International Journal of Research in Engineering Technology – Volume 3 Issue 1, Nov – Dec 2017

RESEARCH ARTICLE OPEN ACCESS

3 D Computational Fluid Dyanmics Method used for Performance Prediction of Axial Steam Turbine

Dr. Praveen. T

Professor,Department of Mechanical Engineering, Avanthi Institute of Engineering and Technology,Gunthapally, Abdullapoormet.Hyderabad.

Abstract:

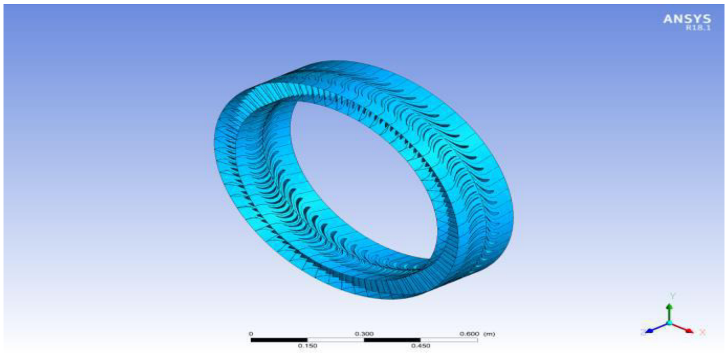
Steam Turbines are the main prime movers for electrical power generation. Besides power generation these turbines find extensive applications for marine propulsion with horsepower ratings in excess of 10000 HP. A single axial turbine stage with 58 and 60 blades in the rotor and stator respectively is modelled and analysed for performance using Ansys-CFX CFD software. Superheated steam at temperatures 500 and 600 K at 2 @ 3 bar respectively was chosen as the working for the turbine stage. The thermodynamic properties of the working fluid are taken from IAPWS97 database. It is always mandatory to carry out extensive simulation studies for turbo machinery problems prior to arriving at the optimum conditions for efficient operation during design stage. The turbine is run at different speeds ranging from 6000 to 12000 rpm at both inlet conditions. The power output was found to increase more than double at higher pressure of 3 bar and the efficiencies are more or less same ,in excess of 90 % for both inlet conditions . However the speeds at which maximum efficiencies occurred differed by 15 to 20 % i.e. 8000 rpm at 2 bar and 10000 rpm at 3 bar .Large kinetic energy increase in the stator blades is observed indicating high nozzle efficiency. The temperatures and relative mach number are found to decrease with increase in speed due to dependence of relative mach number on rotor speed and simultaneous reduction in sonic velocity with fall in temperature.

Key words: **Prime movers, Ansys, CFX, CFD, IAPWS97, Turbo Machinery, Nozzle Efficiency, Kinetic Energy.**

1. INTRODUCTION

A steam turbine is a mechanical device that extracts thermal energy from pressurized steam and converts it into rotary motion. Because, the turbine generates rotary motion, it is particularly suited to be used to drive an electrical generator – about 80% of all electricity generation in the world is by use of steam turbines. The efficiency of a turbine is largely dependent on its aerodynamic performance. Hence, the design of blade profiles for stators and rotors are continuously improved over the decades to achieve better overall efficiency for the turbine. The available energy in the hot and high pressure steam is first converted into kinetic energy by the expansion of steam in a suitably shaped passage known as nozzle from which it issues as a high velocity jet having a high tangential component. Then a part of this kinetic energy and sometimes part of the pressure energy are converted into mechanical energy by directing the jet at a proper angle, against curved blades mounted on a rotating disc. An ideal steam turbine is considered to be an isentropic process or constant entropy process, in which the entropy of the steam entering the turbine is equal to the entropy of the steam leaving the turbine. The efficiency of the steam turbine is often described by the isentropic efficiency for expansion process. No steam turbine is truly "isentropic", however, typical isentropic efficiencies ranging from 70%-90% can be realized based on the applications of the turbine. The presence of water droplets in the steam will reduce the efficiency of the turbine and cause physical erosion of the blades. Therefore the dryness fraction of the steam at the outlet of the turbine should not be less than 0.9.

