Numerical Analysis to Predict the Fluid Flow Pattern Through Convergent Nozzle

Manoj Kumar*, A. K. Gupta, P. Kumar
Ph.D. Scholar, Cryogenics Engineering Laboratory,
Department of Mechanical Engineering
National Institute of Technology, Rourkela
Odisha, India, 769008

R. K. Sahoo
Professor, Cryogenics Engineering Laboratory,
Department of Mechanical Engineering
National Institute of Technology, Rourkela
Odisha, India

***Abstract*—Current paper reports the fluid flow characteristics inside an airfoil convergent nozzle used in a turboexpander. The coordinates of upper (5th order) and lower (3rd order) surface of the nozzle is obtained using curve fitting method in Matlab© after that the computational domain has been generated in SolidWorks©. Numerical analysis has been done to visualize the fluid flow and thermal characteristics inside the nozzle at a pressure ratio of two. Shear stress transport (SST) turbulence model has been used to solve the computational domain. The key feature of this implementation is to obtain subsonic velocity at the nozzle exit. The computing results show that the reduction in temperature for air is approximately 14.2 K with Mach number of 0.88 at the outlet.**

Keywords—Fluid flow; Convergent nozzle; Ideal gas

I. INTRODUCTION

Cryogenic fluids like liquid helium, nitrogen, oxygen, hydrogen, etc. are used due to its variety of applications in the fields such as rocket propulsion and aerospace appliances, superconducting equipment, industrial applications, etc. The design of an efficient nozzle plays a vital role in this process. In this framework, researchers are interested to design an efficient and optimized nozzle profile, which is desirable to minimize the losses and compact size of a turboexpander [1, 10].

The effect of geometrical parameters on the performance of the supersonic micro-nozzle which reduces the thrust losses by increasing the divergent angle above 300 and discussed the thermal, viscous and effect of rarefaction on the performance of nozzle using continuum approach [2]. Researchers suggest that shock wave generation and boundary layer separation in over-expanded supersonic nozzles have unsteadiness during the flow, which causes unsymmetrical flow separation [3]. The experimental analysis of double divergent nozzle to find the shock wave behavior under over-expanded regimes and concluded that the flow field was similar to that of dual-bell nozzle [4]. Geron et al. [5] investigated the three-dimensional flow field inside a plug nozzle by partitioning it into different modules to enhance the performance of rocket engines. The selection of cross-section at the exit plays a vital role to reduce the thrust loss. Researchers suggest that rectangular cross-section possess less thrust loss as compare to circular one whereas density profiles through the circular nozzle have good agreement with the experimental results as compare to that of for the rectangular or square nozzles [6]. The numerical work was carried on to visualize the oscillatory flow pattern and hysteresis phenomenon for different pressure ratios between two shock structures (FSS and RSS) and its effect on annulus height, wall pressure and shear stress in axisymmetric and truncated contour nozzle [7]. The computational work to visualize the effect of nozzle pressure ratio on flow structure, shock-induced boundary layer separation inside a non-axisymmetric supersonic convergent-divergent nozzle was performed by Hasan [8]. Mousavi and Roohi [9] investigated the shock train in a three-dimensional convergent-divergent nozzle for compressible and turbulent fluid flow using the Reynolds stress turbulence model (RSM) which is validated with the experimental data.

In this paper, the convergent type non-axisymmetric airfoil nozzle has been designed which can convert the pressure and thermal energy of the fluid into kinetic energy to achieve subsonic velocity at the outlet, which is essential for an effective design of turboexpander used in gas liquefaction applications. The design method is based on 3rd and 5th order curve fitting under a specified boundary condition which is suitable for the mean-line design and provides uniform flow at the inlet of the turbine. The flow behavior of cryogenic fluid air is predicted inside the nozzle using commercially available software ANSYS CFX©. This type of nozzle is essential for mixed-flow type having radial inlet and an axial outlet. The rotor is of impulse type with a small amount of reaction. The expansion ratio in the nozzle is approximately two. For exact analysis point of view, experiments provided the exact results, but it is expensive. Accurate prediction of flow field behavior, flow separation, pressure, velocity, Mach number, temperature, TKE, etc. inside the domain is necessary to design a nozzle for high performance of the turboexpander.

