ELSEVIER

CFD analysis of turbo expander for cryogenic refrigeration and liquefaction cycles

Rahul Verma, Ashish Alex Sam, Parthasarathi Ghosh*

Cryogenic Engineering Centre, Indian Institute of Technology Kharagpur, India-721302

Abstract

Computational Fluid Dynamics analysis has emerged as a necessary tool for designing of turbomachinery. It helps to understand the various sources of inefficiency through investigation of flow physics of the turbine. In this paper, 3D turbulent flow analysis of a cryogenic turbo expander for small scale air separation was performed using Ansys CFX®. The turbo expander has been designed following assumptions based on mean line blade generation procedure provided in open literature and good engineering judgement. Through analysis of flow field, modifications and further analysis required to evolve a more robust design procedure, have been suggested.

1. Introduction

Cryogenic process plants in recent years are almost exclusively based on the low-pressure cycles which use a radial inflow expansion turbine to generate refrigeration. Ghosh (2002) has pointed out that the use of radial inflow expansion turbines brings forth attractive features like better efficiency and reliability in the operating range of the cryogenic process plants, which considerably affects the economical parameters of the process plants. Agahi and Ershaghi (1991) has found out that the efficiency of the expander is one of the major factors on which the

* Corresponding author. Tel.: +91-993-258-4162.
  *E-mail address:*psghosh@hijli.iitkgp.ernet.in

performance of the liquefier is depended. Thomas et al. (2011) has studied the impact of expander isentropic efficiency on the performance of Collins liquefaction cycle. It has been observed that liquid yield increases almost linearly with increase in efficiency.

Improving the efficiency of a turbo expander is possible by developing a design procedure through a deep understanding of the flow characteristics and the origins and effects of various losses. Sauret (2012) has mentioned its importance in radial turbines, where viscous 3D rotational and curvature effects and turbulence effects are present. Moreover, Ghosh (2002) has mentioned that the design of turbo expander for liquefaction systems is critical at low temperatures due to process conditions such as lower volumetric flow rate as compared to the high temperature turbines and variation of thermophysical properties of fluids at low temperature. CFD allows one to determine the flow and thermodynamic parameter fields, which is often not possible through experiments.

In the present work, CFD techniques has been applied to analyze the performance of a turbine that has been designed by Ghosh (2002) for small scale air separation. The expected performance based on the design methodology has been compared with the data obtained from the simulation. Through the analysis of flow field the sources of losses that are responsible for deviation have been explored. Important geometrical parameters have been identified for parametric analysis in order to evolve a modified design.

2. Brief description of design methodology and major dimensional parameters

One dimensional mean line design methodology obtaining the overall geometrical dimensions of the components of the turbo expander is adopted from Kun and Sentz (1985) which is based on Balje's (1981) n_s-d_s chart, empiricisms based on practices for other turbine applications and good engineering judgement. The blade for turbine has been generated following methodology prescribed by Hasselgruber (1958) and Balje (1981). The input conditions for the turbine are presented in Table 1. Fig. 1 and Fig. 2 provide the major dimensions of different components of the turbine.

