Aerodynamic Analysis of High Speed Flow in Turbine Using Fluent

John Maclins.P[1], S. Ruban Viany[2], G.S. Gobu[3], R. Hariharan[4]
[1, 2, 3, 4] Department of Aeronautical Engineering
[1, 2, 3, 4] RVS Educational Trust's Group of Institutions

Abstract- *This paper deals with the three dimensional aerodynamic analysis of high speed flow in turbine. The 3D model and meshing of the turbine is done in GAMBIT2.4 and analyses are to be carried out by FLUENT6.3. The boundary conditions are also defined in GAMBIT and their values are defined in FLUENT. The equations, pressure-velocity coupling, the discretization scheme, Initialization and the solver options are also defined in FLUENT. The model is analyzed for various pressure inlet conditions and the parameters taken into consideration for the flow analyses are temperature, pressure and velocity. The respective contours are drawn and graphed in this project to analyze the flow through the turbine for various cases. The results are compared and inference is that the performance is increased.*

Keywords- high speed flow, turbine, GAMBIT, FLUENT

I. INTRODUCTION

This project deals with the aerodynamic analyses of flow through a cryogenic turbo expander. The main purpose of the project is to improve the performance and to prove that the turbine can be used for refrigeration process and power generation. Nitrogen is used as the refrigerant. The software used is gambit 2.4 and FLUENT 2.3. The turbine model is aerodynamically analyzed with various inlet pressures to determine the velocity, temperature and pressure distribution. GAMBIT 2.4 is used to build the model and mesh it. FLUENT 6.3is used for the flow analysis. For the given inlet conditions, the respective parameters are generated. The pressure, temperature and velocity contours are obtained and the results are plotted. The comparisons for various cases are done and the inference is that the performance is improved.

II. CRYOGENIC TURBINE EXPANDER

A cryogenic turbo expander consists of the following components-

- Nozzle
- Turbine Wheel
- Diffuser
- Shaft
- Brake Compressor
- Radial Bearing
- Thrust Bearing
- Housing

A cryogenic turbo-expander is complex equipment whose design depends on the working fluid, the flow rate and the thermodynamic states at inlet and exit. The high-pressure process gas enters the turbine through piping, into the plenum of the cold end housing and from there, radially into the nozzle ring (1). A tangential velocity is imparted to the fluid, which eventually provides the torque to the rotor. The fluid accelerates through the converging passages of the nozzles. Pressure energy is transformed into kinetic energy, leading to a reduction in static temperature. The high velocity fluid streams impinge on the rotor blades, imparting force to the rotor creating torque. The nozzles and the rotor blades are so aligned as to eliminate sudden changes in flow direction and consequent loss of energy. The turbine wheel is of radial or mixed flow geometry, i.e. the flow enters the wheel radially and exits axially. While larger units are generally shrouded, smaller wheels are open, the turbine housing acting as the shroud. The blade passage has a profile of a three dimensional converging duct, changing from purely radial to an axial tangential direction. Work is extracted as the process gas undergoes expansion with corresponding drop in static temperature. The diffuser is a diverging passage, and acts as a compressor that converts most of the kinetic energy of the gas leaving the rotor to potential energy, in the form of a gain in pressure. Thus the pressure at the outlet of the rotor is lower than the discharge pressure of the turbine system. The expansion ratio in the rotor is thereby increased with a corresponding gain in efficiency and rate of cold production. A loading device is necessary to extract the work output of the turbine. This device, in principle, can be an electrical generator, an eddy current brake, an oil drum, or a centrifugal compressor. The turbine wheel is mounted on a rotating shaft at one end. The torque produced by the expanding gas is transmitted by the shaft to a braking device which can be an oil drum, an electrical generator or a compressor.

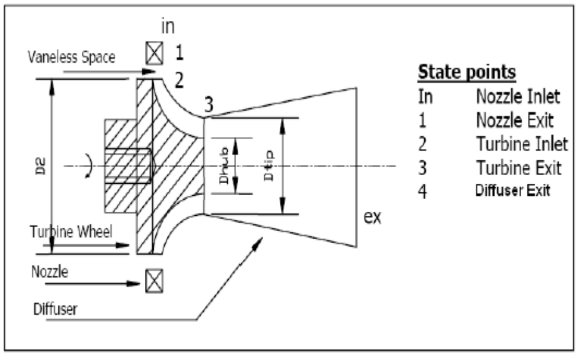

Figure 1.

