Numerical Investigation to Visualize the Flow Field Characteristics of a Cryogenic Turboexpander

Manoj Kumar[1], Debashis Panda[2]
[1,2] Ph.D. Scholars,
Department of Mechanical Engineering,
National Institute of Technology Rourkela,
Odisha, India, 769008

Ranjit K. Sahoo[3], Suraj. K. Behera[4]
[3,4]Professor,
Department of Mechanical Engineering,
National Institute of Technology Rourkela,
Odisha, India, 769008

***Abstract*–The design and optimization of blade profile of cryogenic radial expansion turbine play a significant role in the development of an efficient turboexpander due to increasing demand of cryogenic fluids in research and industrial applications. The primary objective of the present studies is to propose an optimized blade profile for a radial turbine with nitrogen as a working fluid which is a part of turboexpander. Adequately designed turbine blade profile can increase the efficiency of the liquefaction unit. In this regard, the three-dimensional numerical analysis is performed using the shear stress transport turbulence model in ANSYS CFX. The pressure, velocity, temperature, static enthalpy and entropy are reported. With numerical results, the initial profile is optimized to enhance the efficiency of the turbine. The results obtained from the numerical analysis visualize the fluid flow physics inside the turbine. The designed model can predict turbine efficiency and power with an accuracy of ±16% of operating conditions.**

Keywords: Turboexpander, Fluid flow, Thermal behavior, CFD

I. INTRODUCTION

Cryogenic liquefaction system requires some essential components, such as expansion turbine, nozzle, diffuser, brake compressor, bearings, heat exchangers, etc. Out of which, expansion turbine plays a vital role to increase the efficiency of the turboexpander. In this regard, the computational study is essential to visualize the fluid flow characteristics. Researchers suggest that the radial turbine is ideally suited for these types of systems. In the design of turbo-expander for the liquefaction of various cryogenic gases, the design of turbine plays an important role, which affects the performance of the system. A two-stage expansion turbine was developed by Yang et al. [1] and performed the experiments to produce 1.5 l/hr of liquefied helium A small high-speed turboexpander operating at 600,000 rpm with externally pressurized gas bearings was designed by the National Bureau of Standards (NBS), USA [2]. The CFD analysis of helium and nitrogen turboexpander was performed by Sam and Ghosh [3] to develop a helium turboexpander with a variable flow capacity mechanism.

In this paper, the design of a radial turbine and numerical analysis to visualize the flow field behavior are determined using nitrogen. Initially, the blade profile of a turbine is obtained from Blade-Gen using the operating conditions as mentioned in Table I. The computational grid is created using Ansys ICEM. Finally, the three-dimensional numerical simulation is performed to visualize the fluid flow and thermal characteristics of the turboexpander using the commercially available software ANSYS® CFX. The pressure, Mach number, velocity, temperature, static enthalpy and entropy contours are obtained at different locations of the turboexpander. The development of an experimental set-up is under process in our laboratory.

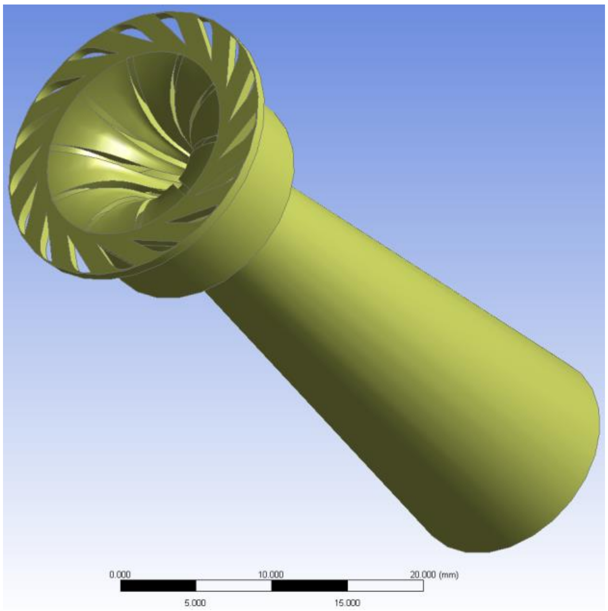
TABLE 1: SPECIFICATION OF THE DESIGNED TURBOEXPANDER

