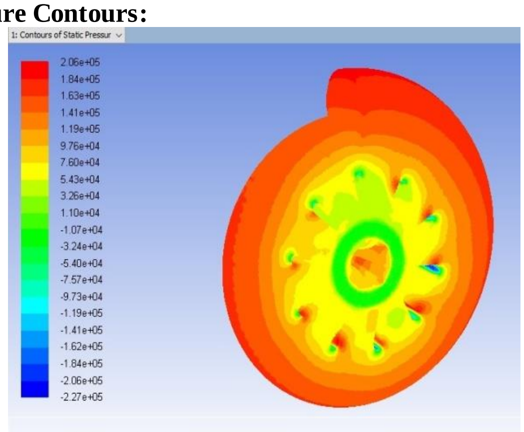

Fig-8: CFD Analysis for Guide Vane Angle of 15°
Pressure Contours

Velocity Contours:

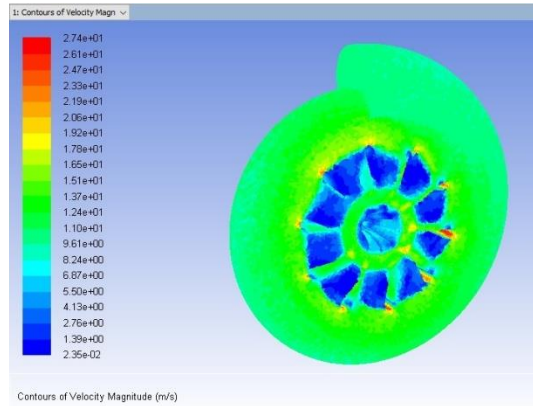

Fig-9: CFD Analysis for Guide Vane Angle of 15°
Velocity Contours

From the velocity contours the maximum flow velocity is i.e. $V_{f1} = 19.2$ m/s

$V_1 = V_{f1}/\sin(\alpha)$

$V_1 = 74.19$ m/s

$V_{w1} = V_1 \cos(\alpha)$

$= 71.66$ m/s

$\tan(\phi) = V_{f2} / u_2$

$\Phi = 85^o$

$V_{f1}/ = V_{f2}$

$u_2 = 1.67$ m/s

$u_2 = \frac{\pi D_2 N}{60}$

$1.67 = \frac{\pi * 0.311 * N}{60}$

N= 102.67 rpm

$u_1 = \frac{\pi D_1 N}{60}$

$u_1 = \frac{\pi * 0.5 * 102.67}{60}$

$u_1 = 2.68$ m/s

Power developed (P) = $\rho * Q * Vw_1 * u_1$

P = 664.48 KW

Simulation Error

% error = (Analytical – CFD) /Analytical

% error = (878.62 – 664.48) / 878.62

% error = **24.37 %**

3.3 CFD Analysis for Guide Vane Angle of 20°

Pressure Contours:

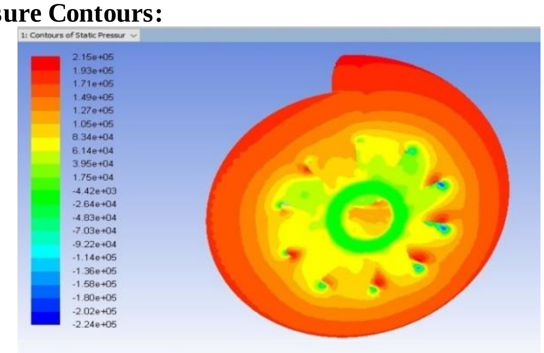

Fig-10: CFD Analysis for Guide Vane Angle of 20°
Pressure Contours

Velocity Contours:

