Review On Design and Flow Analysis of Radial Flow Turbine

Mr. Anand .V. Karandikar[1], Prof. A.A.Kanaskar[2], Prof. S.S.Jawre[3]
[1]*Department of Mechanical Engineering S.P.Agnihotri College of Engineering, Wardha,*
[2]*Asst. prof, Department of Mechanical Engineering S.P.Agnihotri College of Engineering, Wardha,*
[3]*Asst. prof, Department of Mechanical Engineering S.P.Agnihotri College of Engineering, Wardha,*

Abstract- Hydraulic energy is clean, safe because it required water. Hydraulic turbines are the machine which converts hydraulic energy into electricity. Mechanical efficiency of this turbo machine is quite well that 95%. However reaching such efficiency is difficult task and it require a high engineering effort because hydraulic turbines are usually unique product which must be designed for determined local conduction that is head and discharge. For this reason for each component of machine a specific design is needed. The traditional design process is based on experiments, measurement and model test s which is too much time consuming and costly. But for last decades CFD simulation is done to make necessary change to improve the design .From the economical design of turbine it is very important to understand the flow characteristics indifferent part which help in analyzing their performance advance before manufacturing them. Instead of detailed analytical treatment and using complicated velocity triangle diagram 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 and study of its effect on the turbine performance

Keyword- Radial Turbine, CFD, Turbine Performance

I. INTRODUCTION

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.

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.

II. SCOPE OF WORK

- To study effect of variation of guide vane angle on the performance of radial turbine.
- To carry out numerical analysis of the radial turbine.
- To carry out CFD analysis of the radial turbine.
- To validate the CFD process by comparing with the numerical results.

III. LITERATURE SURVEY

Tarun Singh Tanwar, DharmendraHariyani&Manish Dadhich [1] describes the fluid flow conditions and parameters within a Radial Turbine with regards to each part of the turbine in contact with the working fluid and all working parts of the Radial turbine. The process of obtaining the fluid flow condition and characteristic within the turbine is done by Computational Fluid Dynamics (CFD) simulation with the help of Ansys. In this present work, flow behaviour is observed inside the turbine at different guide vane angles and got the maximum Hydraulic efficiency for all cases and comparison is done between theoretical and experimental (CFD) efficiency. Characteristic curve is verified for all the different guide vane angles. For every case inlet velocity is changed as 10m/s, 7m/s, 4 m/s. CFD simulation is done with K-ω (SST) model and simplec algorithm. Before CFD simulation is done, a model of the Radial turbine needs to be selected as there are wide ranges of model ranging from conventional usage. After getting the best efficient model of radial turbine through CFD simulation, structural analysis would be done for runner and guide vanes. Gerber zero based model is used for the static structural analysis.

Fredrik Hellstr¨om [2] studied Non-pulsatile and pulsatile flow in bent pipes and radial turbine has been assessed with numerical simulations. The flow field in a single bent pipe has been computed with diff erent turbulence modelling approaches. A comparison with measured data shows that Implicit Large Eddy Simulation (ILES) gives the best agreement in terms of mean flow quantities. All computations with the diff erent turbulence models qualitatively capture the so called Dean vortices. The Dean vortices are a pair of counter-rotating vortices that are created in the bend, due to inertial eff ects in combination with a radial pressure gradient. The pulsatile flow in a double bent pipe has also been considered. In the first bend, the Dean vortices are formed and in the second bend a swirling motion is created, which will together with the Dean vortices create a complex flow field downstream of the second bend. The strength of these structures will vary with the amplitude of the axial flow. For pulsatile flow, a phase shift between the velocity and the pressure occurs and the phase shift is not constant during the pulse depending on the balance between the diff erent terms in the Navier- Stokes equations. The performance of a radial turbocharger turbine working under both non-pulsatile and pulsatile flow conditions has also been investigated by using ILES. To assess the eff ect of pulsatile inflow conditions on the turbine performance, three diff erent cases have been considered with diff erent frequencies and ampli-tude of the mass flow pulse and diff erent rotational speeds of the turbine wheel. The results show that the turbine cannot be treated as being quasi-stationary; for example, the shaft power varies with varying frequency of the pulses for the same amplitude of mass flow. The pulsatile flow also implies that the incidence angle of the flow into the turbine wheel varies during the pulse. For the worst case, the relative incidence angle varies from approximately -80° to $+60^{\circ}$. A phase shift between the pressure and the mass flow at the inlet and the shaft torque also occurs. This phase shift increases with increasing frequency, which aff ects the accuracy of the results from 1-D models based on turbine maps measured under non-pulsatile conditions. For a turbocharger working under internal combustion engine conditions, the flow into the turbine is pulsatile and there are also unsteady secondary flow components, depending on the geometry of the exhaust manifold situated upstream of the turbine. Therefore, the eff ects of diff erent perturbations at the inflow conditions on the turbine performance have been assessed. For the diff erent cases both turbulent fluctuations and diff erent secondary flow struc-tures are added to the inlet velocity. The results show that a non-disturbed inlet flow gives the best performance, while an inflow condition with a certain large scale eddy in combination with turbulence has the largest negative eff ect on the shaft power output.