Parameter	Value
Inlet Pressure	8 bar
Temperature	150 K
Pressure ratio	3.86
Turbine diameter	24.86 mm
Turbine inlet blade height	1.36 mm
Turbine outlet blade height	4.54 mm
Axial length of the turbine	10.52 mm
Number of turbine blades	13
Number of nozzles	17
Tip clearance	0.2 mm

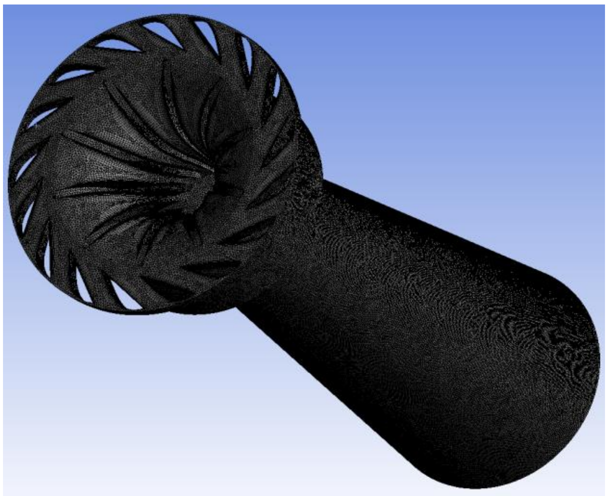
II. NUMERICAL SET-UP AND BOUNDARY CONDITIONS

The three-dimensional Reynolds-averaged Navier-Stokes equation is used to investigate the flow field and thermal behavior of the turboexpander. The high-resolution scheme (second order) is used to discretize the convection term whereas spatial derivatives of diffusion terms are computed by shape functions using finite element approach. The fluid is assumed to be an ideal gas.

The computational grid is generated using ANSYS ICEM software. Fig.1 shows the three-dimensional model of turboexpander and its mesh.



(a)



(b)

Fig 1: (a) Three-dimensional model of a turboexpander (b) Mesh generation using ANSYS ICEM

Total pressure and total temperature is imposed at the nozzle inlet boundary. The mass flow rate is set at the outlet of the diffuser. The wall is treated as adiabatic with no slip boundary condition. The boundary conditions are mentioned in Table II.

TABLE II: BOUNDARY CONDITIONS OF A TURBOEXPANDER

Parameter	Value
Total inlet pressure	8 bar
Total inlet temperature	150 K
Rotational speed of the turbine blade	86,496 rpm
Mass flow rate outlet	0.05 kg/s
Walls	Adiabatic and no-slip
Working fluid (Ideal gas)	Nitrogen

Turbulence intensity is set to be 5%. Convergence criteria are taken as the maximum residuals for mass and momentum equations are less than 10^{-6} as recommended in the ANSYS CFX user manual.

The governing equations such as continuity, momentum, and energy equations, which are used in the numerical simulation are mentioned as follows:

$$\frac{\partial \rho}{\partial t}+\nabla.\left(\rho U\right)=0 \tag{1}$$

$$\frac{\partial\left(\rho U\right)}{\partial t}+\nabla.\left(\rho U\otimes U\right)=-\nabla p+\nabla.\tau+S_M \tag{2}$$

$$\frac{\partial\left(\rho h_{tot}\right)}{\partial t}-\frac{\partial p}{\partial t}+\nabla.\left(\rho U h_{tot}\right)=\nabla.\left(\lambda\nabla T\right)+\nabla.\left(U.\tau\right) \\ +U.S_M+S_E \tag{3}$$

For the present study, the Shear Stress Transport (SST) k- ω turbulence model is opted for the turbulence closure and flow separation effects on the eddy viscosity. This turbulence model can accurately predict the turbomachinery flows having boundary layer separations [4]. It combines the advantages of two commonly used turbulence models: the k- ω model and k- ε model. The k-ω model can predict the flow near-wall region whereas k-ε model away from the wall. In this aspect, the k-ω SST model can predict the boundary layer through the passage. The model is most accurate for flow field like airfoil, shock waves, adverse pressure gradient etc. The k- ω transport equations are as follows:

$$\frac{\partial}{\partial t}\left(\rho k\right)+\frac{\partial}{\partial x_i}\left(\rho k u_i\right)=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial k}{\partial x_j}\right)+G_k-Y_k+S_k \tag{4}$$

$$\frac{\partial}{\partial t}\left(\rho \omega\right)+\frac{\partial}{\partial x_i}\left(\rho k\omega u_i\right)=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial \omega}{\partial x_j}\right)+G_\omega-Y_\omega+S_\omega \tag{5}$$

Where G_k, Y_k, and S_k are the generation of turbulent kinetic energy, the fluctuating dilation and the source term. Similarly, G_ω, Y_ω, and S_ω represents the dissipation rate of turbulent kinetic energy, the fluctuating dilation and the source term.