Table 1.Turboexpander specifications and boundary conditions for CFD simulations

Total pressure at inlet	$P_{0,in}$	6.0 bar
Total temperature at inlet	$T_{0,in}$	120 K
Static pressure at exit	P_{ex}	1.5 bar
Mass flow rate	$\dot{m}$	0.06 kg/s
Working fluid		Nitrogen
Expected efficiency	$\eta_{T\text{-st}}$	75%
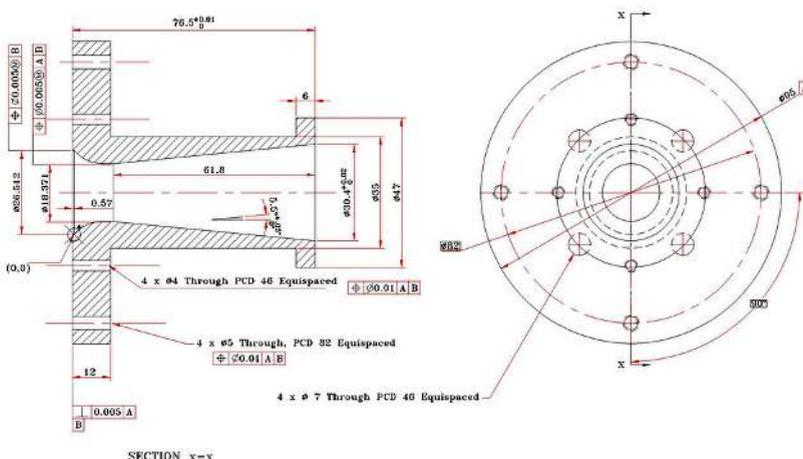
Fig. 1. Major dimensions of the diffuser
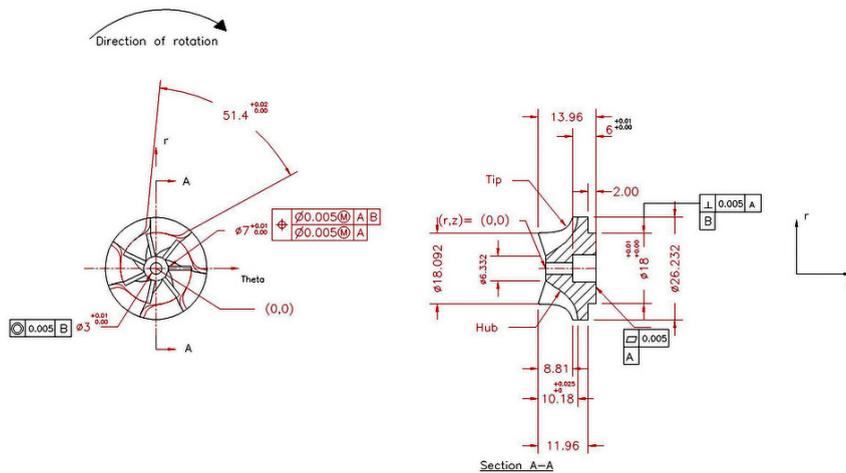
Fig. 2. Major dimensions of the expansion turbine

3. Methodology

3.1 Geometry and grid

DASSAULT SYSTEMES SolidWorks® was used for creating three-dimensional model for fluid domain of nozzle and diffuser section and was then exported to ANSYS DesignModeler®. The three-dimensional model of turbine blade was created in ANSYS BladeGen® following Hasselgruber's (1958) design methodology and it was then meshed in ANSYS TurboGrid®. The details of mesh characteristics have been provided in Table 2.

Table 2. Mesh specifications for various components

Domain	Number of nodes	Number of elements	Method	Mesh type/ type of elements
Nozzle	72928	54194	Sweep	Unstructured/ Mostly Hexahedral, small no. of wedge
Turbine	557487	506639	ATM optimized	Structured (H and O type topology)/ Hexahedral only
Diffuser	39452	210157	Patch conforming method	Unstructured/ Tetrahedral

3.2 Boundary conditions

Mass flow rate of 0.06 kg/s and inlet total temperature of 120 K were specified at the nozzle inlet which is the inlet boundary. Mass influx was assumed to be uniformly distributed over entire boundary. At the diffuser outlet, static pressure of 1.5 bar has been used for specifying outlet boundary condition. Flow regime is subsonic in nature at both inlet and outlet surface. A medium turbulent intensity of 5% was assumed for the present simulation. All walls were considered as smooth, no slip and adiabatic.

3.3 Grid independence test

Grid independence test was done to study the effect of grid size on the results. This involved analysis of the computational results of different size meshes to check whether there is any significant variation in the results. The number of nodes for each component of the turboexpander as given in Table 1, was chosen based on this grid independence study.

3.4 Numerical model

Flow inside turbo expander is highly turbulent. RANS (Reynolds–Averaged Navier–Stokes) equation based SST (Shear Stress Transport) turbulence model was used for turbulence modelling. Gatski et al. (2007) has described this model as a blend of the original k-ω for near wall and k-ε for free stream calculations

ANSYS CFX® is a coupled solver, in which all the hydrodynamic equations are solved in a single system. High resolution advection scheme has been used for present case. Frozen rotor model was used for modelling the rotor stator interface. Ideal gas equation of state has been used for present simulation.

