IJESR/ICETEISM-2019/Special Issue/ Article No-32/151-155 ISSN 2277-2685
International Journal of Engineering & Science Research

Performance Analysis of Constant and Taper blade for Steam Turbine by using CFD

Murulidhar K S
Assistant Professor, Department of Mechanical Engineering
K S Institute of Technology
Bengaluru, India
murulidharks@ksit.edu.in

Dr. Puttaboregowda B
Professor, Department of Mechanical Engineering
Ramaiah Institute of Technology
Bengaluru, India
pbgowda@msrit.edu

Manjunatha B R
Assistant Professor, Department of Mechanical Engineering
K S Institute of Technology
Bengaluru, India
manjunathbr@ksit.edu.in

Ranganath N
Assistant Professor, Department of Mechanical Engineering
K S Institute of Technology
Bengaluru, India
ranganathn@ksit.edu.in

Gautham S
Assistant Professor, Department of Mechanical Engineering
K S Institute of Technology
Bengaluru, India
gauthams@ksit.edu.in

Dr. Punith Gowda K
Associate Professor, Department of Mechanical Engineering
East West Institute of Technology
Bengaluru, India
pgpunith@gmail.com

***Abstract*—The losses in the last stages of low pressure (LP) steam turbines are difficult to estimate. Two types of losses have to be considered in the last stages: aerodynamic losses due to the interaction between the fluid and the wall boundaries and thermodynamic losses occurring during the phase change. The aim of this work is to estimate the losses in blade profile through an LP steam turbine rotor. 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 increase 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).**

Keywords—CFD, Constant blade, Steam turbine, Taper blade.

I. INTRODUCTION

In the power generation industry gas turbines and steam turbines are widely used for generating power. These industrial machines are capable of producing power in hundreds of megawatts. The efficiency of a turbine is largely dependent on its aerodynamic performance. Hence, the design of blade profiles for nozzles and rotors are continuously improved over the decades to achieve better overall efficiency for the turbine.

Currently over 35% of electrical energy demands are generated by steam turbine. Considering, the vast amount of fuel associated with this, it is importance that the steam turbines are running at their highest possible efficiencies. Any improvement in efficiency will result in a considerable amount of saving and it is anticipated that a single percentage change in steam plant efficiencies will result in an annual saving. The overall efficiency of a power plant is directly dependent on the turbine entry temperature and the back pressure, therefore it is important to realize that for a given main steam condition and back pressure, the overall plant efficiency can be improved by increasing the efficiency of the individual plant components. 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 great deals of attention in recent years. 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.

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. 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.

A. Problem statement

Steam turbine cascade analysis is usually conducted with the compliment of steam; instead air is being use as the analysis medium. Current setup for cascade analysis involves 2types of blade vanes used in the high-pressure stage of a Rateau steam turbine. These blades have a 2-D cross-section without twist and are used in multiple stages keeping similar blade profiles. The purpose of the nozzles is to accelerate and guide the flow into the next stage of rotors. Since a steam turbine can spend a considerable amount of time operating at off-design conditions, the mass flow in the turbine and the rotors' speed varies. Hence, the flow entering each stage of nozzles and rotors is inclined at an incident angle. Current analysis blades are made with taper & constant cross section. In both cases Taper & constant blade profiles flow path will be analyzed in CFD software & also report shall be compared with analytically for constant section profile.

B. *Objectives of the study*

Steam turbine in most of the cases it operates for longer period (approximate estimated life is 25 years) to suit the customer requirement. In such a condition it is very difficult to design to suit many different types of operating condition & give the performance evaluation accordingly. So blade selection should always be important parameter .To design a blade to predict Design performance, blade profile shall be selected to satisfy design point requirement considering all design constraint. To operate a steam turbine for longer period it is necessary that stress should be minimized as much as possible. So In this project to reduce the stress blade is made taper section without any twist factor & compared with constant section blade profiles. In the taper section blade weight shall be considerable less weight than constant section of the blades resulting taper centrifugal force & stresses will be reduced.

Analytical Method		
Sl. No	**Parameter**	**Units**
1	Absolute velocity of steam from the stator blade V1	480.77m/s
2	Tangential velocity of the blade U2	259.6 m/s
3	Absolute velocity at the outlet of moving blade C2	212 m/s
4	Power developed in the stage P	755.5 kW
5	Blade exit relative velocity W2	374.7 m/s
6	Stage efficiency η stage	71.18 %
7	Blade height	153 mm
8	Pressure at the exit of the stage	0.10 bar

II. Conclusion Of Literature Review

From the literature reviews we may say that many people have carried out analysis on steam turbine casings, blades, losses in steam flow, performance of steam turbine using CFD and ANSYS software for different capacities of steam turbine.