III. RESULTS AND DISCUSSION

The primary numerical results are used to optimize the blade profile by resizing the high-pressure and vortex formation zone in BladeGen. The final optimized meridional profile of turbine is shown in Fig. 2.

The temperature drop is the primary aim of the cryogenic turboexpander. Simulation contours are represented in terms of contours at different positions in nozzle and turbine. Fig. 3 illustrates the pressure and temperature contour at nozzle and turbine mid cross-sections along the flow passage are taken and compared. It shows that the drop in temperature is approximately 26 K. The temperature drop is obtained due to high rotational speed of the turbine blade which converts the pressure energy into the kinetic energy. During the energy transformation process, the static enthalpy reduces which is responsible for the refrigerating capacity of the turboexpander (Fig. 4(a)). Similarly, the static entropy increases in the nozzle and turbine. The increase in entropy inside the turbine is higher as compared to nozzle because of rotational speed and higher pressure drop (Fig. 4(b). Therefore, the secondary losses, boundary layer separation, and vortex formation in the blade passage is usual phenomenon which is reduced in the refined blade profile.

The Mach number and velocity contours are illustrated in the Fig. 5. It is observed that the Mach number at the inlet of the turbine is approximately 0.70 which is the desirable condition. In this zone the maximum velocity of the fluid is 168 m/s.

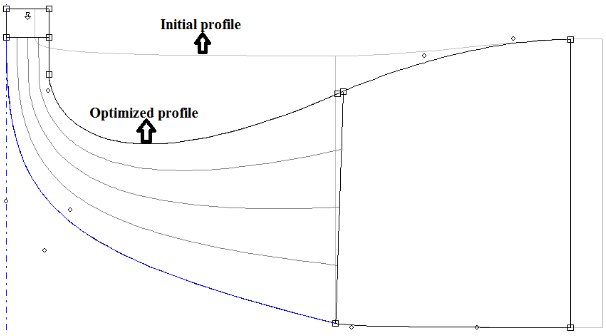
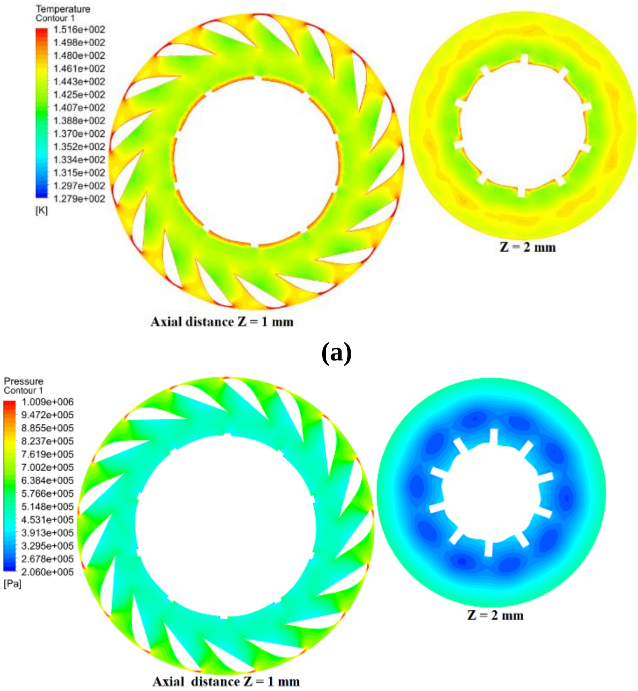


