

Computational Fluid Flow Analysis of Turbine Used in OTEC System

Sonam Nagar¹, H C Thakur², Roma singh³

P.G. Student, Department of Mechanical Engineering, Gautam Buddha University, greater noida, UP, India^{1,3}

Assistant Professor, Department of Mechanical Engineering, Gautam Buddha University, Greater Noida, UP, India²

ABSTRACT: Ocean Thermal Energy Conversion (OTEC), it is an innovative non-conventional source of energy based on the technology currently being developed. As 80% of sun's radiations falling on earth are been absorbed by the ocean or the sea water, by using this advance technology we can extract solar energy entrapped in the sea/ocean water. The OTEC system comprises of expansion turbine, heat exchangers evaporator, condenser, instrumentations etc. The Expansion turbine is the indispensable component of the OTEC plant. The power output of the system depends on the efficiency of turbine designed to operate at the inlet and outlet temperature of 23°C and 14.5°C respectively. The present works deals with the enhancement of turbine efficiency. In this analysis ammonia is taken as the working fluid that is used to operate the radial flow turbine coupled to a generator. The emphasis of the given work is to increase the output power for very low working temperature cycle. Computational Fluid Dynamic (CFD) analysis have been done on turbine by considering the three dimensional fluid flow. The blade design, numbers of blades, type of fluid – are highly desirable while optimizing the efficiency of turbine.

KEYWORDS: Ammonia, ANSYS, CFD, Ocean Thermal Energy Conversion, OTEC, Turboexpander..

I. INTRODUCTION

Due to enormous consumption of conventional forms of energies, that has lead to the degradation in availability of the conventional energy source and has been speculated its existence for the coming 50 years. Researchers started to find out the other alternative of renewable energy. OTEC is the recent innovative initiation that came up very recently. Due to its complex design and high investment, makes it very hectic to practice commonly. OTEC works on the principle of D'Arsonval closed cycle.

It works on the principle of temperature gradient that exists between the depth of sea and the water surface. This energy conversion does not require any kind of fuel to burn to extract power and work output from the turbine.

Its components involve very low evaporator/condenser temperature, low pressure turbine with working fluid like ammonia, R134a. The mechanism involve in which the working fluid exchanges heat with surface water and get converted into saturated vapour, then finally expands in turbo expander, rotating the shaft and generating power through the generator.

The present work in this paper emphasis on the computational fluid flow study of fluid passing through the OTEC closed cycle and how the output power or the efficiency can be increased. As the efficiency of OTEC is very low i.e (3%-7%) it is becoming a challenge for the researcher and scholars to find eminent ways to increase its power output. The working efficiency has been optimized in turbo expander. The enhancement depends on various parameters includes – number of turbine blades, turbine blades design, type of fluid used, power ratio, mass flow rate of the working fluid.

A 3D analysis has been carried out on turbine, this simulation hence evaluated the performance of turbine including the temperature distribution on turbine blades, velocity, enthalpy change, pressure variation etc. This provide an overview

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

of the energy losses and how by changing the parameter such as by changing the size of the rotor i.e radius, or by changing the working fluid or the number of blades a more efficient turbine output can be obtained
Paper is organized as follows. Sections II describe the literature review on design and performance analysis of the turbo expander. This section also portrays the recent development/contributions that have been achieved in this area. Various physical problems can be solved by CFD approach by discretization and by applying the appropriate boundary conditions that is given in Section III. Section IV presents CFX generated contours and graph showing results of CFD fluid flow analysis representing variations in properties across Streamwise location, inlet and outlet. Finally, Section V presents conclusion and result

II. RELATED WORK

One of the main components of most OTEC plants is the expansion turbine or the turboexpander. Since the turboexpander plays the role of an important part in the OTEC system as well as cryogenic systems where low temperature working fluid is desirable its reliability and working efficiency must be high. And it also depends on the cost effectiveness parameters of the entire ocean thermal energy conversion plant.

Due to their extensive practical applications, the turboexpander is grabbing the attention of a huge number of researchers or scholars all over the years. These Investigations have involved an experimental as well as the theoretical studies that have occupied a huge research in ongoing literature. The process of designing turbomachine is very seldom straightforward. As in any turbomachine the final design is the result of several engineering research disciplines such as the fluid dynamics, stress analysis of mechanical elements, mechanical vibration, tribology, controls, mechanical design of body parts and machines and fabrication to uphold. The process design parameters which specify the selection are the flow rate of the fluids, gas compositions of elements, the inlet pressure, inlet temperature and outlet pressure. The section working on design and development of the turboexpander is intended to explore the basic components of a turboexpander .