II. MATHEMATICAL MODEL

The physical model considered for the present study is non-axisymmetric convergent nozzle designed from the curve-fitting approach. The oblique shocks occurred inside the nozzle can be minimized by obtaining the continuous contour profile of a nozzle. Due to this, the curve-fitting method has been used. The slope of the nozzle wall is continuously decreasing (convergent type) from the inlet to the outlet. After the outlet of the nozzle, it is assumed that the fluid strikes the rotor tangentially. From the manufacturing point of view, the clearance of 1 mm has been taken in the present case.

A. Turbulence model and discretization scheme

The Shear-Stress Turbulence (SST) model is widely used due to it possesses combined advantages of k-ω near walls and k-ε in wakes and free-shear regions in the outer boundary layer. It can also control the eddy-viscosity by restricting the turbulent shear stress, which improves the model performance in adverse pressure gradients and flows separation cases in particular. The governing equations, mass, momentum, and energy along with pressure based solver transient formulation are adopted in this case. To achieve the accurate simulation results, the selection of the numerical scheme is equally important as the turbulence model. Due to this reason, in the current study, the second-order upwind scheme is selected for discretization of momentum and energy equations.

B. Governing equations

Governing equations, which is used to solve the computational domain, are as follows:

Continuity Equation:

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

Momentum Equations:

$$\frac{\partial(\rho U)}{\partial t} + \nabla.(\rho U \times U) = -\nabla p + \nabla.\tau + S_M \quad (2)$$

Where the stress tensor τ is related to the strain rate as:

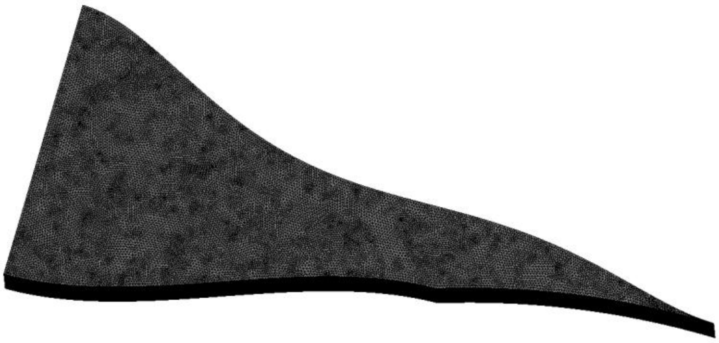
$$\tau = \mu\left(\nabla U + (\nabla U)^T - \frac{2}{3}\delta\nabla.U\right) \quad (3)$$

Total Energy Equation:

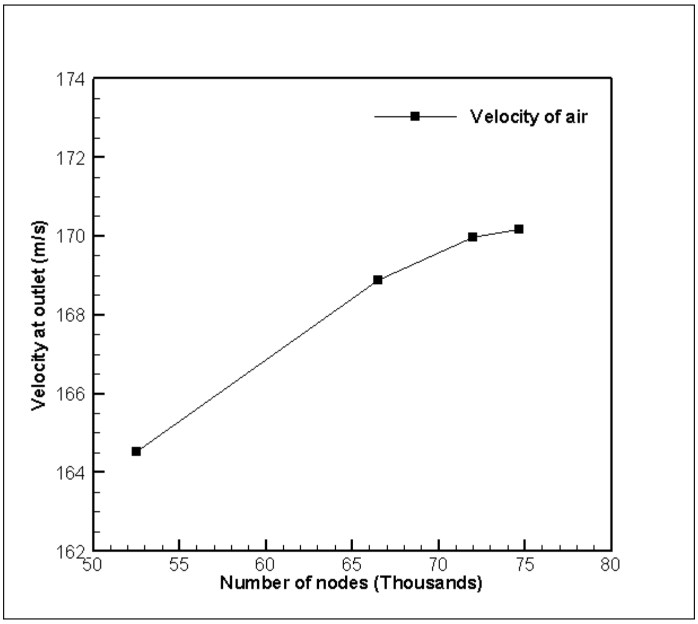
$$\frac{\partial(\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla.(\rho U h_{tot}) = \nabla.(\lambda \nabla T) + \nabla.(U.\tau) + U.S_M + S_E \quad (4)$$

Where h_{tot} is the total enthalpy, which is related to static enthalpy:

$$h_{tot} = h + \frac{1}{2}U^2 \quad (5)$$



(a)



(b)

Figure1. (a) Computational mesh used in this study (b) Grid independence test

C. Boundary conditions

Time integration uses a second-order implicit scheme for the advancement of the solution with a specified linear solver. The upwind and high-resolution methods are selected for a different scheme. The CFD models and their boundary and initial conditions are shown in Table 1. The convergence criteria (RMS) and conservation target are set to be 10-6 and 0.001 respectively. For dynamic model control, automatic pressure level information, temperature damping, and velocity pressure coupling with Rhie-Chow fourth order model are used.

