International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878, Volume-1, Issue-3, August 2012

CFD Analysis for Estimation of Efficiency of Low-Pressure Steam Turbine

A. Chenchu Deepa, B.Jayachandraiah

Abstract: The performance of steam turbine blade is related to many factors. One of the important factors is the degradation and change in turbine blade profile after many hours of operation. This leads to increased in flow losses and hence reduction in overall turbine efficiency. The performance of turbine blade can be predicted and improved by using Computational Fluid Dynamics (CFD). CFD is the art of numerically solving the governing fluid flow equations in order to obtain the descriptions of the complete flow-field of interest. With the availability of computer power and efficient numerical algorithm, CFD is becoming a very important tool for engineers in improving the performance of component involving fluid flows. The reason is that CFD effectively replace the needs to perform expensive experimental measurement and testing of new design and prototype. To develop better performing blades, it is essential to identify the losses generating mechanism and study their influence and effects on performance. This paper outlines design considerations and the estimation of efficiency of LP Steam Turbine using CFD, thus aiding in optimizing the design and helps in integrating CFD into the design process itself. The CFD results are in concurrence with the analytical values.

Keywords: CFD, steam turbine, Performance.

I. INTRODUCTION

The steam turbine is one of the key components because it is the steam turbine that converts the thermal energy of the steam into rotational kinetic energy, which in turn, drives the generator shaft. It is therefore very important to keep the steam flow energy losses at a low level as possible. Research work aimed at reducing the aerodynamics and wetness losses and hence improving the steam turbine efficiency has attracted a great deals of attention in recent years e.g. [1], [2], [3], Although the majority of the research work is directly relevant to steam turbine manufacturers, the tools developed can also be used by steam turbine operator to predict and improve the steam turbine efficiency. It is well known that after many hours of operation, the blade profile of turbo machines will deviate from its design shape and this leads to increase in flow losses and reduction in expansion efficiency. The tools can be used in predicting the reduction in efficiency and also the possible improvement that may be achieved if the blade row is retrofitted by a new blade that employs the latest technology in blade profile design. The predicted improvement in efficiency can then be converted to possible future saving and compared with the cost of blade retrofitting. The analysis can be used in judging whether blade retrofitting is a viable option.

Revised Manuscript Received on 30 August 2012.
* Correspondence Author
A.Chenchu Deepa, B-tech, Department of Mechanical, JNTUA, M-tech, Department of CAD CAM, JNTUA, Anantapur. India.
Dr. B. Jayachandraiah, Head vice principal, Sri kalahasteeswara Institute of Technology.

The key in reducing the flow energy losses is the understanding of the steam flow behavior inside the steam turbine. Considerable experimental work has been performed in studying the flow [4]. In parallel with this, due to the limitation of experimental measurement and to aid in interpreting experimental results, Computational Fluid Dynamics (CFD) analysis is used. In CFD, the relevant fluid flow governing equations are solved numerically using digital computer and applied to flow inside the turbine blade rows.

II.LITERATURE REVIEW

Edwin Kramer, Hans Huber and Dr. Brendon Scarlin [5], LP steam turbine retrofits are generally undertaken as a result of mechanical problems, for example due to stress corrosion cracking, torsional vibration or erosion. They present plant operators with an opportunity to introduce advanced aerodynamic technology, and at the same time improve efficiency and availability. Standard components, the majority proven in long-term service, can be used to obtain a permanent solution to the problems. In several European countries, retrofits have been carried out on intact LP steam turbines owned by utilities whose primary motivation is efficiency improvement. Paul Albert [6] this paper has presented some of the latest advancements used for evaluating and assessing the performance of steam turbine, including methods for periodic data acquisition, interpretation of performance data, and inspection of the turbine steam path, monitoring the performance of steam turbine and evaluating the total plant. These programs are essential in order to achieve and maintain the highest level of thermal performance of a turbine-generator unit.

III. DATA COLLECTION

Turbine: Steam turbines belong to power generating turbo machines which uses the steam as a working fluid. In steam turbines, high pressure steam from the boiler is expanded in nozzle, in which the enthalpy of steam being converted into kinetic energy. Thus, the steam at high velocity at the exit of nozzle impinges over the moving blades which cause to change the flow direction of steam and thus cause a tangential force on the rotor blades. Due to this dynamic action between the rotor and the steam, thus the power is developed. Impulse-Reaction turbine is used for the 10MW capacity steam turbine, which is the combination of both impulse and reaction turbine.

Impulse turbine: pressure drop completely, occurs in the nozzle itself and when the fluid passes over the moving blades, pressure drop does not take place again. Hence pressure remains constant when the fluid passes over the moving blades. As the steam flows through the nozzle its pressure falls from inlet pressure to the exit pressure (atmospheric pressure, or more usually, the condenser vacuum).