III. PROBLEM SETUP

A. Designing in Bladegen

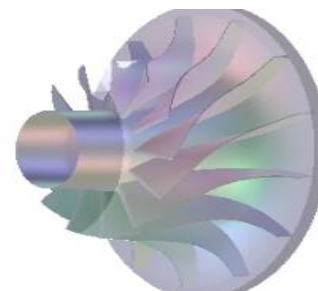
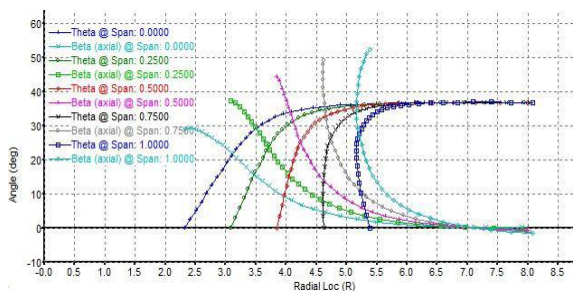


FIG:1. VARIATION OF BETA AND THETA ANGLE AT DIFFERENT SPANS

FIG:2 SOLID MODEL OF TURBINE GENERATED IN BLADEGEN

Fig:1. Variation of beta and theta angle at different spans

Fig:2 Solid model of turbine generated in bladegen

Ansys provides the platform to create complex geometries in Bladegen. The present model includes predefined hub, shroud and blade profile coordinates. The surface so generated is considered as mean surface within blade. The dimensions and design procedure has been extracted from literature provided by S K ghost [1]here we are using ten

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

number of blades.fig(1) shows the change of beta angle to the theta angle along the radius.fig(2)represent a solid model of turbine generated in bladegen.

B. Meshing

The meshing of the model has been done on Turbogrid. The complex shaped turbine has been meshed with the finest quality. For meshing it is essentially imported in turbogrid. Where hexagonal meshed element are generated by choosing fine element size. After setting topography definition, mesh data setting is used to control the number and distribution of mesh element. After correcting mesh quality on layers, we have generated 228600 nodes and 206368 elements.

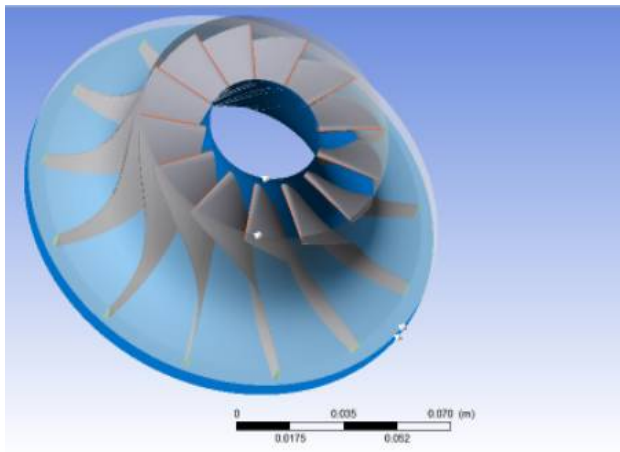


Fig3:Turbine rotor view after importing from bladegen

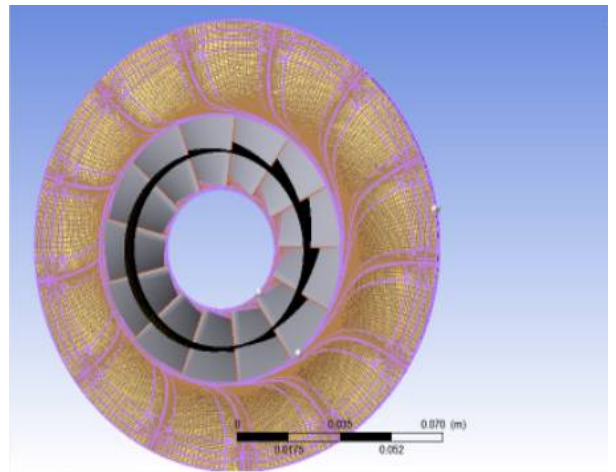


Fig4: Meshed turbine rotor in Turbogrid

C.Physical definition in CFX-Pre:

After meshing, the turbine rotor is been uploaded or imported to CFX-Pre , where the boundary conditions are been apply to the turbine .In CFX-Pre the boundary condition such as inlet temperature, inlet pressure , outlet pressure , mass flow rate and the working fluid is been chosen from material library of CFX-Pre solver are provided to the turbine. The inlet temperature is given as 23°C and ammonia is taken as working fluid.