Computational Fluid Dynamics (CFD), is a branch of fluid dynamics which uses numerical methods and algorithms to solve fluid flow problems. Reduction of time and cost to predict the model behavior in real environment is key advantage of CFD analysis. However, the CFD analysis results should be analyzed and validated before the model is accepted. ANSYS CFX and ANSYS Fluent are the commercial CFD codes available. The main difference between these is the way solvers integrate the flow equations and solution strategies. CFX uses finite volume elements to discrete the domain. Contrarily, Fluent utilizes finite volumes. They are both control volume based solvers, which ensures conservation of flow quantities. The CFD analysis of Steam turbine stage in the paper is carried out using ANSYS CFX. CFX is a highly user friendly software specially tailored for turbo machinery problems. By default the stator and rotor interfaces are created in a turbine stage and a comprehensive detailed turbine report is generated in post processing addressing the nozzle and turbine performance in respect of power, efficiency, mass flow rate &mach number etc.



1.1. Modelling Turbine Stage in CFX

In general all turbo machinery problems are solved using rotational periodicity and hence a single or double blade passage in the stator and rotor are only modeled to reduce the overall computing time. In this paper single axial turbine stage is modeled . The stator has 60 blades and rotor has 58 blades .The stator or rotor fluid domain consists of various boundaries for inlet, outlet, hub, shroud and other periodic surfaces. Initially CAD drawings for both the fluid domains are imported into Ansys Turbo grid software and meshing compatible with CFX solver is generated.

The working fluid used in the paper is superheated steam at 2 bar and 3 bar pressure at temperatures 500 and 600K respectively. CFX derives all the thermodynamic properties between inlet and exit pressures through the IAPWS97 database provided as default in the software. Both the fluid domains are imported into the Ansys CFX- pre and the rotational speed is given as input for the rotor. With regard to inlet and outlet boundary conditions CFX has different options, static pressure or velocities and temperature of the fluid at inlet and mass flow rate or average static pressure at outlet. In the present case of the axial turbine the pressure at inlet as well as at outlet are given as boundary conditions. Inlet pressure is 2 bar @500K/ 3 bar @600K and outlet pressure is 1 bar. Thermodynamic properties at inlet as are given below

Inlet condition	2 bar @500K	3 bar @ 600K
Enthalpy, KJ/Kg	2930	3130
Saturation temperature	393K	406K
Degree of superheat	107K	194K
Specific volume, Meter cube/Kg	1.15	0.887

1.2. Turbulence Modelling

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. The strong channeled curvature's and intensive rotations prevalent in turbo machinery resulting in high swirling and secondary flow nictitates choosing appropriate turbulence model for accurate performance prediction. CFX offers many turbulence models and in the present case of axial turbine stage, SST model was chosen as this turbulence model as this model is strongly recommended for turbo machinery problems.

2. RELATED WORK

The efficiency and power output strongly depend on the inlet conditions of the steam turbine and it is essential to predict the performance in the initial stages of design so as to run the turbine at optimum conditions of maximum efficiency and power output. The turbine at both inlet conditions i.e. 3 bar 600K and 2 bar 500 K is simulated at different speeds i.e. from 6000 to 12000 rpm to obtain optimum conditions.

