CFD analysis of turboexpander for cryogenic refrigeration and liquefaction cycles

Rahul Verma, Ashish Alex Sam, Parthasarathi Ghosh

CRYOGENIC ENGINEERING CENTRE

INDIAN INSTITUTE OF TECHNOLOGY KHARAGPUR, INDIA

Abstract

Computational Fluid Dynamics analysis of cryogenic turboexpander 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 work, 3D turbulent flow analysis of a cryogenic turboexpander for small scale air separation was performed using Ansys CFX®. The turboexpander has been designed following assumptions based on a meanline 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.

Introduction

- ❑ Efficiency of the expander is one of the major factors on which the performance of the liquefier is depended [1,2].
- ❑ Development of a turboexpander design procedure, through a deep understanding of the flow characteristics and the origins and effects of various losses is required for improving the efficiency of the expander.
- ❑ The design of turboexpander 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 thermo physical properties of fluids at low temperature [3].
- ❑ CFD allows one to determine the flow and thermodynamic parameter fields, which is often not possible through experimental techniques.

Objective

In the present work, CFD techniques has been applied to analyze the performance of a turbine that has been designed for small scale air separation [3]. 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.

Design methodology and major dimensional parameters

One dimensional meanline design methodology obtaining the overall geometrical dimensions of the components of the turboexpander is adopted from Kun and Sentz [4] which is based on Balje's n_s-d_s chart [5], empiricisms based on practises for other turbine applications and good engineering judgement. The blade for turbine has been generated following methodology prescribed by Hasselgruber [6] and Balje [5]. Figure 1 provides the major dimensions of different components of the turbine.

Table 1. Turboexpander specifications and boundary conditions for computational analysis