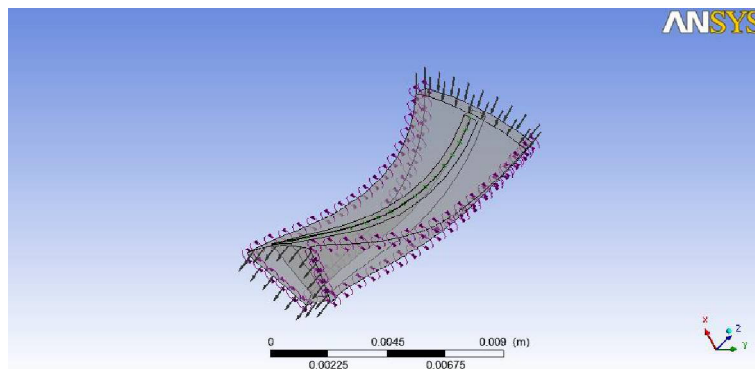


Fig 5:Flow direction at inlet and outlet of turbine blade

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

D. Analysis of CFX-Post:-

It is a flexible state and is also named as post-processor as file is imported to it from pre-processor. It gives and allows us to see easy visualization and quantitative analysis of the results obtained by CFD simulations. Turbo workspace is helpful in increasing or to improve and speed up post-processing of the turbomachinery simulation. It includes all the plotting objects like, different plans, isosurfaces, vectors, streamlines of flow, different contours, and animations of working of the machinery, etc. Fig(6) shows velocity variation at inlet, outlet and mid span of the turbine. Variation of M_{rel} and M_{abs} along the Streamwise location is given in Fig (7)

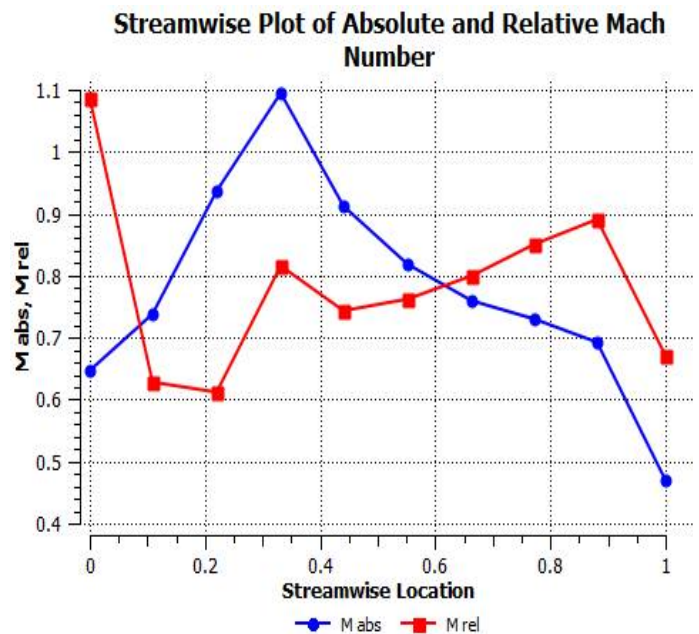
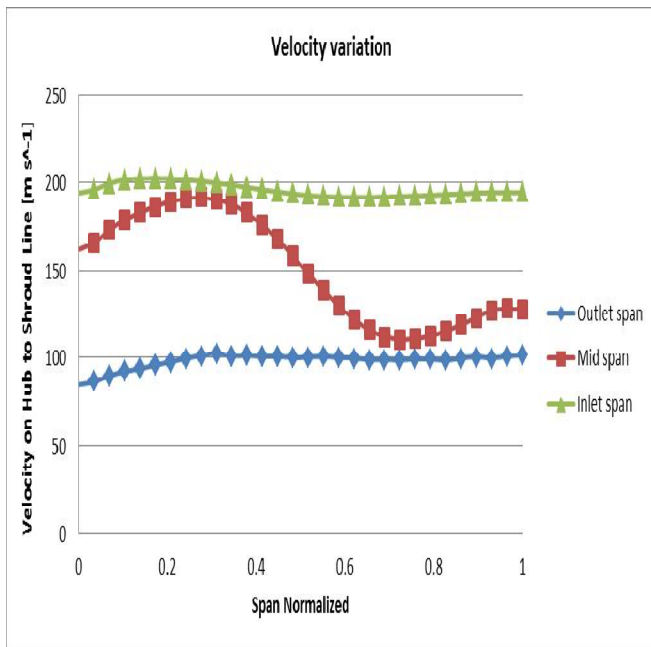