M.G.Patel, A.V.Doshi [3] studied the effect of changing some geometric characteristic of the impeller in centrifugal pumps improves their performance. It is known that blade exit angle plays very important role in the performance of a centrifugal pump. To investigate effect of blade exit angle on the performance of centrifugal pump by means of experiment is very expensive and lengthy process. Due to expensive and lengthy process, it can be obtained by using mathematical model. In the present study three pumps of different specific speeds are taken for the investigation. Initially mathematical model presented by Gulich is validated with manufacturer's head-flow curve and then it is used to investigate the effect of blade exit angle. It is seen that the Gulich model is in very good agreement with the manufacturer's head-flow curve. Moreover the blade exit angle has significant effect on the head and the efficiency of the centrifugal pump. It is found from this investigation, both head and efficiency of centrifugal pump increases with increasing in blade exit angle.

Samip Shah, GaurangChaudhri, DigvijayKulshreshtha& S. A. Channiwala [4] addressed that before attempting to the design of radial inflow turbine, some of the techniques used to describe and present the effect of coefficient on geometry, need to be appreciated. The user of turbine will generally require parameters which readily describe the overall dimension of the machine so that assessments and comparisons can be easily made. The designer requires parameters which will enable him to select the correct machine and make valid comparisons between competing designs. This allows the designer to compute more easily the dimension of the machine at different coefficient, to assess the performance of a range of geometrically similar machines. A paper describes the basic design parameters and effect of coefficient on radial inflow turbine impeller geometry for 25kW application.

S.Rajendran, Dr.K.Purushothaman [5] used ANSYS software package to develop a three dimensional, fully turbulent model of the compressible flow across a complex geometry of impeller, such as those found in centrifugal pump. It is a most common pump used in industries and domestic applications. The Flow through centrifugal pump impeller is three dimensional and fully turbulence model. The present paper describes the simulation of the flow in the impeller of a centrifugal pump. The analysis of centrifugal pump impeller design is carried out using ANSYS-CFX. The complex internal flows in Centrifugal pump impellers can be well predicted through ANSYS-CFX. The numerical solution of the discredited three-dimensional, incompressible Navier-Stokes equations over an unstructured grid is accomplished with an ANSYS-CFX. The flow pattern, pressure distribution in the blade passage, blade loading and pressure plots are discussed in this paper.

Sehyun Shin, In-CheolBae, In-SikJoo, Hong-Hae Hong & Tae-kyung Lee [6] studied the performance characteristic of a torque converter strongly depends on the blade geometry, which directly affects its torque ratio and input capacity factor. The present study developed a program that design a blade system and analyze its performance characteristics. Using the design and analysis program, TorconMaster , several blade systems were newly generated and compared their performance characteristic with reference blade system. The objective of this paper is to investigate the effect of the blade geometry on the performance of a torque converter. As analysis tools, one-dimensional performance analysis were used for sensitivity analysis, where a full three-dimensional flow computation was used for investigation of three-dimensional blade geometry on performance. The analytical and numerical results obtained a refined relationship between geometry and performance. Furthermore, the results can also be used as reference data for performance enhancement in the design process of torque converter blades.