D. Grid independence test

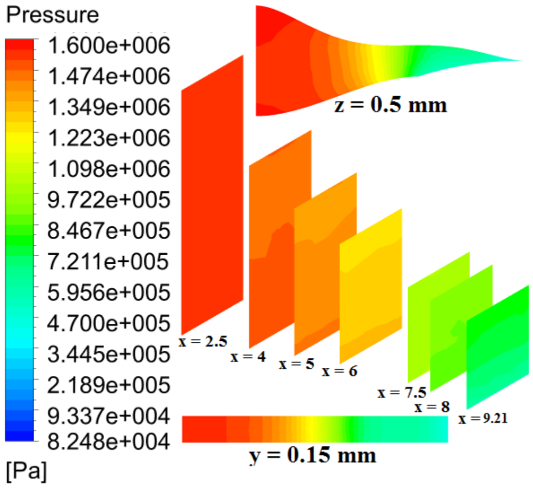
To compute the flow field inside the nozzle, a grid independence analysis is done for air with the four different mesh resolutions. Figure 1 represents the velocity at the outlet of the nozzle for four different mesh. Further refinement is not realistic due to very small changes in results but takes more computational times. The computational mesh inside the domain contains 71,972 nodes which are sufficient to predict the flow and thermal characteristics as shown in Figure 1 (a).

Table 1. Boundary conditions

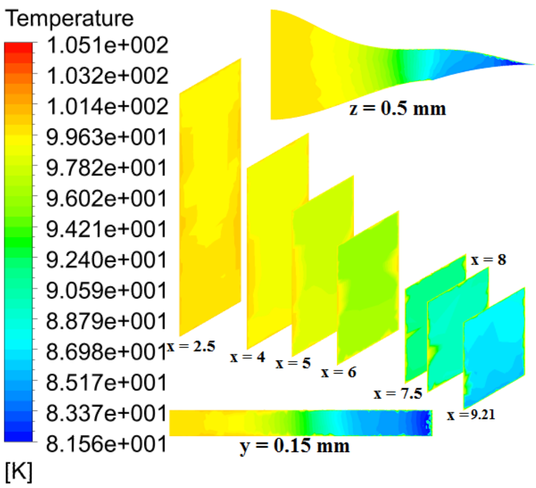
	P (bar)	T (K)	Mass flow rate (kg/s)
Inlet	16	100	-
Outlet	-	-	0.01
Wall	No-slip	Adiabatic	

III. RESULTS AND DISCUSSION

The flow and thermal behavior of air, assuming as an ideal fluid, has been predicted inside the nozzle. Generally, ideal nozzles provide uniform and parallel flow at the exit. Figure 2 (a) represents the variation of pressure along the axial distance at different planes for the pressure ratio of two. It clearly shows that the pressure variation inside the domain is from 16 bar (inlet) to 8 bar (outlet).



(a)

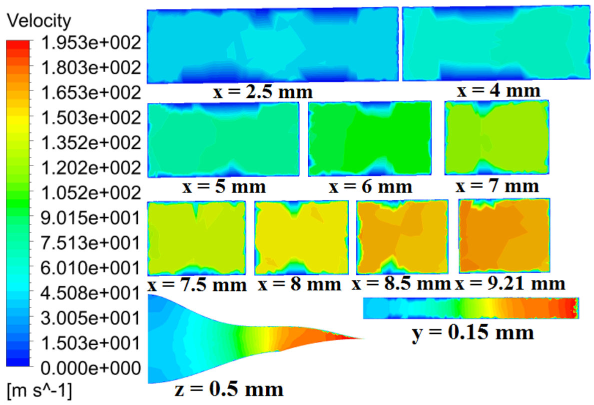


(b)