In the given design, a compressor has been chosen for ease of manufacture and dynamic balancing of the rotor. The shaft is supported on a pair of journal bearings and a pair of thrust bearings, the thrust bearings being placed on opposite sides of a collar built on the shaft. The wheel, the shaft (with the thrust collar built on it) and the brake compressor constitute the rotor. The rotor is surrounded by bearings, turbine and compressor shrouds, and a housing holding them in place. In addition, there are a set of small but critical parts, such as seals, fasteners and spacers.

III. DESIGNING AND MESHING OF TURBINE

A. Modeling of the blade

First the impeller is designed then the hub and the casing were modelled. The casing was modelled by projecting the top phase of the blade. The hub was modelled by projecting the bottom phase of the blade. Casing was designed with zero clearance.

B. Design Considerations

The dimensions used for designing the blade profile are given below:

Table 1. Dimension for Blade Profile

Diameter hub	350 mm
Blade inlet	20 mm
Blade outlet	50 mm
Nozzle outlet	75 mm
Inlet blade angle	20
Outlet blade angle	0
Nozzle height	20 mm
Inner diameter	175 mm

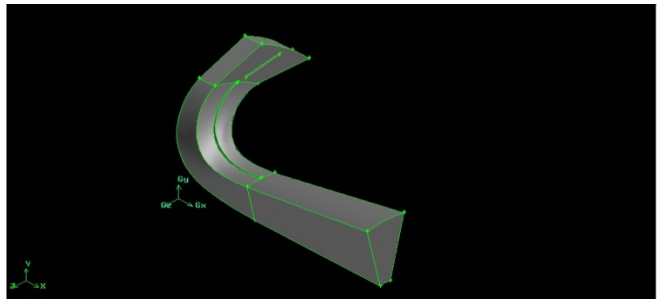

Figure 2. Single flow domain

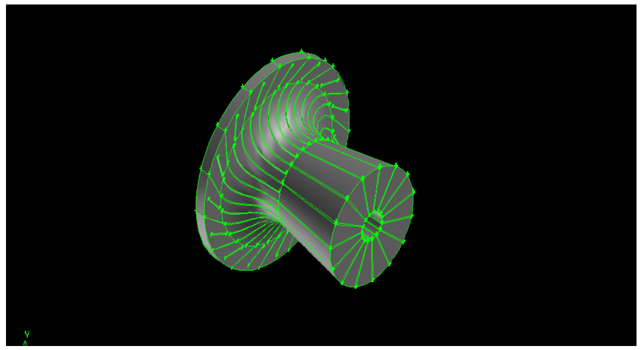

Figure 3. Isometric View

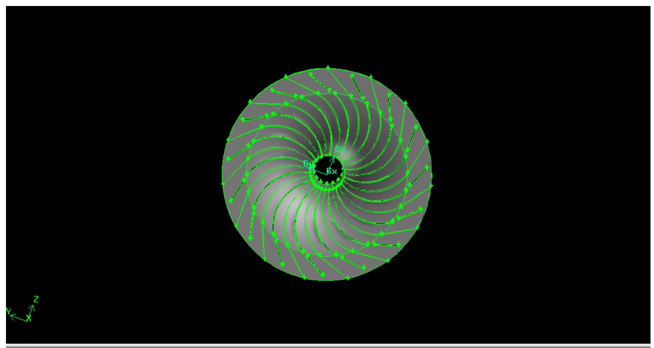

Figure 4. Rear View

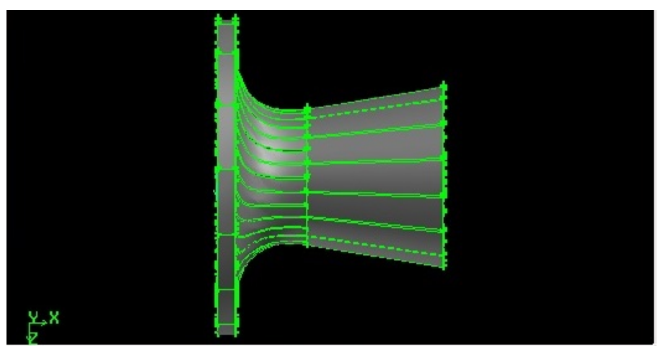

Figure 5. Side View

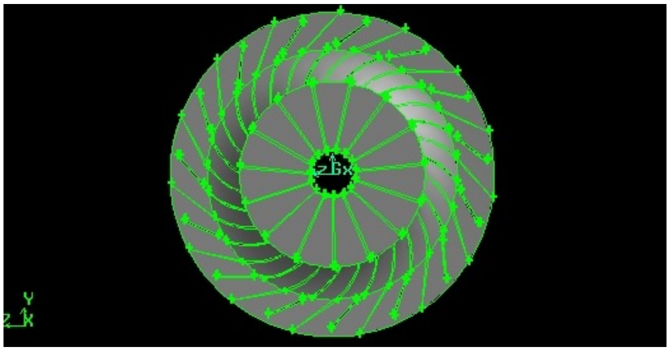

Figure 6. Front View

C. Meshing of the model

Meshing is the process of reducing the given models into elements and nodes so that the given problem could be solved with high accuracy

- Faces to be used for periodicity has to be linked first using ‘link/unlink faces command’
- The faces are meshed using ‘tri’ elements, meshing type is given as ‘pave’