Reaction turbine: addition to the pressure drop occurs in the nozzle there will also be pressure drop occur when the fluid passes over the rotor blades. This type of turbines makes use of the reaction force produced as the steam accelerates through the nozzle formed by the rotor. A pressure drop occurs across both the stator and the rotor, with steam accelerating through the stator and decelerating through the rotor, with no net change in steam velocity across the stage velocity across the stage but with decrease in both pressure and temperature, reflecting the work performed in the driving of the rotor.

IV. ANALYTICAL METHOD

Operating Parameters

The following parameters are considered for the steam turbine:

Type of turbine	:	Impulse-reaction turbine
Turbine Capacity	:	10MW
Inlet steam Pressure	:	65bar
Inlet steam Temperature	:	485^{0}c
Turbine Speed	:	6750 rpm
Exhaust steam Pressure	:	0.1765bar
Outlet steam Temperature	:	57.40^{0}c
Number of Stages	:	12
Working medium	:	Steam

Calculations for Stage Performance

The analytical calculations are carried out for last stage (12th stage) of the Low pressure steam turbine.

- In analytical method:
- Blade area
- Blade efficiency
- Stage efficiency
- Power developed
- Absolute velocity of steam at the outlet of moving blade
- Angle made by the absolute velocity of
- Moving blade

The above parameters of steam turbine are calculated using velocity triangles for the impulse-reaction turbine.

Table 1: shows the results obtained from analytical method

Analytical Method			
S.No	**Parameter**	**Units**	**Value**
1	Absolute velocity of steam from the stator blade V_1	m/s	456.05
2	Tangential velocity of the blade U	m/s	280.976
3	Absolute velocity at the outlet of moving blade V_2	m/s	241.93
4	Power developed in the stage P	kW	1470.538
5	Blade efficiency η_{blade}	%	81.74
6	Stage efficiency η_{stage}	%	79.245
7	Blade area	m^2	0.30722
8	Pressure at the exit of the stage	bar	0.176
9	Temperature of steam at the exit of the stage	^{0}c	57

CATIA

CATIA V5 is a powerful software package yet has a relatively short learning curve. One of the reasons for the short learning curve is that it is fully Windows compatible and the processes are consistent across the workbenches, toolbars and tools. If you learn the basics of a particular workbench the same process can be used for more complex problems. Several tools are used in more than one workbench. Modeling of steam turbine rotor, blade is done using CATIA V5. CATIA is mechanical design software. It is a feature-based, parametric solid modeling design tool that takes advantage of the easy-to-learn Windows graphical user interface. You can create fully associative 3D solid models, with or without constraints, while using automatic or user-defined relations to capture the design intent. Generative Shape Design is used to model the rotor blade. It is a complete surfacing tool used to create complex shape parts.

Fig 01: shows the CATIA model of rotor and blades of a LP impulse-reaction turbine

CFD Grid Generation

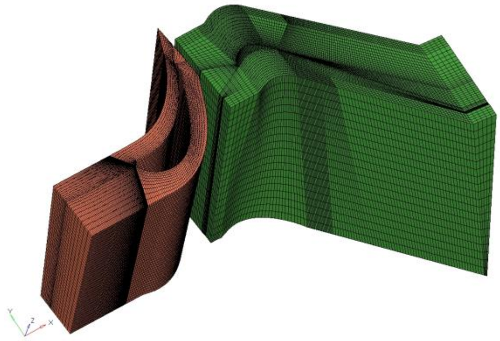

Fig 02: shows the body fitted structured mesh of hexahedral elements of stator blade and rotor blade

Hyper mesh tool is used to create the flow domain and the grid. The turbine blades mesh is shown in Fig 02. The figure show 3-D mesh of stator blade in brown collector with hexahedral element and rotor blade in green collector with hexahedral elements, first the QUAD element are created on the surface of the blade and 2-D element are dragged or given thickness to create volume mesh and the meshed module is export to the analysis software as .PRP., were imported file from the mesh module is used to carry out the analysis in CFX software. After the geometric model of the turbo machinery component has been established, the next step in the process of communicating this configuration to the CFD analysis program is to define the computational grid within the physical domain. The boundaries of this region are typically defined by the flow-path surfaces (end walls, blades, etc.) and by the periodic boundaries between blade passages, where appropriate.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

Inlet and exit boundaries are established at points upstream and downstream, where the necessary flow conditions are assumed to be known. Within this region, a three dimensional computational grid is applied, such that the governing equations will be solved at every point on the grid, or within every cell formed by the grid. The grid imposed on the physical domain must conform to the boundaries of that domain and must provide adequate resolution in all areas of the flow field to permit accurate prediction of the flow behavior. The mesh was generated as shown in Fig.02. And the grid independence check was done and the optimum number of elements found for stator or rotor is around 4,50,000. The typical mesh pattern around blade profile is shown. Sufficient number of nodes is taken around the blade profile and wall boundaries to capture wall boundary layer effect.