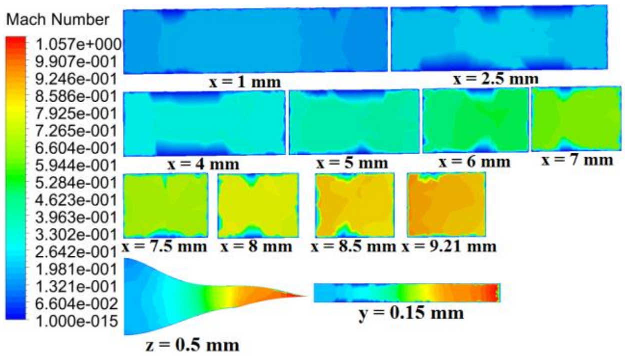
Figure 2. Contour of (a) Pressure (b) Temperature at different locations inside the domain

Figure 2 (b) represents the temperature contours of at different cross-sections. It shows that the temperature of the air at the outlet of the nozzle is 86.5 K which shows the decrement of 14.5 K as compared to that of an inlet. This decrease in temperature is very much significant at ultra-low temperature because it happens inside the nozzle which reduces the turbine work and hence increases the efficiency of turbo-expander. It is desirable for the liquefaction of gases.

Figure 3 represents the velocity and Mach number profile along the axial distance. The velocity at axial distance x = 2.5 mm is approximately 35.20 m/s and continuously increases as one move towards the outlet where it is approximately 166.00 m/s with Mach number of 0.88. It is also observed that the velocity increases rapidly after x = 5 mm and becomes stable at the time of impact with the turbine (outlet of the nozzle) due to which a uniform subsonic flow has occurred. The Mach number at the outlet for helium is lowest for this nozzle.



(a)



(b)

Figure 3. Contour of (a) Velocity (b) Mach number at different locations inside the domain

IV. CONCLUSIONS

The present work reports the numerical investigation to visualize the fluid flow properties such as velocity, Mach number and thermal properties temperature, pressure, etc. inside the non-axisymmetric convergent nozzle. Shear stress transport (SST) turbulence model is applied in the current simulation. Three-dimensional CFD model for turbulent fluid flow inside the convergent nozzle is simulated. The interesting point of this study is the reduction in temperature for air is around 14.2 K with subsonic Mach number of 0.88 at the outlet of the nozzle. The investigation shows the productive results for the design and numerical analysis of nozzles used in turbomachinery applications.

REFERENCES

[1] Kumar, M., Behera, S. K., Sahoo, R. K. “Design and Numerical Analysis of Fluid Flow Characteristics in a Non-axisymmetric Convergent Nozzle.” 5th International Conference on Computational Methods for Thermal Problems. 2018

[2] Louisos, William, and Darren Hitt. "Optimal expansion angle for viscous supersonic flow in 2-D micro-nozzles." 35th AIAA Fluid dynamics conference and exhibit. 2005.

[3] Hadjadj, A., Y. Perrot, and S. Verma. "Numerical study of shock/boundary layer interaction in supersonic overexpanded nozzles." Aerospace science and technology 42 (2015): 158-168.

[4] Arora, Rajat, and Aravind Vaidyanathan. "Experimental investigation of flow through planar double divergent nozzles." Acta Astronautica 112 (2015): 200-216.

[5] Geron, Marco, et al. "Flowfield analysis of a linear clustered plug nozzle with round-to-square modules." Aerospace Science and Technology 11.2-3 (2007): 110-118.

[6] Nguyen, T. V., R. G. Spencer, and A. Siebenhaar. "Aerodynamic performance of a round-to-square nozzle." (1997).

[7] Lijo, Vincent, et al. "Numerical simulation of transient flows in a rocket propulsion nozzle." International Journal of Heat and Fluid Flow 31.3 (2010): 409-417.

[8] Hasan, ABM Toufique. "Characteristics of overexpanded nozzle flows in imposed oscillating condition." International Journal of Heat and Fluid Flow 46 (2014): 70-83.

[9] Mousavi, Seyed Mahmood, and Ehsan Roohi. "Three dimensional investigation of the shock train structure in a convergent–divergent nozzle." Acta Astronautica 105.1 (2014): 117-127.

[10] Kumar, M., Behera, S. K., Sahoo, R. K. “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) (Accepted)