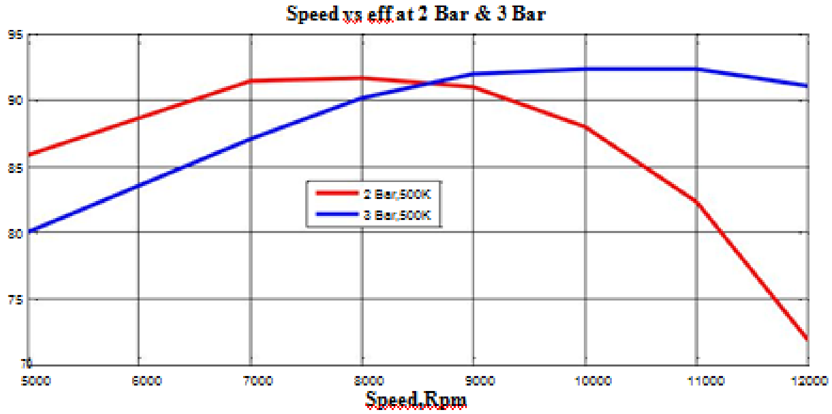


Figure 1.1 Speed vs. Efficiency at 2 Bar & 3 Bar

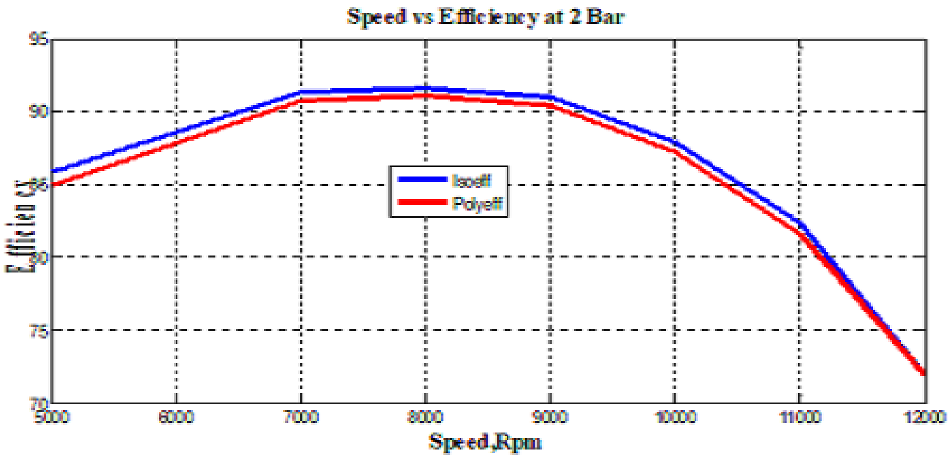


Figure 1.2 Speed vs. Efficiency at 2 Bar

It is seen from above figures 1.1 and 1.2 that both isentropic and polytrophic efficiencies show increasing trend initially and there after drops off after reaching maximum value. However at both inlet conditions the maximum efficiencies are above 90% but they do not occur at the same speed i.e. 91% at 8000 rpm for 2 bar while 93% at 10000 rpm at 3 bar. Both polytrophic and isentropic efficiencies show similar trend and are almost equal at all speeds as seen in fig 1.2.

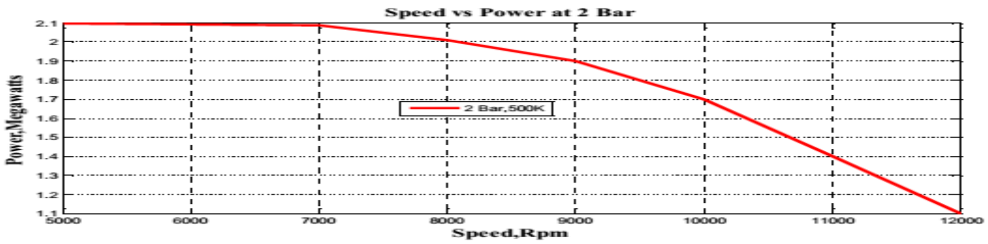


Figure 1.3 Speed vs. Power at 2 Bar

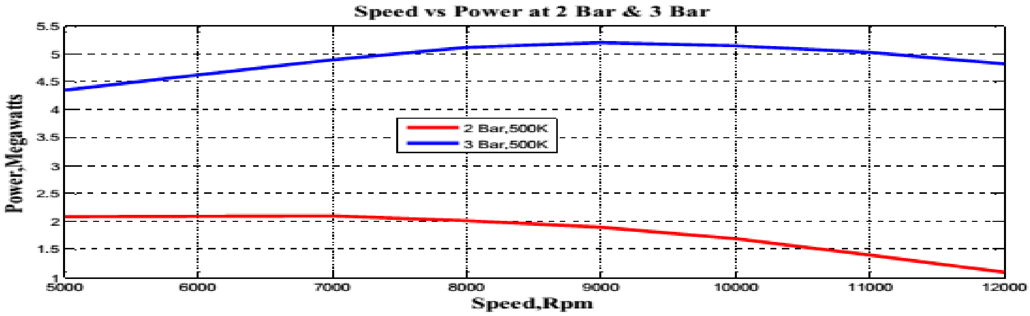


Figure 1.4 Speed vs. Power at 2 Bar & 3 Bar

Power output of the turbine stage depends on the enthalpy drop in the rotor which in turn depends on the rotor speed. As observed in figs 1.3 to 1.4 power output variation with speed shows more or less similar trend with efficiency variation at both inlet conditions. Maximum power produced is more than double at 3 bar i.e. 5 Megawatts at 3 bar and 2 Megawatts at 2bar.