Fig 6 : Velocity variaion along streamwise inlet and outlet

Fig7: Mach number variation along Streamwise location

IV. EXPERIMENTAL RESULTS

The computational fluid flow analysis is done in CFX post after completion of cfx simulation. A tabulated result can be seen in generated report. This report gives the variation of different properties from inlet to outlet and hub to shroud, by graphs and counters. Variation of thermodynamic properties, at different locations of turbine rotor is as following.

The velocity contour show the area of high and low velocity throughout the turbine rotor and blades as shown in Fig(8). Similarly the pressure and entropy contours represent the area of high and low pressure at leading and trailing edge of the turbine rotor and the turbine blades. As we know there is also some ultimate temperature and pressure limit so these analysis through CFD provide us to take suitable input values of temperature, pressure and velocity.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

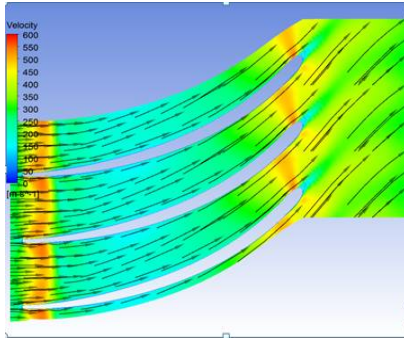


Fig 8: velocity contour across blades

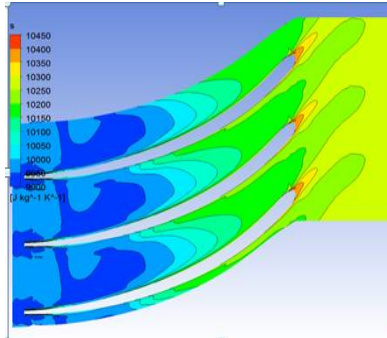


Fig9: entropy contour across blades

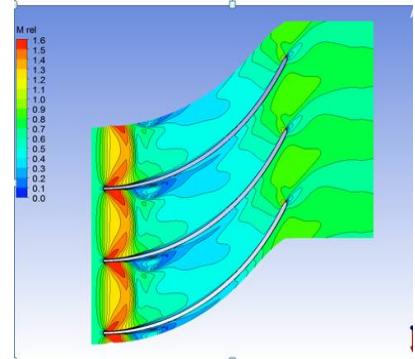


Fig 10: Relative Mach no

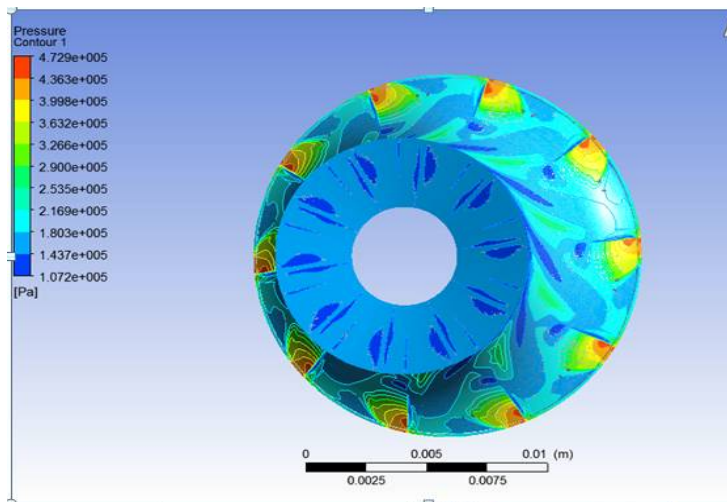


Fig11: pressure contour across turbine rotor

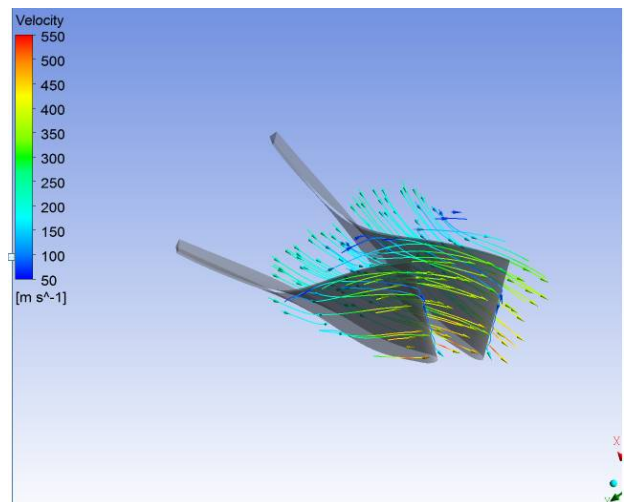


Fig 12: velocity flow across blades

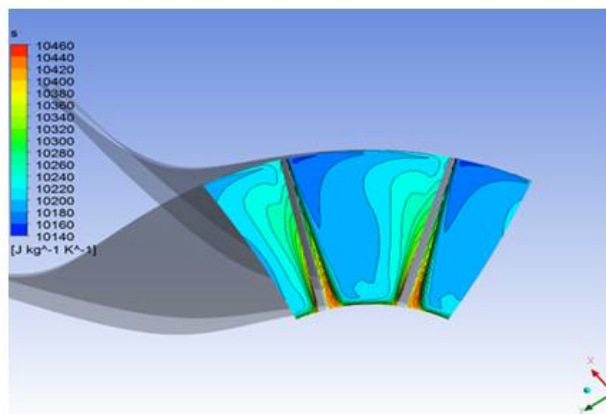


Fig13: Coutour of entropy variation at blade TE

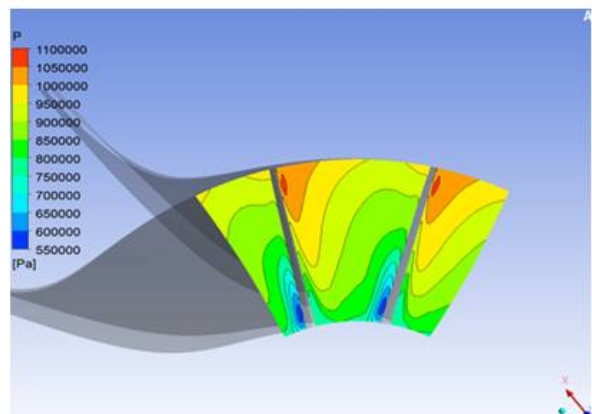


Fig14: Pressure contour at Blade TE

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

V. CONCLUSION