In this project we are considering 15MW capacity steam turbine and comparing performance of the low pressure steam turbine blade profiles analytical with the CFD analysis software.

III. Turbine Cascade Design And Analysis

- The conceptual design of the steam turbine blades is analyzed through analytical methods.
- Design of the steam turbine blades is created in Solid works tool.
- The 3D model blades and fluid region is created on Solid works for the CFD solver requirement.

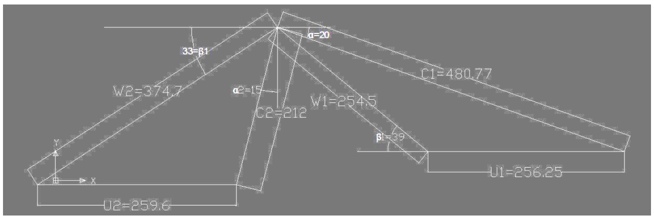

Fig. 1. velocity triangle – Last stage of turbine

A. *Analytical Methods*

- Operating parameters

The following parameters are considered for the steam turbine:

Type of turbine : Impulse-reaction turbine

Turbine Capacity : 15MW

Inlet steam Pressure : 65bar

Inlet steam Temperature : 485^0C

Turbine Speed : 6900 rpm

Exhaust steam Pressure : 0.1bar (a)

Outlet steam Temperature : 45.40^0C

Number of Stages : 12

- Last Stage Operating Parameters

Stator blade

Inlet steam pressure Pi1 : 0.40 bar

Inlet Dryness fraction x : 0.88

Outlet steam pressurePo1 : 0.15 bar

Inlet temperature. : 76^0C

Angle made by the stator α1 : 19 degree

Inlet Flow : 5.82 kg/sec

Mean diameter of stator D_m : 709 mm

Moving blade

Inlet steam pressure Pi2 : 0.15 bar

Inlet Dryness fraction x : 0.850

Outlet steam pressure Po2 : 0.1 bar

Inlet temperature. : 54^0C

Flow rate of fluid : 5.82 Kg/s

Blade angle at exit of β2 : 33 degree

Mean diameter of rotor D_m : 718 mm

TABLE 1-Summary of analytical calculations

B. *Computational Fluid Dynamics Analysis Of Turbine Blades*

Complexity of the turbo machinery flow field limits CFD simulations to Reynolds-averaged Navier-Stokes (RANS) approximations. The flow field of a transonic fan over its entire operating range is particularly troublesome; it contains all the flow aspects most difficult to represent – boundary layer transition and separation, shock-boundary layer interactions, and large flow unsteadiness. Multistage configurations further the complexity as “neither in the stator nor rotor frame of reference is the deterministic flow steady in time” (Adamczyk, 1999).

Direct Numerical Simulations (DNS) and Large-Eddy Simulations (LES) are not currently practical for the fan/compressor flow field. The DNS explicitly solves for the instantaneous flow field and requires extremely fine gridding to resolve the smallest length scales – on the order of Reλ9/4 (Metais, 1996). For a blade chord Reynolds number of 500,000 (a minimum design goal; see Hill and Peterson, 1992, p.315), a DNS solution would require on the order of

1012 grid points per blade passage. Even the relatively coarse grid of the LES (which explicitly solves down to “large” length-scale eddies and “models” the energy exchanges with small scales) poses a large computational requirement when applied to turbo machinery problems.

C. Geometrical Model

Figure 2 shows the blade profile CAD model constant section with flow region.

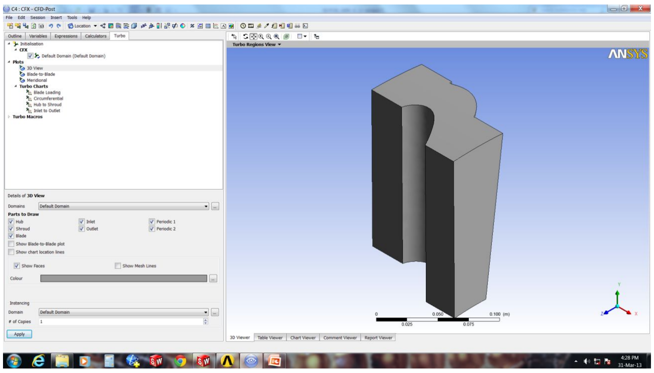

Fig. 2. Blade profile cad model constant section with flow region.

Figure 3 shows the blade profile CAD model tapered section meshed with flow region.