2.1. Velocity distribution in the axial turbine stage

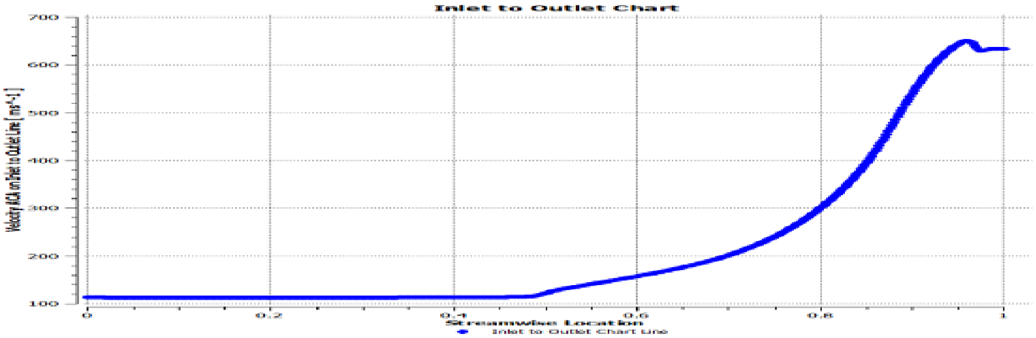


Figure 1.5 Velocity distribution across the Stator

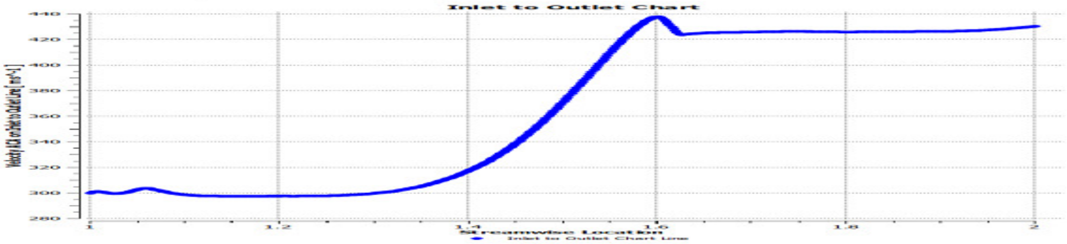
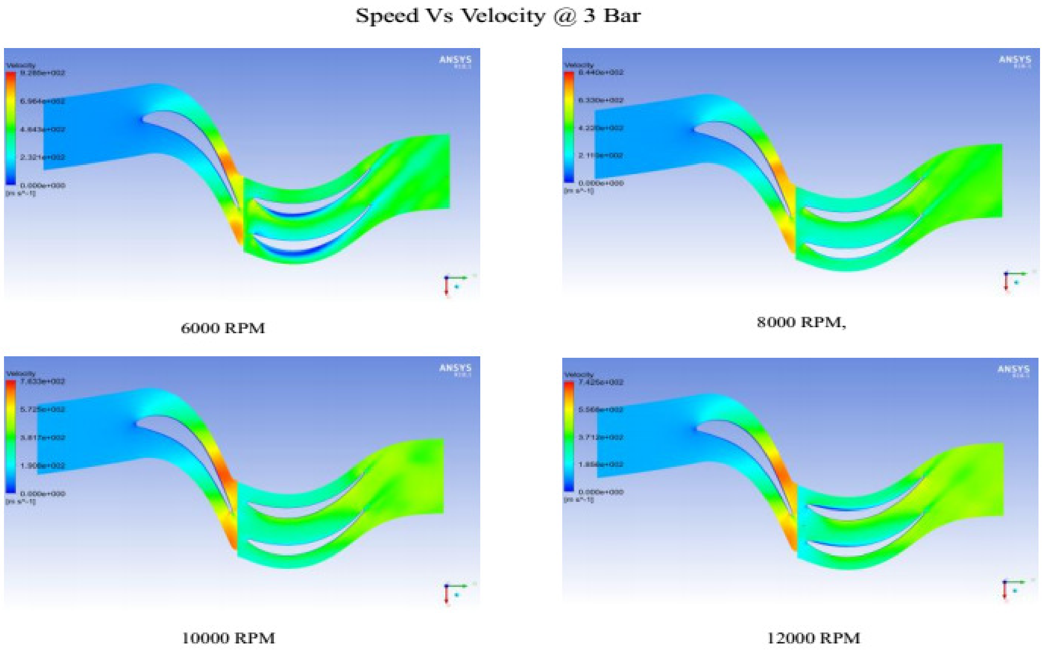


Figure 1.6 Velocity distribution across the rotor



6000 RPM

8000 RPM,

10000 RPM

12000 RPM

Figure 1.7 Velocity distribution across the Stator &Rotor at different speeds

Figs 1.5 to 1.7 indicate the velocity variation across the stator and rotor. The Kinetic energy increase through the stator is clearly seen in the stator guide vanes since these vanes acts as steam nozzles. The velocity at exit is 650 m/sec while at inlet it is only 100 m/sec. thought the exit stator velocity is very high; the rotor inlet velocity is only 300 m/sec and gradually increases to 430 m/sec due to conversion of part of pressure energy into kinetic energy in the rotor.