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%

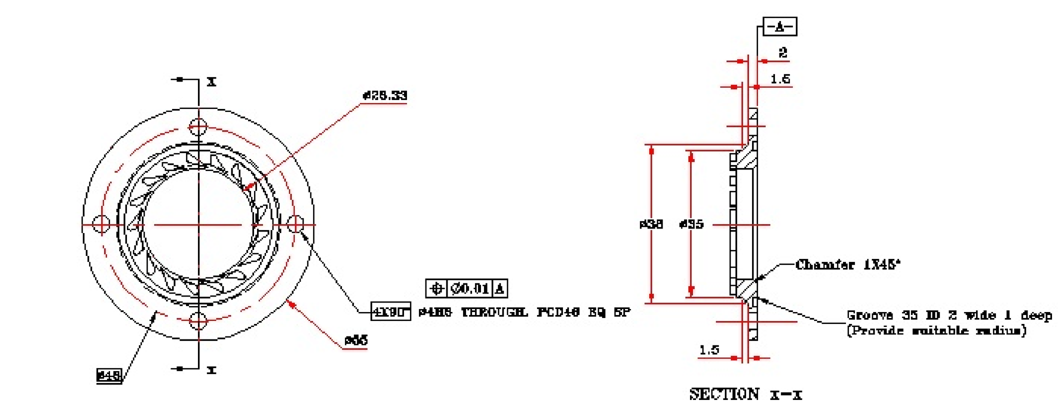

Figure 1. Major dimensions of nozzle

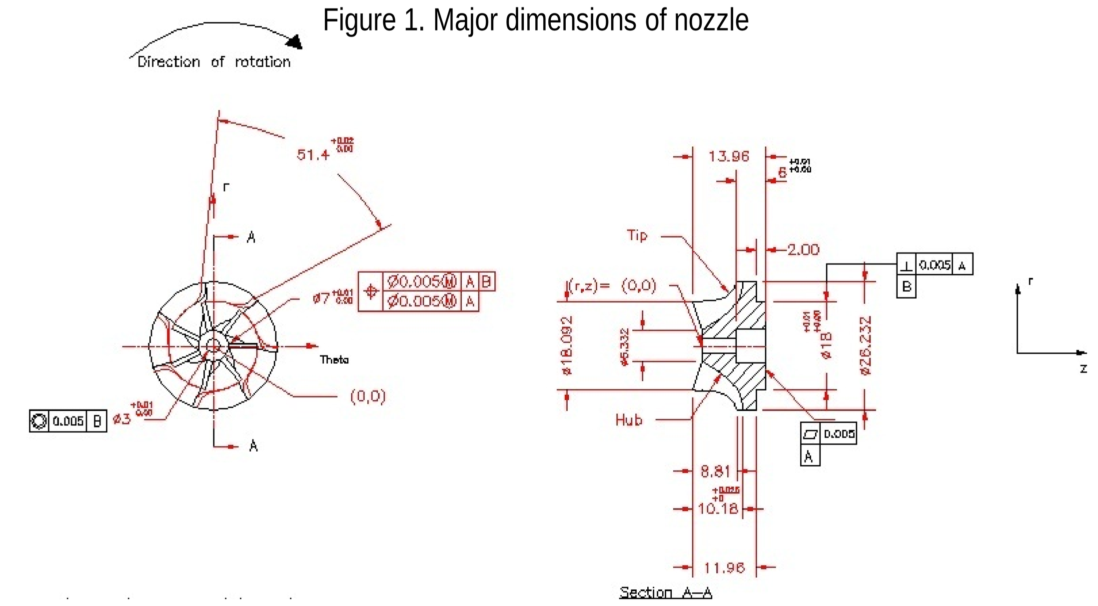

Figure 2. Major dimensions of turbine

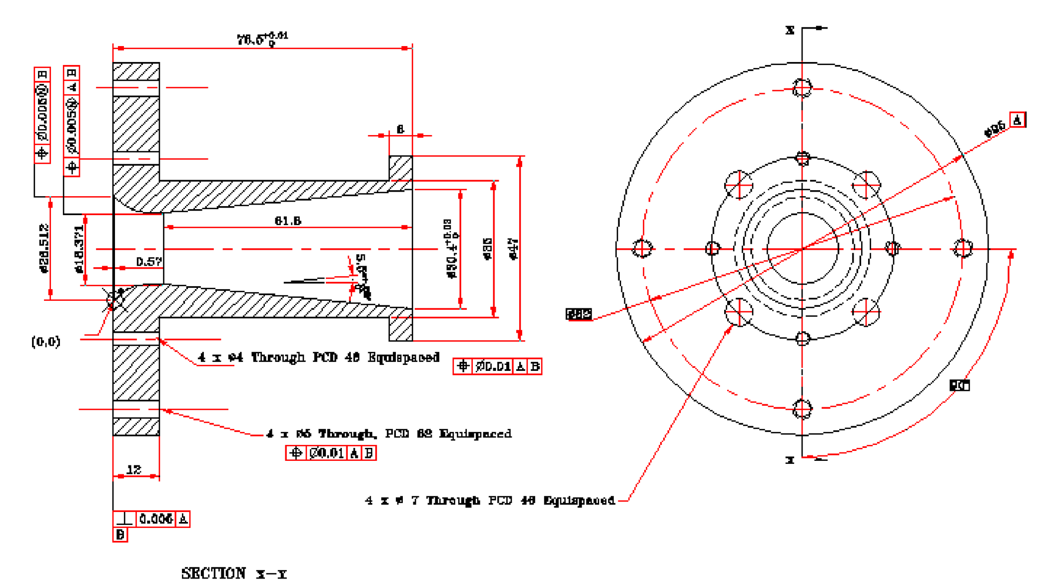

Figure 3. Major dimensions of diffuser

Methodology

Geometry and grid

- ❑ DASSAULT SYSTEMES SolidWorks® was used for creating the fluid domain of nozzle and diffuser section and was then exported to ANSYS DesignModeler®.
- ❑ The turbine blade was created in ANSYS BladeGen® following the design methodology of Hasselgruber and was then meshed using ANSYS TurboGrid®.

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 optimised	Structured (H and O type topology)/ Hexahedral only
Diffuser	39452	210157	Patch conforming method	Unstructured/ Tetrahedral

Boundary conditions

- ❑ Mass flow rate of 0.06 kg/s was specified at the nozzle inlet which is the inlet 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
- ❑ All walls are considered as smooth, no slip and adiabatic
- ❑ Medium turbulent intensity of 5% was assumed for present simulation

Numerical model

- ❑ RANS equation based SST model was used for turbulence modelling.
- ❑ Frozen rotor model was used for modelling the rotor stator interface.
- ❑ Ideal gas equation of state has been used for present simulation.