- After all the faces were meshed, the volume is meshed with ‘tri/hybrid’ element and meshing type is given as ‘T-grid’.
- Number of nodes and elements for single profile is given below

Table 2. Meshing

Number of elements	*162204*
Number of nodes	*33310*

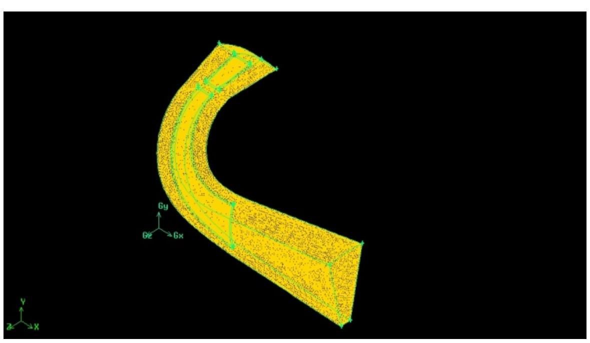

Figure 7. Meshed Single Blade Profile

D. Boundary Conditions

The boundary conditions can be defined as the conditions that are applied to the faces that affect the flow. They are the set of conditions specified for the behaviour of the solution to a set of differential equations at the boundary of its domain. Boundary conditions are important in determining the mathematical solutions to many physical problems .The boundary conditions that are applied in the model are listed below.

- Pressure inlet
- Pressure outlet
- Wall
- Periodic

The boundary conditions are defined as given below

Table 3. Boundary Conditions for Respective Faces

Inlet	Pressure Inlet
Nozzle Outlet	Pressure Outlet
Blade	Wall
Casing	Wall
Stator	Wall

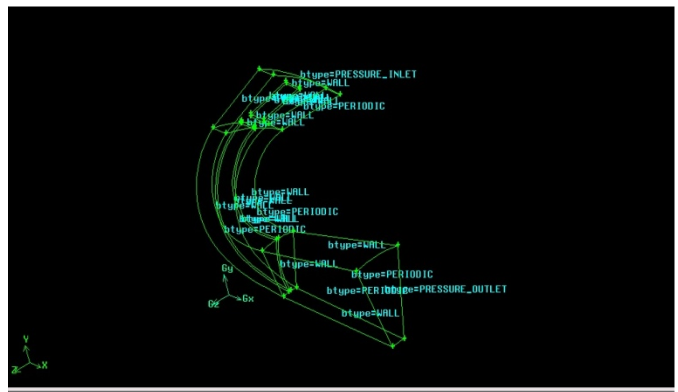

Figure 8. Boundary Conditions with Label

The respective boundary condition provided for the designed model is shown below

Table 4. Colours Representing Boundaries In Single Blade Profile

Red	Pressure Outlet
Cyan	Periodicity
White	Wall
Sky blue	Pressure Inlet

IV ANALYSIS USING FLUENT

A. Checking of Mesh and Scaling

The procedure for checking the mesh and scaling is given below

- The fluent solver is opened where 3D is selected
- Then the importing of the meshed file is done.
- The checking of the imported file is done by selecting the option ‘grid-check’.

- The meshed file then undergoes a checking where number of grids and number of nodes is found. After this grid check is done.
- Following this the scaling is done. Scale is scaled to mm.