4. Results and discussion

The computations were done till the required convergence criteria of 0.0001 (Root Mean Square) were satisfied for all the conservation equations. Initially, the computational results for the nozzle and turbine efficiencies and power developed were compared with that of the 1D mean line analysis (Table 3). A detailed analysis of CFD results was done to identify the different sources of losses in the various components of the turbo expander.

Table 3. Comparison of 1D mean line analysis and CFD analysis

Parameters	1D mean line analysis	CFD analysis
Nozzle Efficiency	93%	94.1%
Turbine efficiency (total to static)	75%	75.13%
Power developed	1.73 kW	2.09 kW
Diffuser pressure recovery factor	0.7	0.5607

4.1 Nozzle

Nozzle accelerates the fluid and guides it to the rotor blades, with low incidence. The expansion in a nozzle should take place with minimum drop in total pressure. The total pressure contour plot in Fig. 3 shows that there is a small drop in total pressure as not all the pressure available at the inlet is converted to velocity. This is because; a part of the total pressure is used in overcoming the friction. The efficiency of the nozzle from the computational results was found to be higher compared to the mean line analysis as in Table 3, as the one dimensional mean line design procedure involves certain assumptions. From the Mach number contour plot in Fig. 4, a severe but asymmetric shock wave pattern can be seen at the nozzle discharge. This strong shock leads to unsteady interaction at the nozzle – wheel interface. The shock wave pattern may be due to the arbitrary selection of number of nozzle vanes and rotor blades. The criteria for this selection is that these numbers should be mutually prime so that the periodic excitation of the wheel will be higher compared to the operating speed and thus structural vibration can be reduced.
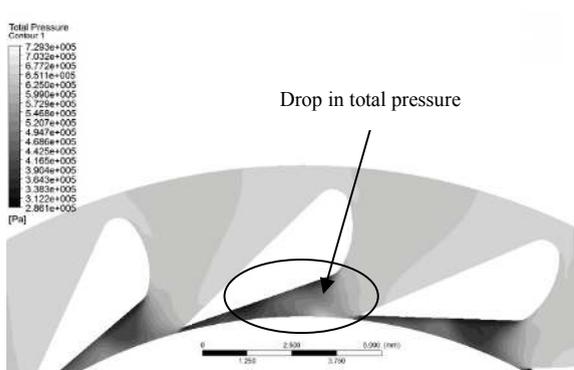
Fig. 3.Total pressure contour in the nozzle   Fig. 4. Mach Number contour in the nozzle
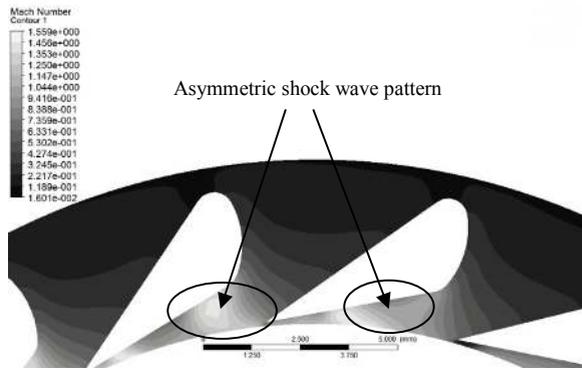
4.2 Turbine

The presence of viscous effects, secondary flows, flow separation and tip clearance makes the flow phenomenon in a radial turbine extremely complex. Laxminarayana (1996) has pointed out that any improvement in the design of a radial turbine is cumbersome as the blade curvature and rotation makes the flow field complex, three dimensional and highly turbulent. Fig. 5 and Fig. 6 shows the velocity vectors nearer to the hub and the tip in the blade to blade view. The presence of centrifugal and coriolis forces in addition to the pressure and viscous forces makes the flow inside the boundary layers different from the inviscid region. In the inviscid regime away from the walls, the pressure and inertia forces are in equilibrium, whereas an imbalance exists between the pressure force and the inertia forces within the boundary layer. This imbalance is the outcome of the retardation in the boundary layer which leads to lower inertia force as compared to that of the inviscid region as mentioned by Schobeiri (2005). Thus secondary flow is generated within the blade channel from the pressure side of one blade to the suction side of the other in the end wall boundary layers at the hub and at the shroud.