Convergence criteria

- ❑ The computations were done till the required convergence criteria of 0.0001 (RMS) were satisfied for all the conservation equations.

Results and Discussion

Table 3. comparison of 1D meanline analysis and CFD analysis

Parameters	1D meanline analysis	CFD analysis
Nozzle efficiency	93%	95.6%
Turbine efficiency (total to static)	75%	73%
Power developed	1.73 kW	2.0718 kW
Diffuser pressure recovery factor	0.7	0.5607

Nozzle

- ❑ There is a drop in total pressure as a part of it is used in overcoming the friction.
- ❑ A severe but asymmetric shock wave pattern can be seen at the nozzle discharge in the mach number contour. 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.

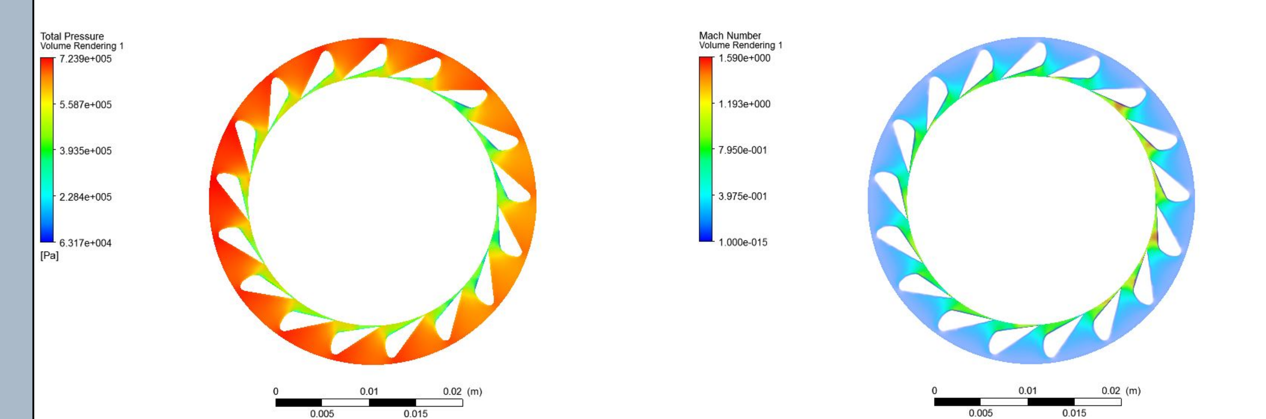

Figure 4. Contours of total pressure and mach number in the nozzle

Turbine

- ❑ Secondary flow is generated within the blade channel from the pressure side

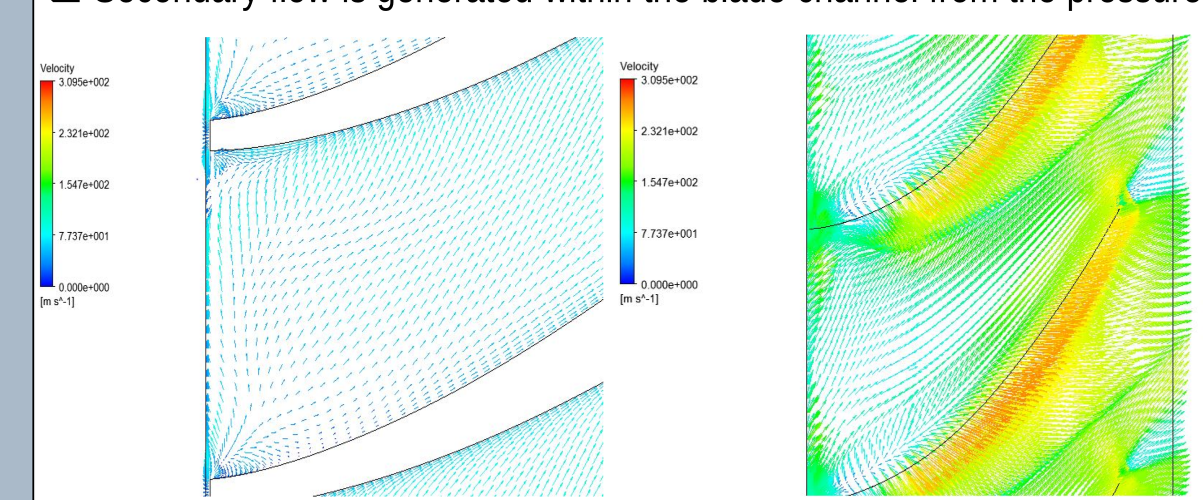

Figure 5. Velocity vector plot in the blade to blade view (a) nearer to the hub (b) nearer to the tip

to the suction side in the end wall boundary layers due to the imbalance between the pressure force and the inertia force within the boundary layer.