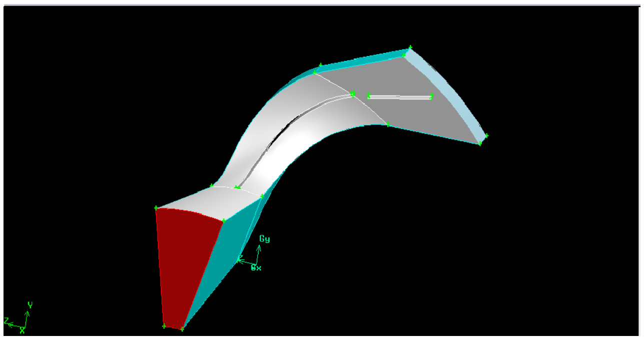

Figure 9. Boundary Conditions on Single Blade Profile

B. solver, material condition and operating condition

- The solver is defined first.
- Solver is taken as pressure based and formulation as implicit, space as 3D and time as steady.
- Velocity formulation as absolute and gradient options as Green-Gauss Cell based are taken.
- Energy equation is taken into consideration.
- The viscous medium is also taken.
- The analysis is carried using k-epsilon.
- 2 results are to be found out with different inlet pressures.
- The selection of material is done.
- Material selected is nitrogen gas.
- The analysis is carried out under operating conditions of 101325 Pascal.
- Gravity is not taken into consideration.

C. Boundary conditions

Nozzle inlet

Pressure inlet was taken for the nozzle inlet and the value of gauge total pressure is taken as 200000 Pascal for case (1) and the gauge total pressure was taken as 500000 Pascal for case (2). Initial gauge pressure was taken as 101325 Pascal. Temperature was taken as 120K.

Casing

Casing is chosen to be wall and it is set as stationary wall. No slip shear condition is given. Thermal conditions for heat flux as 0 and the heat generated is given to be 0. The Temperature of wall is given as 300k. In Convection free stream temperature is given to be 300k. The solid material name is also specified here. Under radiation, External Emissivity is given to be 1. External Radiation Temperature is given as 300k.

Stator

Stator is defined as wall and it is set as stationary wall. No slip shear condition is given. Thermal conditions for heat flux as 0 and the heat generated is given to be 0. The Temperature of wall is given as 300k. In Convection free stream temperature is given to be 300k. The solid material name is also specified here. Under radiation, External Emissivity is given to be 1. External Radiation Temperature is given as 300k.

Blades and Flow Path

Blade is defined to be wall and it is set as stationary wall. No slip shear condition is given. Under wall roughness, Roughness Height is given to be 0 and Roughness Constant is given to be 0.5. Thermal conditions for heat flux as 0 and the heat generated is given to be 0. The Temperature of wall is given as 300k. In Convection free stream temperature is given to be 300k. The solid material name is also specified here. Under radiation, External Emissivity is given to be 1. External Radiation Temperature is given as 300k.The meshed region provides the flow path. The fluid is selected to be nitrogen

Outlet

The diffuser was set as pressure outlet and the pressure was set to 101325 Pascal. Back flow direction specification method is given as Normal to Boundary. Under Turbulence Specification method is given as k & e. The Backflow Turbulent Kinetic Energy is given as 0.2 (m2/s2) and the Backflow Turbulent Dissipation Rate is given as 0.01(m2/s3). Under Thermal, Backflow Total Temperature is 300k.Here turbulence energy and dissipation rate are taken into consideration.

Periodic Conditions

Single blade is provided periodicity. Periodic type is give as rotational. Number of blades to be applied is given as 15 and the rotation angle at which it is needed is given as 24°. Type is given as specify pressure gradient, the flow direction as (1, 0, 0) and the upstream bulk temperature is given to be 300k. The relaxation factor is 0.5.

Initialization

Solution initialization is done. Initial values of velocity are taken as zero along x, y and z direction. Temperature is taken as 300K.

Residual Monitorization is done and convergence criteria are set up. The convergence criteria of various parameters are listed below.
Continuity- 0.001
X-Velocity- 0.001
Y-Velocity- 0.001
Z-Velocity- 0.001
Energy- 1e-06

The number of iterations is then set up and iterations starts. The iteration continues till the convergence is reached.