Analysis

CFD-CFX post processer tool is used to carry out the analysis on the meshed models. Significant percentage of the total time spent on a CFD analysis is involved in pre and post processing activities. Both the setup for an analysis and the evaluation of results require considerable effort on the part of the component designer. Therefore, the use of software tools to automate or facilitate these activities has the potential to substantially reduce the time required for the analysis and improve the overall efficiency of the process. Preprocessing involves the definition of the boundaries of mesh elements and interfaces between rotor and stator etc. The basic boundary conditions used for an element are shown in Fig 03, 04, 05. And for inflow boundary condition total temperature, total pressure and flow angle are given. For outlet flow boundary condition static pressure outlet model was selected. The basic boundaries and inputs given in CFD model are given in below.

Boundary conditions

Simulation Type	:	steady
Heat Transfer Model	:	Total Energy
Turbulence Model	:	K Epsilon
Inlet	:	total pressure total temperature turbulence level wetness fraction
Outlet	:	static pressure
Material	:	steam
Speed	:	in rpm

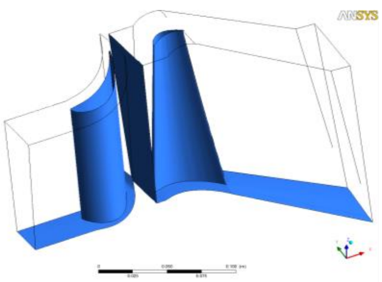

Fig 03: Boundary condition stator and rotor blade

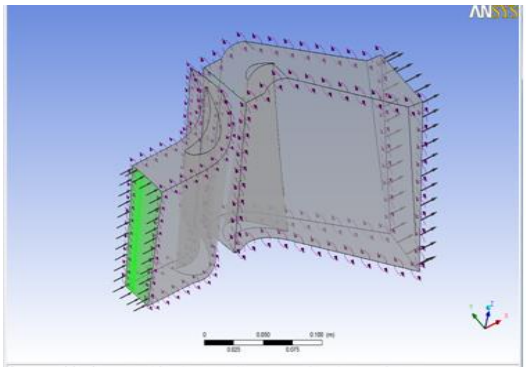

Fig 04: Inlet Pressure boundary condition-Stator blade

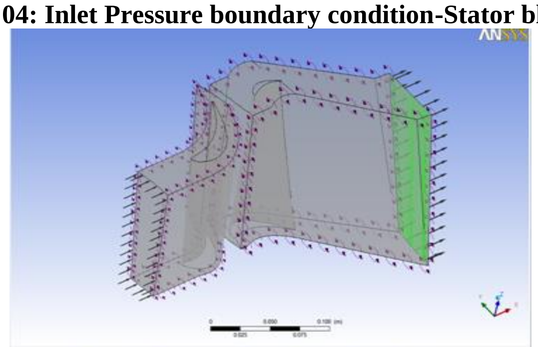

Fig 05: Outlet Mass flow rate boundary condition-Rotor blade

V. RESULTS AND DISCUSIONS

The simulation and post processing was carried out using CFD-CFX and the above problem was run for the convergence level in 450 iterations. Post processing function for a CFD analysis provides the necessary information of variables such as pressure, temperature, velocity and enthalpy etc.

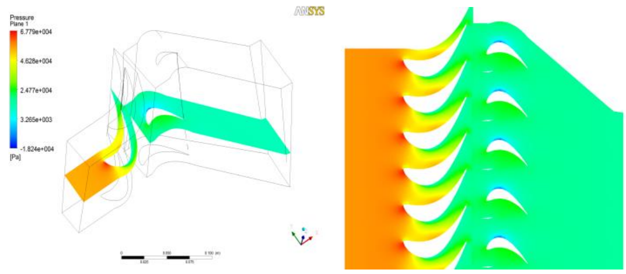

Fig 06: Contours of pressure distribution across the last stage of LP steam turbine

From the above contours shown in fig 06, we can conclude that pressure is maximum at the being i.e. inlet to stator blade and the pressure drops as the steam passes through the stator blades and the rotor blade, the pressure drops, as the working medium passes through the stator and rotor blades, this condition satisfies Impulse-reaction turbine. The pressure of steam at the inlet of the stage is 0.65 bar, pressure of steam at the outlet is 0.172 bar and state of the working medium is wet steam. Fig 5.1 and 5.2 shows the pressure distribution from 0.65 bar to 0.172 along stage of the steam turbine.