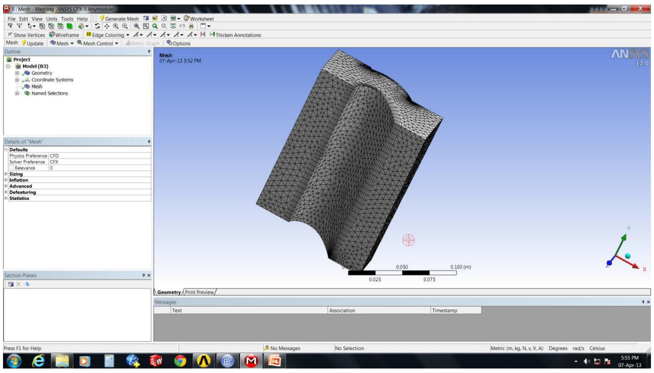

Fig. 3. Blade profile tapered section with flow region

Figure 4 shows the run CFX solver till the convergence

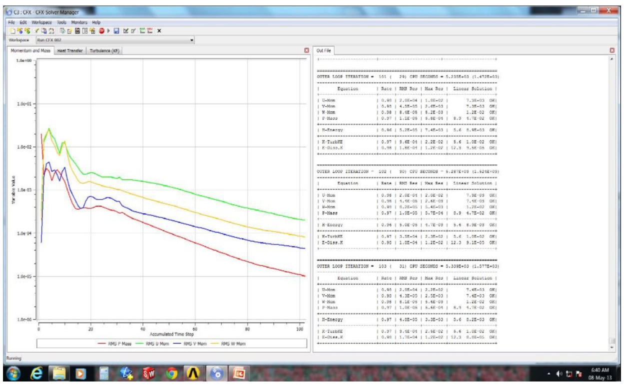

Fig. 4. shows the run CFX solver

IV. Results And Discussions

The simulation and post processing was carried out using CFD-CFX and the above problem was run for the convergence level in around 2500 iterations.

Figure 5 shows velocity distribution over the constant profiles in the entire circular region.

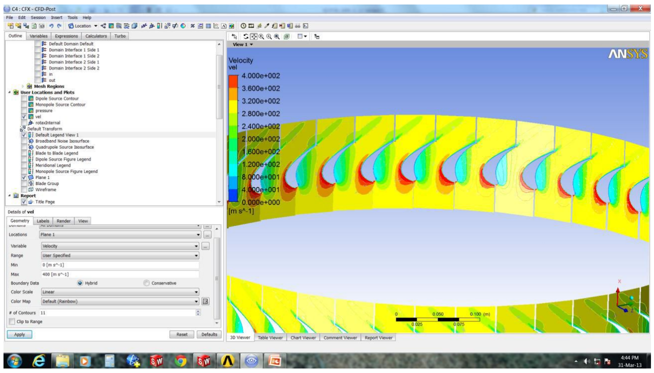

Fig. 5. velocity distribution over the constant profiles in the entire circular region

Pressure, Velocity And Mach Number Distribution For Taper Section Profiles

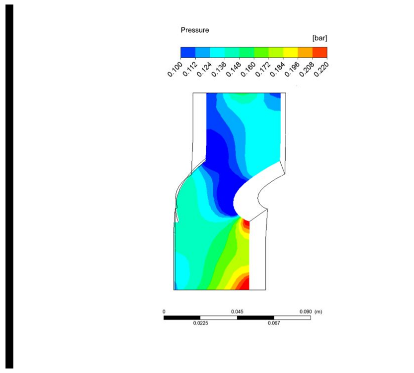

Fig. 6. Pressure distribution over the taper profiles @ mid-section

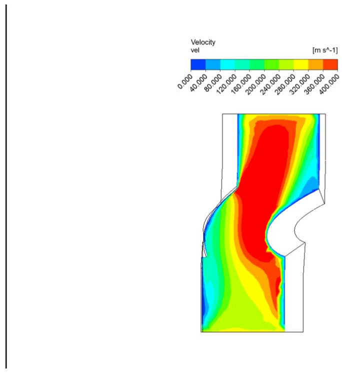

Fig. 7. Velocity distribution over the constant profiles@ mid –section

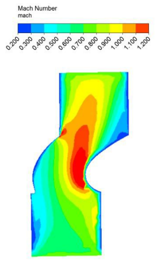

Fig. 8. Mach number distribution over the constant profiles @ mid-section

TABLE II. SUMMARY OF TAPER AND CONSTANT SECTION BLADE

Specification	Constant section	Taper section.
Inlet Pressure(bar)	0.15	0.15
Inlet temp. (oc)	54	54
Exhaust pr.(bar)	0.1	0.1
Exhaust temp.(oc)	45.4	45.4
Exit Mach no.(hub/tip)	0.95/1	0.92/1.07
Exhaust enthalpy(kj/kg)	2221.3	2226
No. of blades	47	47
Exit Blade velocity(m/s)	383.44	382.3
Stage power(kW)	778.4	775.7
Stage efficiency (%)	72.7	72.4
Mass flow(kg/sec)	5.82	5.82