D. Solution Limits

Table 5. Control Limits

QUANTITY	LIMIT
Minimum Absolute Pressure	1e-10Pa
Maximum Absolute Pressure	5e+10Pa
Minimum Temperature	1e-10K
Maximum Temperature	5e+10K
Minimum Turb. Kinetic Energy	1e-14
Maximum Turb. Viscosity Ratio	1e-20
Minimum Turb. Dissipation Rate	1e+10

E. Material Properties

FLUID: NITROGEN

Table 6. Control Limits

PROPERTY	UNITS	VALUES
Density	kg/m3	1.138
Cp(specific heat)	j/kg-k	1040.67
Thermal Conductivity	w/m-k	0.0242
Viscosity	kg/kgmol	28.0134
Energy parameter	K	97.53

SOLID: ALUMINIUM

Table 7. Material Properties- Solid

PROPERTY	UNITS	VALUES
Density	kg/m3	2719
Cp (specific heat)	J/kg-K	871
Thermal Conductivity	W/m-K	202.4

F. Viscous Model

Under Model Condition, k-epsilon is selected. In k-epsilon model standard is selected. In Near-Wall Treatment Standard Wall Functions are defined. Viscous Heating is selected.

V. RESULTS

A. Case I – Inlet Pressure 200000

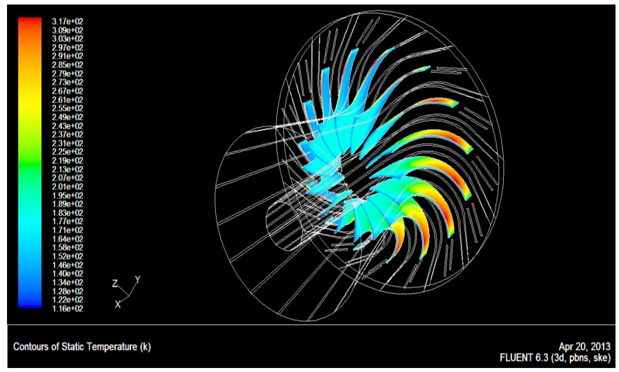

Figure 10. Static Temperature Contour on Blade

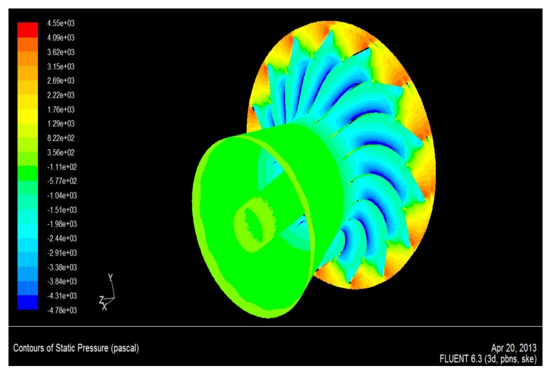

Figure 11. Static Pressure Contour on Casing and Hub

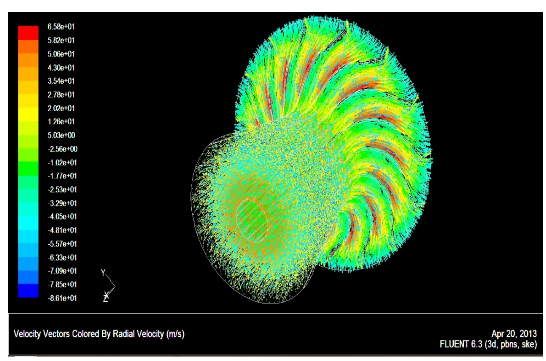

Figure 12. Radial Velocity Vectors through Default Interior

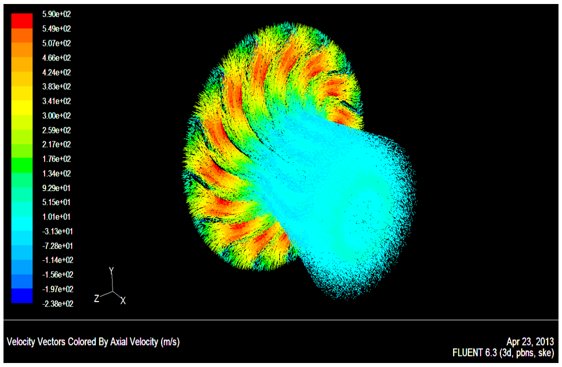

Figure 13. Axial Velocity Vectors through Default Interior