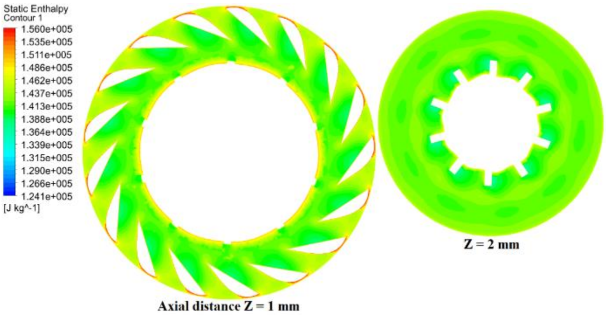
Fig 2: Initial and optimized meridional profile experimental test-rig development



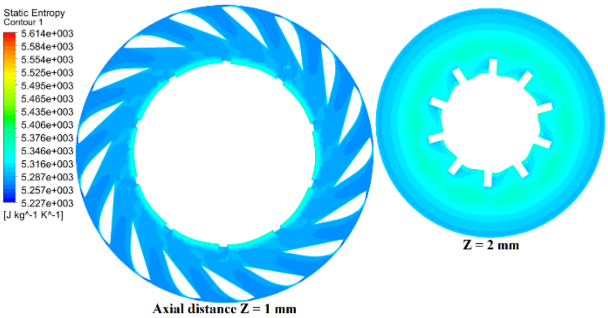
(a)

(b)

Fig 3: Temperature and pressure contours at different cross-sections

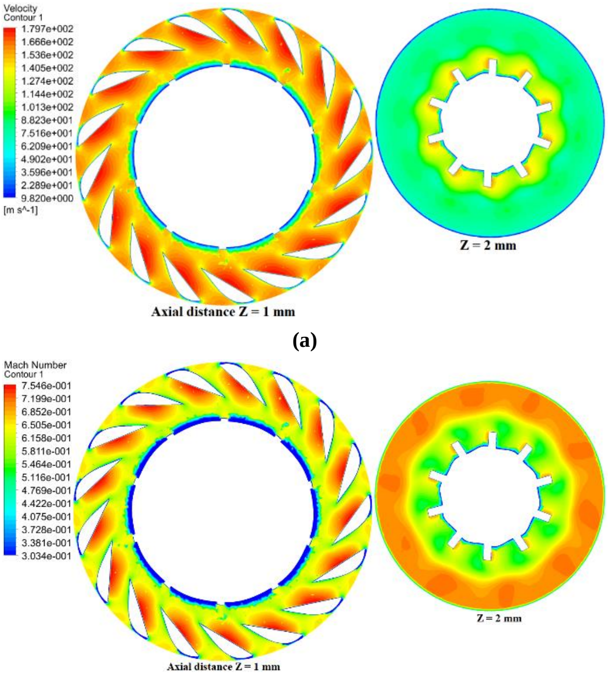


(a)



(b)

Fig 4: Static enthalpy and static entropy contours at different cross-sections



(a)

(b)

Fig 5: Velocity and Mach number contours at different cross-sections

IV. CONCLUSION

The present study reports numerical analysis to visualize the fluid flow and thermal characteristics of a cryogenic radial turbine. The blade profile, blade angle, meridional plane, blade thickness variation, and the number of blades are created using Blade-Gen. Pressure, velocity, Mach number, temperature, etc. are obtained at different locations using nitrogen as a working fluid. It is observed that the maximum temperature drop inside the turbine is approximately 26 K for a pressure ratio of 3.86. The Mach number variation is also to be in the subsonic regime which is desirable for nitrogen liquefaction unit. Based on the design data and numerical work, the experimental test-rig is under development.

REFERENCES

[1] K. J. Yang, H. B. He, G. Ke, and G. Y. Li, "Application and Test of Miniature Gas Bearing Expansion Turbines." Advances in Cryogenic Engineering. Springer, Boston, MA, 1990. 997-1003.

[2] H. Sixsmith, and W. Swift. "A pair of miniature helium expansion turbines." Adv. Cryog. Eng.;(United States)27.CONF-810835- (1982).

[3] A. A Sam and P. Ghosh, "Influence of thermophysical properties of working fluid on the design of cryogenic turboexpanders using N_sD_s diagram", Material Science and Engineering 101, 012179 (2015).

[4] M. Kumar, R. K. Sahoo, and S. K. Behera. "Design and numerical investigation to visualize the fluid flow and thermal characteristics of non-axisymmetric convergent nozzle." Engineering Science and Technology, an International Journal (2018).