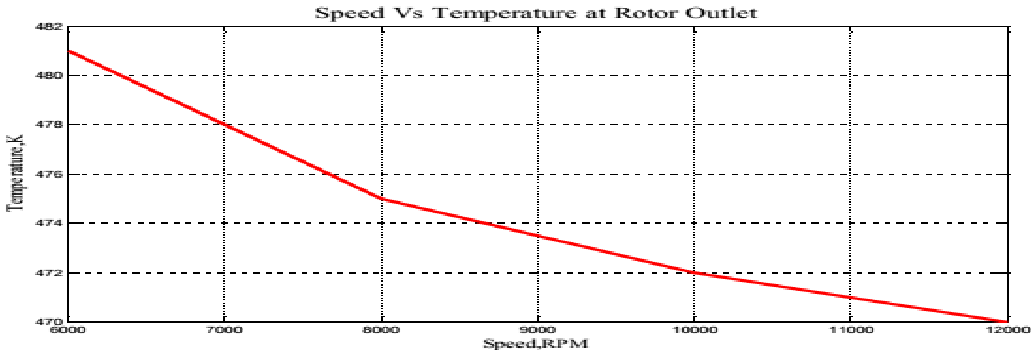


Figure 1.8 Speed vs. Temperature at Rotor Outlet at 3 bar

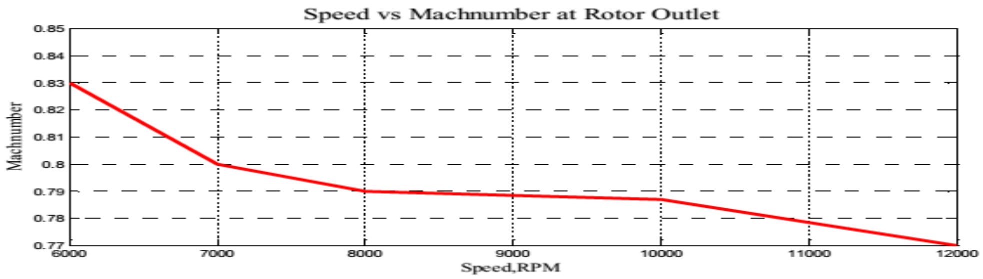
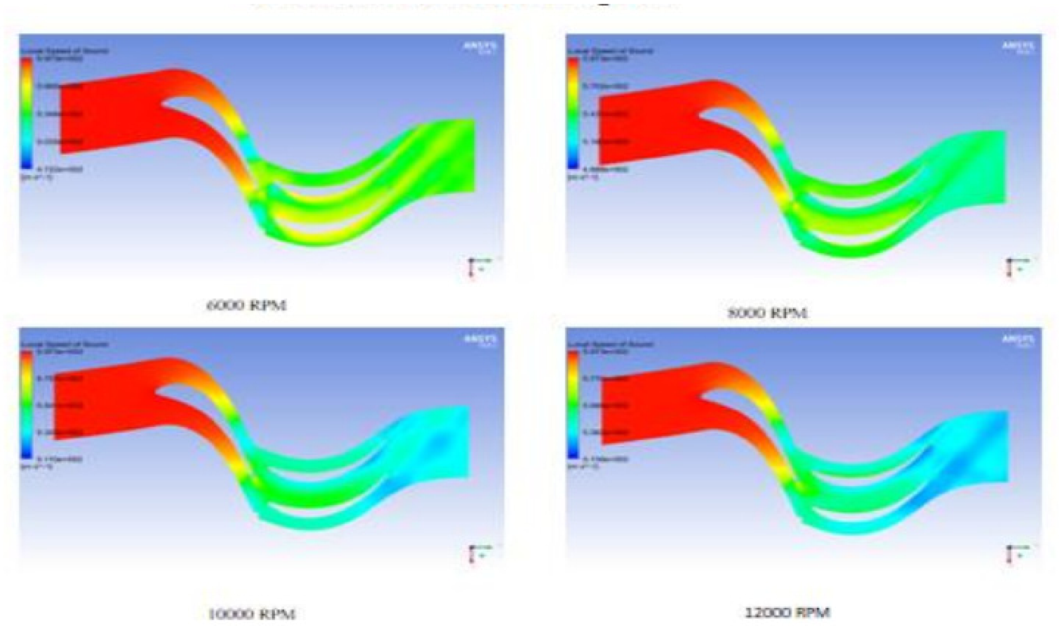