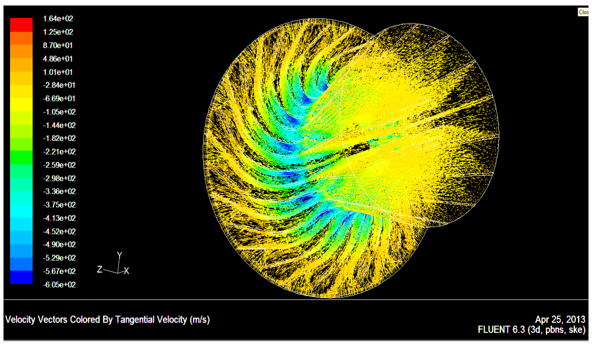

Figure 14. Tangential Velocity Vectors through Default Interior

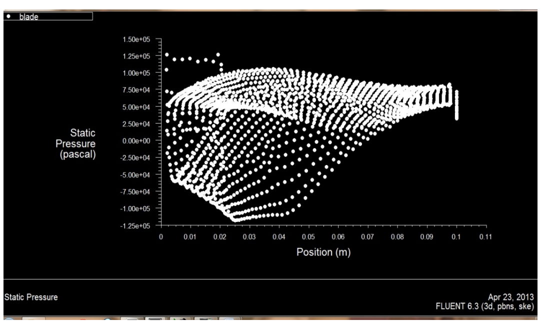

Figure 15. Static Pressure Plot of Blade

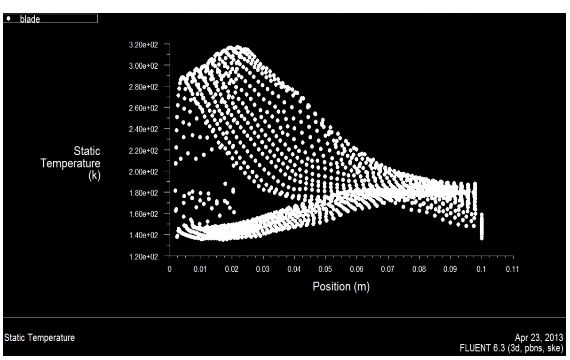

Figure 16. Static Temperature Plot of Blade

B. Case II – Inlet Pressure 500000

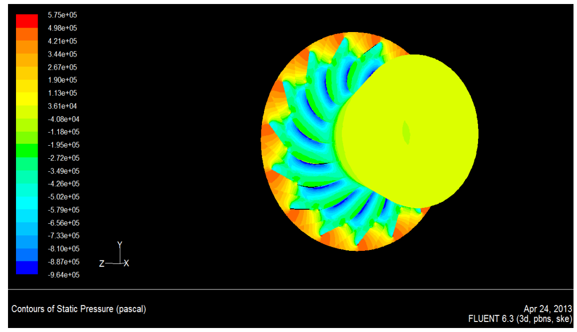

Figure 17. Static Pressure Contours on Casing and Hub

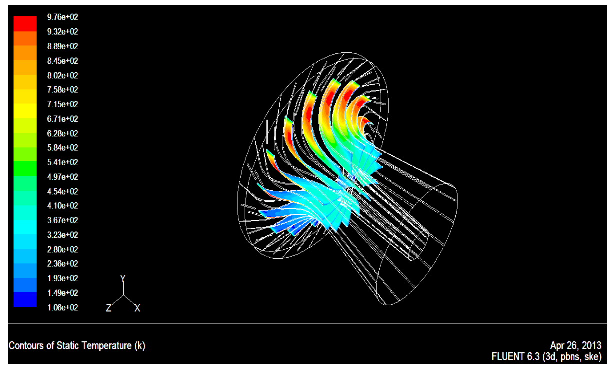

Figure 18. Static Temperature Contours on Blade

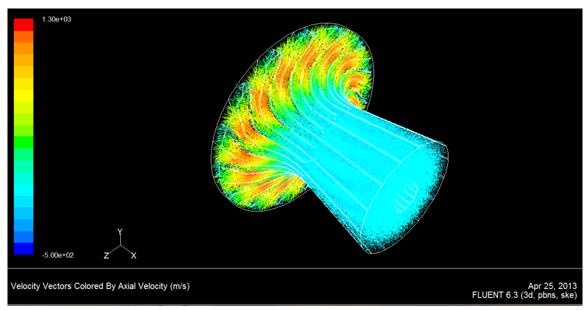

Figure 19. Axial Velocity Vector through Default Interior

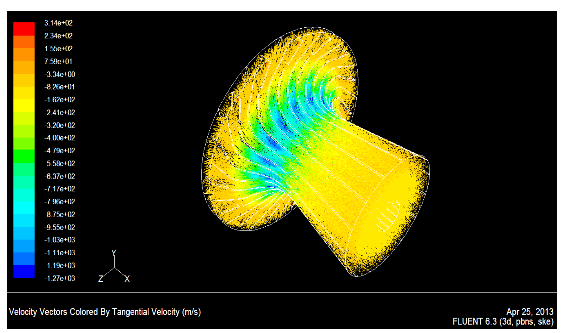

Figure 20. Tangential Velocity Vector through Default Interior

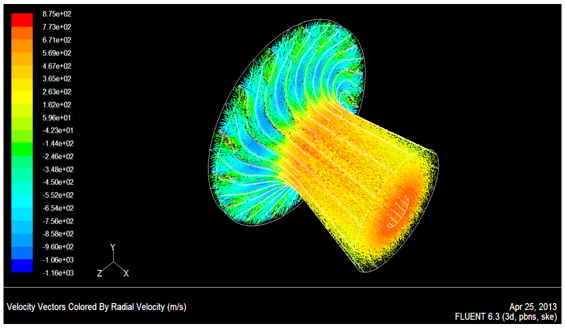

Figure 21. Radial Velocity Vector through Default Interior

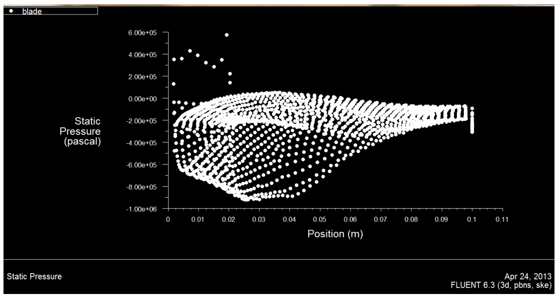

Figure 22. Static Pressure Plot

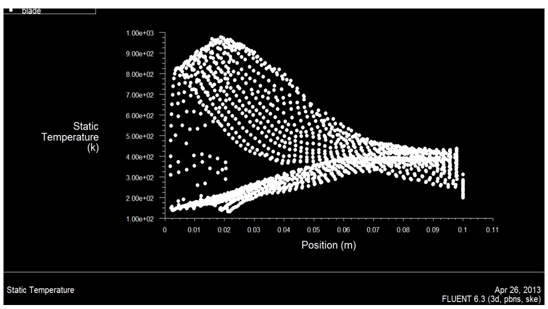

Figure 23. Static Temperature Plot

VI. CONCLUSION

The turbines have great utility in industries. The turbine is analysed for various inlet pressures. The contours and graphs indicating the variation of temperature, pressure and velocity along various regions of turbine is given for different inlet conditions. These results are then