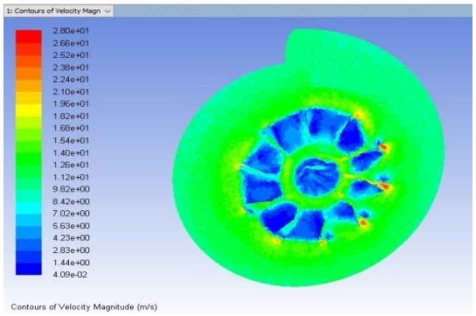

Fig-11: CFD Analysis for Guide Vane Angle of 20^o Velocity Contours

From the velocity contours the maximum flow velocity is i.e. $V_{f1} = 19.6$ m/s

$V_1 = V_{f1}/\sin(\alpha)$

$V_1 = 57.3$ m/s

$V_{w1} = V_1 \cos(\alpha)$

$= 53.84$ m/s

$\tan(\phi) = V_{f2} / u_2$

$\Phi = 85^o$

$V_{f1} / = V_{f2}$

$u_2 = 1.71$ m/s

$$u_2 = \frac{\pi D_2 N}{60}$$

$$1.71 = \frac{\pi * 0.311 * N}{60}$$

N= 105.35 rpm

$$u_1 = \frac{\pi D_1 N}{60}$$

$$u_1 = \frac{\pi * 0.5 * 105.35}{60}$$

$u_1 = 2.75$ m/s

Power developed (P) = $\rho * Q * Vw_1 * u_1$

P = 513.57 KW

Simulation Error

% error = (Analytical – CFD) /Analytical

% error = (646.62 – 513.57) / 646.62

% error = **20.57 %**

3.4 CFD Analysis for Guide Vane Angle of 30^o

Pressure Contours

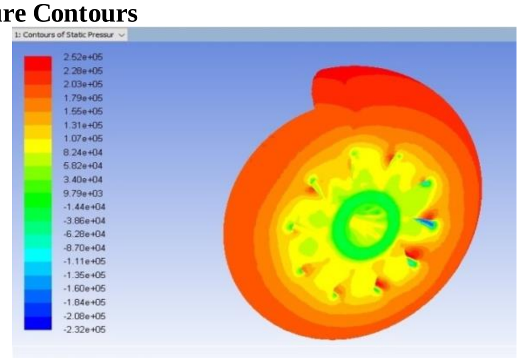

Fig-12: CFD Analysis for Guide Vane Angle of 30^o Pressure Contours

Velocity Contours:

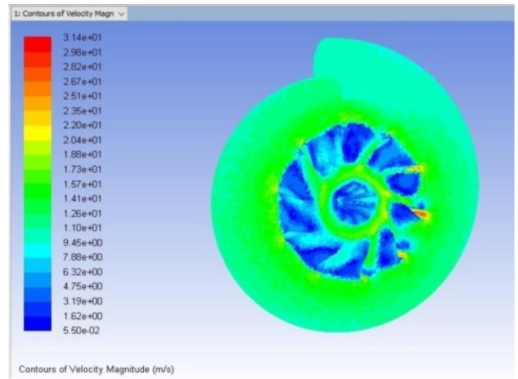

Fig-13: CFD Analysis for Guide Vane Angle of 30^o Velocity Contours

From the velocity contours the maximum flow velocity is i.e. $V_{f1} = 20.4$ m/s

$V_1 = V_{f1}/\sin(\alpha)$

$V_1 = 40.8$ m/s

$V_{w1} = V_1 \cos(\alpha)$

$= 35.33$ m/s

$\tan(\phi) = V_{f2} / u_2$

$\Phi = 85^o$

$V_{f1} / = V_{f2}$

$u_2 = 1.78$ m/s

$$u_2 = \frac{\pi D_2 N}{60}$$

$$1.78 = \frac{\pi * 0.311 * N}{60}$$

N= 109.65 rpm

$$u_1 = \frac{\pi D_1 N}{60}$$

$$u_1 = \frac{\pi * 0.5 * 109.65}{60}$$

$u_1 = 2.87$ m/s

Power developed (P) = $\rho * Q * Vw_1 * u_1$

P = 350.76 KW

Simulation Error

% error = (Analytical – CFD) /Analytical

% error = (407.66 – 350.76) / 407.66

% error = **13.95 %**

3.5 CFD Analysis for Guide Vane Angle of 35^o

Pressure Contours:

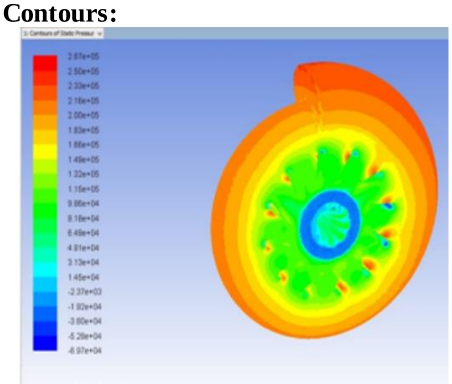

Fig-14: CFD Analysis for Guide Vane Angle of 35^o Pressure Contours

Velocity Contours:

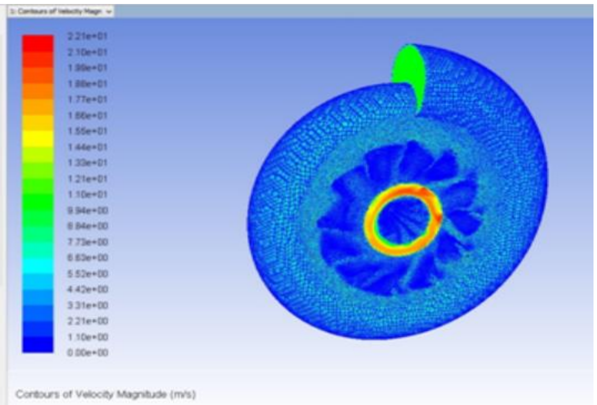

Fig-15 CFD Analysis for Guide Vane Angle of 35°

Velocity Contours

From the velocity contours the maximum flow velocity is

i.e. $V_{f1} = 22.1$ m/s

$V_1 = V_{f1}/\sin(\alpha)$

$V_1 = 38.53$ m/s

$V_{w1} = V_1 \cos(\alpha)$

$= 31.56$ m/s

$\tan(\phi) = V_{f2} / u_2$

$\Phi = 85^o$

$V_{f1} / = V_{f2}$

$u_2 = 1.933$ m/s

$$u_2 = \frac{\pi D_2 N}{60}$$

$$1.933 = \frac{\pi * 0.311 * N}{60}$$

N=118.79 rpm

$$u_1 = \frac{\pi D_1 N}{60}$$

$$u_1 = \frac{\pi * 0.5 * 118.79}{60}$$

$u_1 = 3.10$ m/s

Power developed (P) = $\rho * Q * Vw_1 * u_1$

P = 339.45 KW

Simulation Error

% error = (Analytical – CFD) /Analytical

% error = (337.14 – 339.45) / 337.14

% error = **0.7 %**

4. RESULTS & CONCLUSION

Table-4.1: Numerical Analysis Results

Sr. No.	Guide Vane Angle (°)	Power Output (KW)
1	10	1355.66
2	15	878.62
3	20	646.62
4	30	407.66
5	35	337.14

Table-4.2: CFD Analysis Results

Sr. No.	Guide Vane Angle (°)	Power Output (KW)
1	10	902.04
2	15	664.48
3	20	513.57
4	30	350.76
5	35	339.45

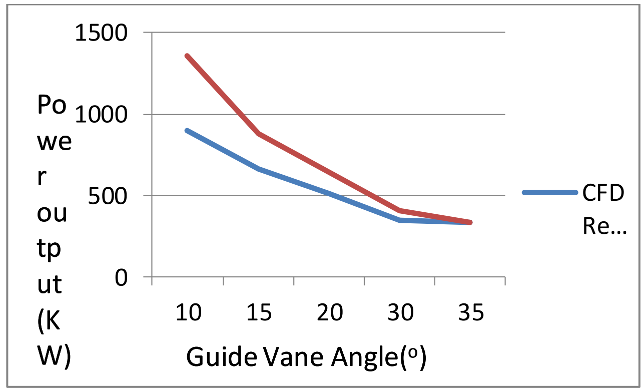

Fig-15: Comparison of Numerical & CFD Results

It can be seen from above tables & graph, that the power produced by the turbine is different when the flow dynamics is considered. The maximum power is produced for the lower values of guide vane angle.