TABLE III. SUMMARY OF RESULT COMPARISON OF TAPER & CONSTANT SECTION BLADE PROFILES WITH ANALYTICAL & CFX APPROACH

Specification	Analytical	Constant	Taper
Inlet pr.(Bar)	0.15	0.15	0.15
Inlet mass flow(kg/sec)	5.82	5.82	5.82
Inlet blade velocity(m/sec)	254.5	254.5	254.5
Exit Blade velocity(m/sec)	374.7	383.44	382.3
Stage power(kW)	755.5	778.4	775.7
Stage efficiency (%)	71.18	72.7	72.4
Turbine speed (rpm)	6900	6900	6900
Blade weight(gms)	582	598	265.3
Blade centrifugalForce(N)	112	112	49

V. CONCLUSIONS

The influence of taper & constant section in the profile on velocity distribution is investigated for two steam turbine blades. The inlet flow angles are varied from -36^o to 26^o for the two cascades and the exit Mach number ranged from 0.9 to 1.1.

Trends for velocity loss variation with exit Mach numbers are similar at all tested angles for both blades. Losses are rather constant at subsonic Mach numbers due to accelerating flow and a thin boundary layer from a favorable pressure gradient.

In transonic flow the profile losses increased steeply due to the formation of trailing edge shocks and shock induced boundary layer separation.

When the exit flow goes supersonic, the losses peaked and decreased subsequently due to the transformation of the stronger normal shock into weaker oblique shocks.

The inlet flow angles are varied from -36^0 to 26^0 for the two cascades and the exit Mach number ranged from 0.9 to 1.1.

When the exit flow goes supersonic, the losses peaked and decreased subsequently due to the transformation of the stronger normal shock into weaker oblique shocks. However, there are instances at positive incidence where the losses continued to rise.

The validation from CFD/CFX analysis is reaffirmed by analytical data obtained from calculation. The turbine blades similar geometry and their profile losses are quite similar with losses from nozzle at subsonic exit Mach numbers.

Even though at supersonic exit Mach numbers, the profile losses for taper are much higher due to its larger trailing edge curvature which causes over acceleration.

Shadowgraph studies presented evidence supporting that stronger shocks are formed at the trailing edge for taper blade, hence the higher losses in supersonic exit flow.

The prediction method has adequately predicted the profile losses for the constant section blade in subsonic conditions for near design incidence. However, the taper & constant blade profiles are almost similar in efficiency even not more than 0.004% difference in efficiency has been observed.

Weight is nearly 50 % difference than the constant section profiles, therefore centrifugal force are also 50 % less observed without compromising much on the efficiency of the stages.

Less centrifugal forces taper blade have less stress, resulting high blade life than the constant section blade profiles.

ACKNOWLEDGEMENT

The success of this project depends largely on the encouragement of many individuals who have guided us through this. We take this opportunity to express our gratitude to those who have been instrumental in the successful completion of this project

We wish to express our deep sense of gratitude to our institution, K.S. Institute of Technology for its ideals and inspirations and EWIT to given an opportunity to publish the paper.

We express our sincere thanks to Dr. T V Govindaraju, Principal, KSIT, We take immense pleasure in thanking to our Head of the department, All teaching and non-teaching staff of Mechanical Engineering department for their support and encouragement.

Last but not least, we wish to thank all our friends and our parents who extend their help and cooperation to us.

REFERENCES

[1] M.Y. Zamri, “An Improved Treatment of Two-Dimensional Two-Phase Flows of Steam by Runge-Kutta Method,” Ph.D. Thesis, Department of Manufacturing and Mechanical Engineering, the University of Birmingham, U.K., 1997.

[2] F. Bakhtar, K.S. So, “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] M. Deckers, V. Simon, D., Scheuerer, “The Application of CFD to Advanced Steam Turbine Design,” International Journal of Computer ApplicationsTechnology, 1997.

[4] J.I. Cofer, “Advances in Steam Path Technology,” Power-Gen Europe, Amsterdam RA1, NL. 1995.

[5] R.B. Scarlin, “Advanced Steam Turbine Technology for Improved Operating Efficiency,” Power-Gen Europe, Amsterdam RA1, NL, 1995.

[6] H. Oeynhausen, A., Drosdziok, M. Deckers, “Advanced Steam Turbines for Modern Power Plants,” IMechE Paper No. C522/032, 1997.

[7] F. Bakhtar, M.H. Shojaee-Fard, M.A. Siraj, “An Experimental Facility for Studies of Nucleating and Wet Steam Flows in Turbine Blading,” IMechE Paper No. C423/003, 1991.