compared. From the comparison it can be seen that the high pressure increases the expansion of the turbine. So the performance of the turbine is increased. And as nitrogen is used as a refrigerant, low temperature can also be maintained at high pressure. So it can be also used for the refrigeration purpose. Thus the aerodynamic analysis of the high speed turbine is done

REFERENCES

[1] John.J Adamczyk, Aerodynamic Analysis of Multistage Turbomachinery Flows in Support of Aerodynamic Design, Journal of Turbomachinery, vol 122, issue 2, pp 189-217

[2] Yutaka MIYAKE, Tkuji TSUGAWA, Susumu MURATA, An Analysis of High-speed Type Cross flow Turbines, Bulletin of JSME, vol 26, 1983, pp 43-50

[3] Tao Ning, Chun-Wei Gu, Wei-Dou Ni, Xiao-Tang Li and Tai-Qiu LiuH, Aerodynamic Analysis and Three-Dimensional Redesign of a Multi-Stage Axial Flow Compressor, mdpi journals,2016,pp 1-16

[4] Denton, J.D. Throughflow calculations for transonic axial flow turbines. J. Eng. Power 1978, 100, pp 212–218

[5] Adamczyk, J.J.; Celestina, M.L.; Beach, T.A.; Barnett, M. Simulation of three-dimensional viscous flow within a multistage turbine. ASME J. Turbomach. 1990, 112, pp 370–376.

[6] Ghosh. P, Sarangi. S, Thesis on analytical and experimental studies on Turbo expander, IIT Kharagpur, 2002