M. Odabaee, M. ModirShanechi and K. Hooman [7] investigate a coupled CFD&FE analysis of a high pressure ratio single stage radial-inflow turbine applied in the Sundstrans Power Systems T-100 Multipurpose Small Power Unit. ANSYS turbomachinary package is applied to create 3D geometry of one blade passage, including the stator, rotor and diffuser. CFD simulations are performed with ANSYS-CFX in which three-dimensional Reynolds-Averaged Navier-Stokes equations are solved subject to appropriate boundary conditions. CFD results are then imported to ANSYS Steady-State Thermal and Static Structural module enabling thermal stress and blade deformation analysis. The Von Mises stress distribution is calculated by means of finite element analysis (FEA). Centrifugal forces acting on the turbine wheel are considered along with thermal stresses. Once validated against available experimental data, numerical (CFD-FEA) results are extended to cases where no experimental data could be found in the literature allowing for better understanding of the performance of such radial inflow turbines at higher rpms where significant centrifugal forces can affect the integrity of the turbine.

Diego Silva de Carvalho&JesuínoTakachiTomita [8] studied the dependency of design of radial turbomachines on simplified equations from fluid mechanics (mass conservation, Navier-Stokes and energy conservation). The use of loss models improve the preliminary machine sizing and are very important mainly to determine the blade and flow angles at leading and trailing edges. But, to study the details of the flow in critical regions where there is complex flow phenomenon as recirculation, separation and tip leakage a sophisticated methodology should be applied. In this study, the Computational Fluid Dynamics (CFD) technique was applied to determine the operation characteristics of a turbocharger radial turbine operating at design-point. The 3D flow was evaluated from turbine volute inlet until the rotor outlet. The design-point condition results from CFD calculations are compared with experimental data and are in good agreement. The mesh generation, the numerical methods settings and the flow characteristics are presented and discussed.

Naveen.B &Kallu. Raja Sekhar [9] studied downsizing as a trend in engine development that allows better efficiency and lower emissions based on the increase of power output in reduced displacement engines. A natural gas engine for producer gas operation was adopted. The producer gas fuelled Engines are the upcoming Technology and more friendly to the environment compared to diesel and petrol Engines. There are some issues related to power de-rating from the engine related to the fuel properties. The main cause for the power de-rating is due to the relatively lower heating value of stoichiometric mixture of producer gas and air. This loss in power can be recovered to a much large proportion by turbocharging. Matching of the correct turbocharger to an engine is of great importance and is vital for successful operation of a turbocharged engine. It is important to have a turbine map for matching the turbocharger with an engine. The characteristics of the turbocharger's turbine from the original manufacturers were not available. In this work an attempt is made to establish the Performance Characteristics and hence the turbine map for a stripped out Holset turbocharger turbine to match with a producer gas fuelled engine.

D. Pablo Fajardo [10] studied the increasing use of turbochargers which is leading to an outstanding research to understand the internal in turbo machines. In this frame, computational fluid dynamics (CFD) is one of the tools that can be applied to contribute to the analysis of the fluid-dynamic processes occurring in a turbine. The objective of this work is the development of a methodology for performing simulations of radial turbo machinery optimizing the available computational resources. This methodology is used for the characterization of a vane-nozzle turbine under steady and pulsating flow conditions.

IV. OUTCOME OF LITERATURE SURVEY AND SCOPE FOR PRESENT WORK

In order to perform this project, literature review has been made from various sources like journal, books, article and others. This chapter includes all important studies which have been done previously by other research work. It is importance to do the literature review before doing the project because we can implement if there are information that related to this project. The radial turbine can employ a relatively higher pressure ratio is about 4 per stage with lower flow rates. The radial machines fall in the lower specific speed and power ranges. Variable angle nozzle can give higher efficiency in a radial turbine.

V. EXPECTED OUTCOMES

The expected outcome will be that we can improve the efficiency by means of changing guide vanes angle and to compare efficiency in between theoretical and actual. In this paper we will be focus on getting maximum hydraulic efficiency by changing guide vanes angle of radial turbine and comparison is done between theoretical and experimental efficiency.

VI. 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 Hellstr¨om, "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 ConferenceMelbourne 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 July2012.