Figure 1.8 Stream wise plot of Relative Mach Numbers at 3 bar, 12000 rpm



6000 RPM

8000 RPM

10000 RPM

12000 RPM

Figure 1.9 Speed vs. Local Speed of Sound @3 Bar

Figs 1.7 to 1.9 shows variation of temperature and relative mach number and local sound speed in the rotor when speed is increased from 6000 to 12000 rpm. .Relative Mach number not only depends on the velocity and sonic velocity at the corresponding point, but also on the speed of the rotor. The sonic velocity in turn depends on the temperature and ratio of specific heat at the corresponding point. Fig 1.9 clearly shows that the local speed of sound in the stator fluid domain mostly remains the same at all speeds investigated i.e. from 6000 to 12000 rpm where as the region near rotor out let is different at each speed as evident from different color contours .The temperature plot shows decreasing trend with respect to the rotor speed which in turn cause reduction in sonic velocity and consequently the mach number. The rotor exit temperature is 470K at 3 bar which is more than the saturation temperature corresponding to 3 bar indicating the condition of steam remains superheated with degree of superheat equal to 64 K as against 194K at turbine inlet.

CONCLUSIONS

Steam turbines find extensive applications in power generation sector and marine propulsion. It is essential to carry out detailed simulation studies for performance evaluation of the turbine prior to freezing the final design. Ansys CFX and Fluent CFD software offers efficient performance evaluation and especially CFX is considered more user-friendly for turbo machinery simulation problems. In the present case a single turbine stage is considered with superheated steam at 2 bar 500 K and 3 bar 600K as working fluid at different speeds ranging from 6000 to 12000 rpm. IAPWS 97 database has been used for working fluid. Maximum power output is found to occur at different speeds for the two inlet conditions considered i.e. 5 Megawatts @ 10000 rpm at 3 bar and 2 Megawatts @ 8000 rpm. Large kinetic energy increase is observed in the stator blade passage resulting nozzle in efficiency of 94%.Temparatures and relative mach number showed decreasing trend with increasing speed Maximum efficiency in excess of 90 % was realized at both inlet conditions. The rotor exit temperature is 470K at 3 bar which is more than the saturation temperature corresponding to 3 bar indicating the condition of steam remains superheated with degree of superheat equal to 64 K as against 194K at turbine inlet.

REFERENCES

[1]. V. Boiko, D. I. Maksiuta Optimal Design of High Pressure Steam Turbine Stage Using Computational Fluid Dynamics, National Technical University «Kharkiv Polytechnical Institute» Kharkiv, Ukraine

[2]. Shivakumar Vasmate1 Computational Fluid Dynamics (CFD) Analysis of Intermediate Pressure Steam Turbine, Int. J. Mech. Eng. & Rob. Res. 2014 72nd Conference of the Italian Thermal Machines Engineering Association, ATI2017, 6-8 September 2017, Lecce, Italy

[3] Juri Bellucci Federica Sazzini Filippo Rubechini Andrea Arnone, Using CFD to Enhance The Preliminary Design of High-Pressure Steam Turbines, Proceedings of ASME Turbo Expo 2013: Turbine Technical Conference and Exposition GT2013 June 3-7, 2013, San Antonio, Texas, USA

[4] John D. Anderson. Jr, "Computational Fluid Dynamics, the basics with applications", Inc. New York St. LOlliS San Ffancisco Auckland

[5] Dr. R. K. Bansal, "Fluid Mechanics and hydraulic machines", Tata Mc Graw Hill, 2006

[6] C.W. Haldeman, R.M. Mathison; Aerodynamic and Heat Flux Measurements in a Single-Stage Fully Cooled Turbine – Part II, Journal of Turbomachinery, vol. 130/021016, April 2008. [7] X.Yan, T.Takinuka; Aerodynamic Design Model Test and CFD Analysis for a Multistage Axial Helium Compressor, Journal of Turbomachinery, ASME paper,.

[8] Arun K.Saha, Sumanta Acharya, Computations of Turbulent Flow and Heat Tansfer Through a Three-Dimensional Nonaxisymmetric Blade Passage, Journal of turbomachinery, ASME paper, Vol. 130/031008, July 2008.

[9] Horloc, J.H., The Thermodynamics Efficiency of the Field Cycle, ASME paper Vol. no. 57.A.44, 1957.