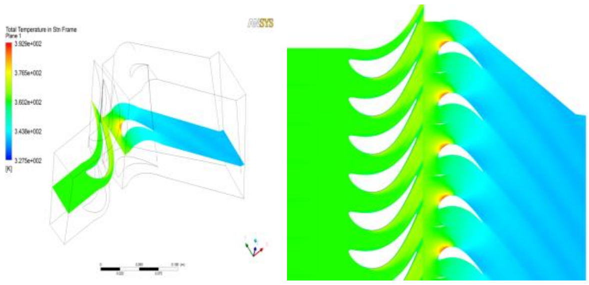

Fig 07 Contours of temperature distribution across the last stage of LP steam turbine

From the above contours shown in figure 07, we can conclude that temperature is maximum at the being i.e.at inlet to stator blade and is around 88^{0}c and the temperature at the exit of the stage is 59.5^{0}c.

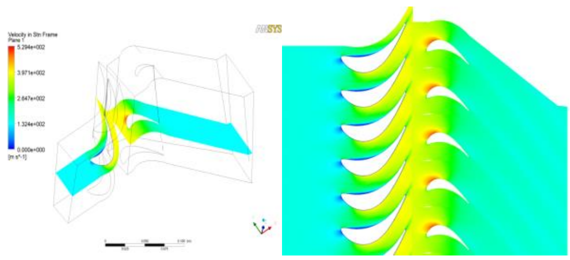

Fig 08: Contours of velocity distribution across the last stage of LP steam turbine

From the above contours shown in figure 08, we can conclude that velocity is maximum at inlet to rotor blade and the velocity increases as the steam passes through the stator blades and the velocity is maximum at the exit of the stator blade and the velocity of the steam decreases as the steam impinges over the moving blade (rotor blade), and thus satisfies condition of Impulse-reaction turbine. The maximum the velocity at the inlet of the moving blade is 456 m/s and the velocity of steam at the exit of stage is 242 m/s.

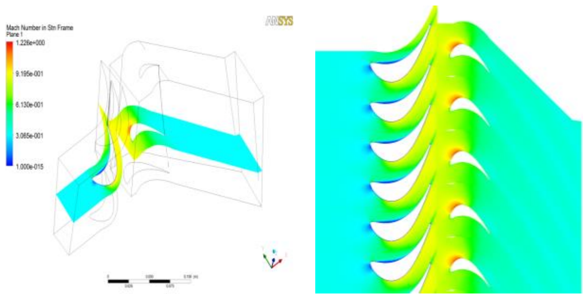

Fig 09: Contours of Mach number distribution across the last stage of LP steam turbine

Table 2 Comparison of CFD results with analytical values

Parameter	Analytical	CFD
Power developed in the stage in kW	1470	1386
Blade efficiency in %	81.74	79.8
Stage efficiency in %	79.245	78.2
Pressure at stage outlet in bar	0.1765	0.172
Absolute velocity at the outlet of the moving blade in m/s	241.93	224(average)
Temperature at the outlet of the stage in ^{0}C	57	59.5

VI. CONCLUSION

Computational Fluid Dynamics is becoming a very important tool for engineers. Although currently CFD is mainly being used in the design office by the steam turbines manufacturer, it can also be employed by power plant engineers to predict and improve the efficiency of the turbo machines. The data obtained from the prediction may then be used to aid in decision making whether to replace the blade row with the more efficient one.

The overall efficiency of turbine was predicted using CFD approach and compared with the model testing results obtained from the manufacturer and very good agreement was found. It can be concluded that CFD approach complements the other approaches, as CFD approach helps in reduction in cost of model testing and saving in time which leads to cost-effective design of the system. CFD approach may be helpful in improvement of the existing efficiency measuring techniques and evaluation of the performance of hydro turbines to enhance the viability of hydropower development.

REFERENCES

1. Zamri, M.Y., " An Improved Treatment of Two-Dimensional Two-Phase Flows of Steam by a Runge-Kutta Method", Ph.D. Thesis, Department of Manufacturing and Mechanical Engineering, The University of Birmingham, U.K., 1997
2. Bakhtar, F., So, K.S., " A Study of Nucleating Flow of Steam in a Cascade of Supersonic Blading by the Time-Marching Method", International Journal oh Heat and Fluid Flow, Vol. 12, pp: 52-64., 1991
3. Deckers, M., Simon, V., Scheuerer, G., "The Application of CFD to Advanced Steam Turbine Design", International Journal of Computer Applications Technology, 1997.
4. Bakhtar, F., Shojaee-Fard, M.H., Siraj, M. A., "An Experimental Facility for Studies of Nucleating and Wet Steam Flows in Turbine Blading", IMechE Paper No. C423/003, 1991.
5. Edwin Krämer, Hans Huber and Dr.Brendon Scarlin, ABB Power Generation, Low-pressure steam turbine retrofits.
6. Paul Albert, Steam Turbine Thermal Evaluation and Assessment.