This work is a modest attempt of flow analysing inside a turbo expander of an OTEC system through computational fluid flow dynamics. A prototype turbine has been designed, meshed and then simulated using appropriate boundary conditions. The design procedure is covering the design of hub, shroud and brake profile that is done in bladegen of ansys . A cfx model is been developed for analysis of flow inside turbine rotor. The modelling of the various parts of the turbine is done bladegen and the meshing is done in turbogrid, the computational fluid flow analysis is done in cfx. Various graphs pressure, velocity, entropy, Mach number inside the turbine along the streamline are obtained by the analysis. Taking ammonia as the working fluid is the recent best choice as the above analysis is also done by taking R22,R134a from which it has been concluded that the optimal value of Mach number, pressure ratio, mass flow to get more efficient turbine used in OTEC is achieved as comparing results to the value achieve by Nithesh[3] in design and numerical analysis of Turboexpander used in OTEC system.

REFERENCES

- [1] SK Ghosh, R.K Sahoo, S.K. Sarangi, Computational Geometry for the blades and internal flow channels of cryogenic turbine.
- [2] Hitesh Dimri, R.K Sahoo, Balaji Kumar Choudhary, ICARI-2014. Computational Fluid Flow Analysis of Cryogenic Turboexpander.
- [3] K.G. Nitesh, Dhiman Chatterjee, Cheol Oh, Young-Ho Lee, C. 2015. Design and Performance Analysis of Radial – inflow turboexpander for OTEC application.
- [4] Daniele Fiaschi, Giampaolo Manfrida, Francesco Maraschiello, C. 2014. Design and Performance prediction of radial ORC turboexpanders
- [5] Laxminarayana, B., 1996. Fluid Dynamics and Heat Transfer in Turbomachinery, John Wiley