Fig. 5 also portrays the corner stall phenomena at the intersection of suction surface and end wall. This occurs as a result of the stream wise pressure gradient, secondary flow and mixing of the end and blade wall boundary layers.

The region of vortices at the trailing edge of the rotor is revealed through Fig. 6. Trailing edge losses is significant in radial flow turbines with thick trailing edges. As by Baines (1998) trailing edges are usually made thick to provide necessary strength to withstand the stress generated by the fluid. The sudden expansion from the throat condition leads to flow separation and formation of vortices which in turn results in trailing edge losses. Unsteady analysis is required to estimate the magnitude of loss due to trailing edge.

Nearer to the shroud as seen in Fig. 6 leakage of flow over the blade tip from the pressure side to the suction side takes place.The mass flow through this clearance depends on the height of the gap. Therefore blade clearance height and blade loading are the two critical parameters that influence the tip leakage flow. Denton (1993) has found out that tip leakage flow results in deteriorating the performance of the turbine. Tip clearance was identified as a major source of inefficiency in cryogenic turbo expanders for small scale air separation by Ghosh et al. (2010) through generation of characteristics curves using one-dimensional mean line approach

4.3 Diffuser

In Fig. 8 velocity along the wall is close to zero as the fluid is experiencing higher frictional losses. Flow separation has occurred at the centre, which can be rectified by change in diffuser angle, length or diffuser type itself (conical or annular).

It has been observed in Fig. 7 that at the entry of the diffuser the flow field is highly non-uniform. As the diffuser works under adverse pressure gradient, a uniform flow field is desirable at the entry of the diffuser, in order to achieve higher pressure recovery. As the flow at the trailing edges of the blades of the wheel is non-uniform, the flow straightener ahead of the diffuser should be designed to provide a uniform velocity field at the diffuser throat.
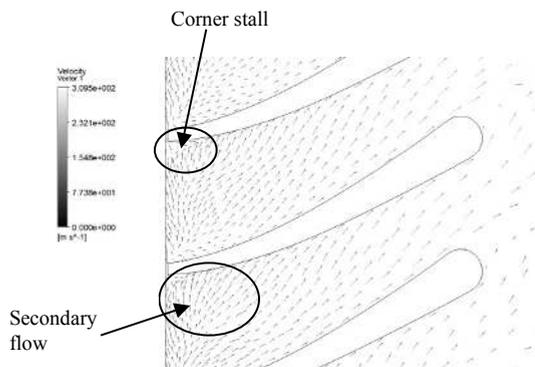
Fig. 5.Vector plot nearer to the turbine hub
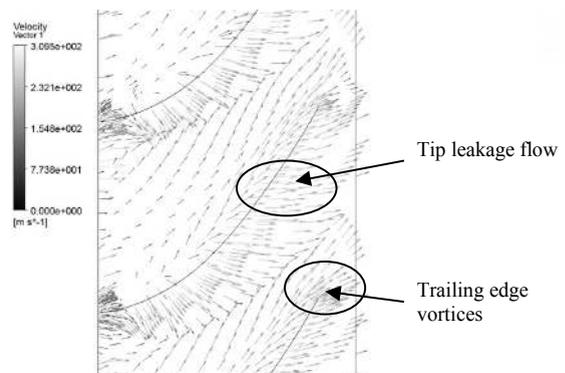
Fig. 6. Vector plot nearer to the blade tip
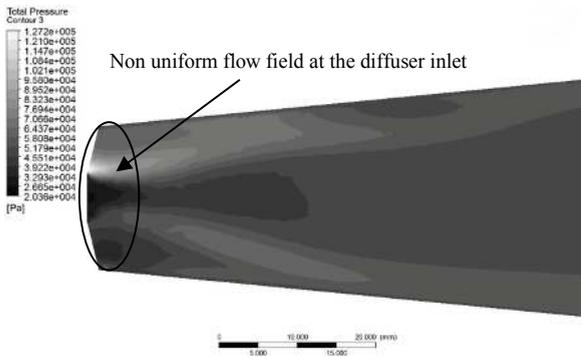
Fig. 7. Total pressure contour in diffuser
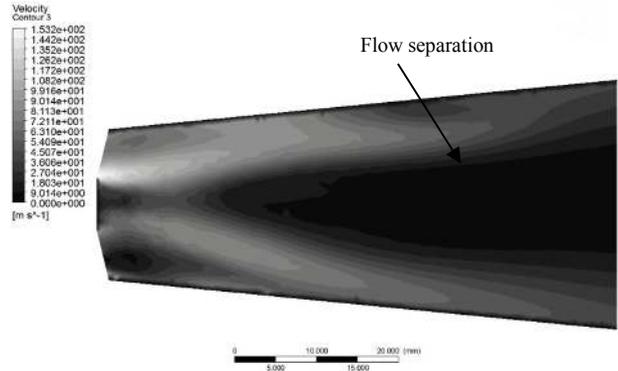
Fig. 8. Velocity contour in diffuser

