**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 turboexpanders 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.

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.

**Objective**

- 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.

**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®.
  - DASSAULT SYSTEMES SolidWorks® was used for creating the fluid domain of nozzle and diffuser section and was then exported to ANSYS DesignModeler®.
  - Most of the elements are prismatic with the help of ANSYS meshing tools.
  - Unstructured/wedge elements are also present in the boundary layer.
  - The total number of elements is 154,342, of which 16% are prism and 84% are tetrahedral.

- **Boundary conditions**
  - Mass flow rate of 0.06 kg/s was specified at the nozzle inlet which is the inlet condition.
  - At the diffuser outlet, static pressure of 1.5 bar has been used for specifying outlet boundary condition.
  - Friction flow 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.
  - 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.

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

- **Turbine efficiency**
  - Turbine efficiency (total to static) 75% 73%
  - Nozzle efficiency 93% 95.6%

**Results and Discussion**

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

<table>
<thead>
<tr>
<th>Parameters</th>
<th>1D meanline analysis</th>
<th>CFD analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nozzle efficiency</td>
<td>93%</td>
<td>95.6%</td>
</tr>
<tr>
<td>Turbine efficiency (total to static)</td>
<td>75%</td>
<td>73%</td>
</tr>
<tr>
<td>Power developed</td>
<td>7.73 kW</td>
<td>2.874 kW</td>
</tr>
<tr>
<td>Diffuser pressure recovery factor</td>
<td>0.7</td>
<td>0.567</td>
</tr>
</tbody>
</table>

- 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.

- Table 2. Mesh specifications for various components

<table>
<thead>
<tr>
<th>Domain</th>
<th>Number of nodes</th>
<th>Number of elements</th>
<th>Method type of elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>nozzle</td>
<td>76485</td>
<td>54794</td>
<td>Unstructured viability</td>
</tr>
<tr>
<td>static pressure at exit</td>
<td>1.5 bar</td>
<td></td>
<td>Unstructured viability</td>
</tr>
<tr>
<td>total pressure at inlet</td>
<td>6.0 bar</td>
<td></td>
<td>Unstructured viability</td>
</tr>
<tr>
<td>working fluid</td>
<td>nitrogen</td>
<td></td>
<td>Unstructured viability</td>
</tr>
<tr>
<td>expected efficiency</td>
<td>75%</td>
<td></td>
<td>Unstructured viability</td>
</tr>
</tbody>
</table>

**Conclusions**

- From the above study it has been found that following parameters are important for 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 vorticities.

**References**