- ❑ The sudden expansion from the throat condition leads to flow separation and formation of vortices which in turn results in trailing edge losses [3], which is significant in radial turbines with thick trailing edges.
- ❑ Tip leakage flow over the blade tip from the pressure side to the suction side, results in deteriorating the performance of the turbine [7].
- ❑ Corner stall phenomena at the intersection of suction surface and end wall occurs as a result of the stream wise pressure gradient, secondary flow and mixing of the end wall and blade wall boundary layers [8].

Diffuser

- ❑ Velocity along the walls is almost zero due to friction.
- ❑ Flow separation has occurred at the centre, which can be rectified by change in diffuser angle, length or diffuser type itself (conical or annular).
- ❑ 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.

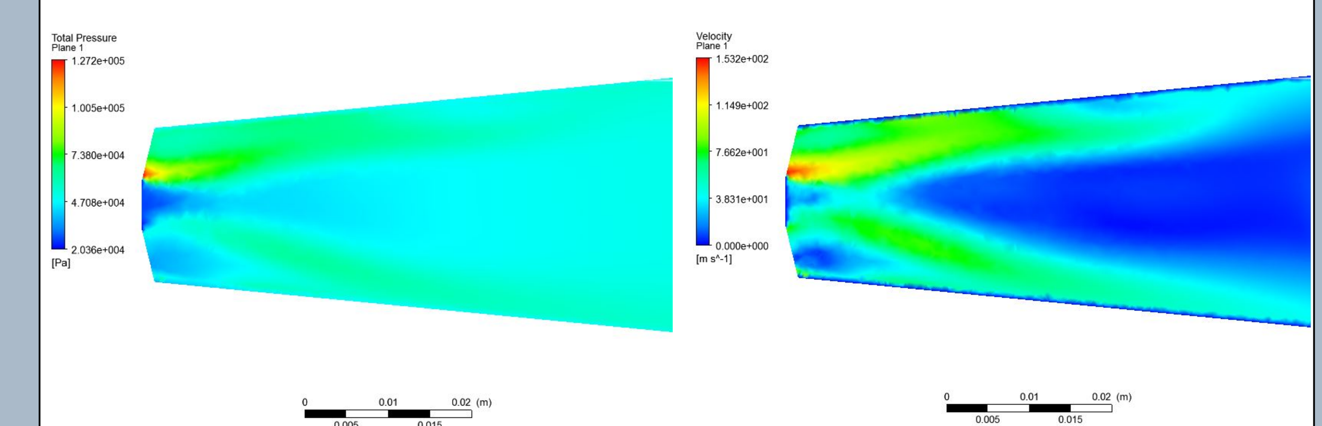

Figure 5. Contours of total pressure and velocity in the diffuser

Conclusions

- ❑ 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
- ❑ Hasselgruber's 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

1. Agahi, R., Ershaghi, B., 1991. Expansion turbine in energy recovery applications. 18th International Congress of Refrigeration.
2. 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.
3. Ghosh, P., 2002. Analytical and Experimental Studies on Cryogenic Turboexpander. Ph.D dissertation, IIT Kharagpur.
4. 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.
5. Balje, O. E., 1981, Turbomachines, John Wiley and Sons.
6. Hasselgruber, H., 1958. Stromungsgerechte gestaltung der laufrader von radialkompressoren mit axialem laufradeintrict Konstruction 10 (1) 22 (in German).
7. Denton, J. D., 1993. Loss mechanisms in turbomachines. Trans ASME J Turbomachinery 115 621-56.
8. Laxminarayana, B., 1996. Fluid Dynamics and Heat Transfer in Turbomachinary, John Wiley.

Designed by: Ashish Alex Sam

ICEC25-ICMC2014: C-03: I.D-36