4.4 Error Analysis

Sr. No.	Guide Vane Angle (°)	Error in Numerical & CFD Results (%)
1	10	33.46
2	15	24.37
3	20	20.57
4	30	13.95
5	35	0.7

5. CONCLUSION

Numerical & CFD analysis is carried out for a radial turbine. A numerical result shows that the maximum velocity of the flowing fluid is a function of some fixed parameters, thereby resulting in the same numerical value. From numerical analysis it can be seen that the power developed by the turbine is a function of guide vane angle, showing that the maximum power is developed at minimum guide vane angle. Whereas, actual flow conditions in the turbine are very complex in nature due to dynamic behaviour of the fluid. Hence the CFD analysis using ANSYS Fluent is carried out to predict the performance of the turbine at different guide vane angles. It is found that the maximum velocity of the fluid varies with the change in guide vane angle, thereby resulting in variation of the power developed by the turbine. It can be observed that the actual power produced by the turbine is different than that of the numerically calculated values. The necessary measures can be taken to predict the dynamic performance of the turbine using CFD analysis tool. The suggested CFD methodology for turbine analysis is validated against the numerical method. The error in the CFD results are well within limits thereby validating the CFD process for the turbine analysis.

REFERENCES

[1] Tarun Singh Tanwar, Dharmendra Hariyani &Manish Dadhich, "Flow Simulation (CFD) & Static Structural Analysis Of A Radial Turbine", IJMET Journal, Vol. 3, Issue 3, Sept-Dec 2014.

[2] Fredrik Hellstrom, "Numerical Computations Of The Unsteady Flow In A Radial Turbine", Technical Reports From Royal Institute Of Technology Sweden 2008.

[3] M.G.Patel, A.V.Doshi, "Effect Of Impeller Blade Exit Angle On The Performance Of Centrifugal Pump", International Journal Of Immerging Technology & Advanced Engineering, Volume 3, Issue 1, Jan 2013.

[4] Samip Shah, Gaurang Chaudhri, Digvijay Kulshreshtha & S. A. Channiwala, "Effect Of Flow Coefficient & Loading Coefficient On The Radial Inflow Turbine Impeller Geometry", International Journal Of Engineering & Technology", Volume 2, Issue 2, Feb 2013.

[5] S.Rajendran, Dr.K.Purushothaman, "Analysis Of A Centrifugal Pump Using ANSYS CFX", International Journal Of Engineering Research & Technology, Volume 1, Issue 3, May 2012.

[6] Sehyun Shin, In-Cheol Bae, In-Sik Joo, Hong-Hae Hong & Tae-Kyung Lee, "The Effect Of Blade Geometry On The Performance Of Automatic Torque Converter", FISITA World Automotive Conference Korea, June 2000.

[7] M. Odabaee, M. Modir Shanechi And K. Hooman, "CFD Simulations & FE Analysis Of A High Pressure Ratio Radial Inflow Turbine", 19th Australasian Fluid Mechanics Conference Melbourne Australia, 8-11 December 2014.

[8] Diego Silva De Carvalho & Jesuíno Takachi Tomita, Kateris D.Tsiropoulos Z, "3D Turbulent Flow Analysis In A Turbocharger Radial Turbine Operating At Design Point", 22nd International Congress Of Mechanical Engineering Brazil, Nov 2013.

[9] Naveen B & Kallu Raja Sekhar, "CFD Analysis Of Turbocharger Turbine", International Journal Of IT & Engineering, Volume 3, Issue 4, Apr 2015.

[10] D. Pablo Fajardo, "Methodology For The Numerical Characterization Of A Radial Turbine Under Steady & Pulsating Flow", Doctoral Thesis, Valencia July 2012.