5. Conclusion

From the above study it has been found that following parameters are important and parametric analysis should be performed in order to improve the design methodology.

- Number of nozzle blades
- Number of blades in the wheel
- Tip clearance of the wheel
- Trailing edge thickness and radius
- Flow straightening before the diffuser
- Diffuser length and cone angle

CFD analysis of the turbine designed based on Ghosh (2002) reveals that the efficiency of the turbine matches with the expected efficiency. It may be noted that Hasselgruber's (1958) method is based on inviscid flow assumption and can be applied for generation of preliminary blade profile as the turbine works under favourable pressure gradient. Transient models are required to understand the stator-rotor interaction and trailing edge vortices.

References

Agahi, R., Ershaghi, B., 1991. Expansion turbine in energy recovery applications.18th International Congress of Refrigeration. Montreal, Canada, August 10-17, 1991.
Baines, N. C., 1998. A meanline prediction method for radial turbine efficiency. Inst Mech Eng Conference on Turbochargers. I Mech E (UK) Publications.
Balje, O. E., 1981, Turbomachines, John Wiley and Sons.
Denton, J. D., 1993. Loss mechanisms in turbomachines.Trans ASME J Turbomachinery115 621-56.
Gatski, T. B., Rumsey, C. L., Manceau, R., 2007. Current trends in modelling research for turbulent aerodynamic flows. Philosophical Transactions of the Royal Society A 365, 2389-2418.
Ghosh, P., 2002. Analytical and Experimental Studies on Cryogenic Turboexpander. Ph.D dissertation, IIT Kharagpur.
Ghosh, P., Nandi, B. R., Sarangi, S., 2010. Prediction on performance of cryogenic turboexpander using meanline approach. Proceedings of ICEC 23 – ICMC 2010, Wroclaw, Poland, 2010, pp 977 – 984.
Hasselgruber, H., 1958.Stromungsgerechte gestaltung der laufrader von radialkompressoren mit axialem laufradeintrict Konstruction 10 (1) 22 (in German).
Kun, L .C., Sentz, R. N., 1985. High efficiency expansion turbines in air separation and liquefaction plants. International Conference on Production and Purification of Coal Gas & Separation of Air, Beijing, China, pp 1 – 21.
Laxminarayana, B., 1996. Fluid Dynamics and Heat Transfer in Turbomachinary, John Wiley.
Sauret, E., 2012. Open design of high pressure ratio radial inflow turbine for academic validation. Proceedings of the International Mechanical Engineering Congress and Exposition, ASME, Houston, Texas, USA, November 9-15, 2012.
Schobeiri, M. T., 2005. Turbomachinery Flow Physics and Dynamic Performance. Springer.
Thomas, R. J., Ghosh, P., Chowdhury, K., 2011. Role of expanders in helium liquefaction cycles: Parametric studies using Collins cycle. Fusion Engineering and Design, vol. 86, issue 4-5, pp. 318-324.