Computational Fluid Flow Analysis Of High Speed Rotating Turbine With Blade Angle 15^0

Manish vyas
Assistant professor, Department of Mechanical Engineering,
KG Reddy College of Engineering and Technology, Moinabad, India.

Abstract: An attempt is made to obtain the minimum temperature and pressure and to study the variation of velocity and entropy. In our present research a model of high speed rotating turbine is created with blade angle 15^0 and later which is simulated for results by using two different fluids i.e., oxygen and nitrogen. Here, creo element/pro-E is used to create the model of turbine using available data of hub, shroud and blade profile. ANSYS FLUENT 14.5 is used to mesh the model, to specify the simulation settings and physical parameters required to describe the flow through turbo expander at inlet and to examining and analyzing the output results. Using these results we can determine that how the variation will occur by using various fluids. Various graphs are plotted indicating the variation of velocity, entropy and mass flow rate to analyze the flow through Expansion turbine.

Index Terms: **Radial turbine, Blade angle 15^0, Pro-e, Ansys Fluent 14.5.**

I. Introduction:

In present research a rotating turbine is used as a turbo expander which is a pressure let-down device that produces low temperatures for the formation of cryogenics liquids this are majorly used in all areas of the gas and oil industries to produce cryogenic refrigeration. Liquefaction of gases is generally used in large scale transportation, storage and low temperature applications. For the production of cryogenic liquids like nitrogen, oxygen and argon in a large scale, cryogenic turbo expander provides the most prominent economical route. Cryogenic turbo expander has the high thermal efficiency and thermal reliability when compared to the high and medium pressure systems.

1.1 Anatomy of a Cryogenic Turbo expander:

Cryogenic turbo expander consists of a compressor, rotating turbine, shaft, nozzle, etc. here; rotating turbine and compressor are connected through a single shaft which is supported by the required number of journal and thrust bearings. These components are placed in an appropriate housing, which also consist the ducts for inlet and outlet of fluid. Most of these plants are installed in the vertical orientation for easy maintenance of the rotors which are of small and medium size. The compressed fluid from the compressor accelerates through the converging passages of the nozzles. Then this high-pressurized gas enters into the rotating turbine through a pipe from the cold end housing here, in the turbine, fluid Pressure energy is transformed into kinetic energy which make static temperature to decrease. The rotor blades are designed to eliminate sudden changes in flow and to consequent loss of energy. The geometry of turbine wheel is in form of radial or mixed flow because of which fluid enters radial and exit axially. The fluid gas undergoes expansion through which work is extracted inform of drop in static temperature.

2. Literature Review:

The major component of any cryogenic plant is turbine or the turbo expander. As the major role of turbo expander is to phase change from gas to cryogenics liquids by decreasing static temperature below the refrigeration point. This concept was first introduced by Lord Rayleigh in June 1898. He discussed regarding use of turbine as a expansion device in place of other expansion device. In 1898, a British engineer named Edgar C. Thrupppatenteda liquefying machine using an expansion turbine. A simple method sufficient for the design of a high efficiency expansion turbine is outlined by Kun et. al [1-2].
Agahiet. al. [3-4] have explained the design process of the turboexpander utilizing modern technology, such as CFD software, Computer Numerical Control Technology and Holographic Techniques to further improve anal ready impressive turboexpander efficiency performance. Several characteristics values are used for defining significant performance criteria of turbo machines such as turbine velocity ratio, pressure ratio, flow coefficient factor and specific speed [5]. Balje has presented a simplified method for computing the efficiency of radial turbo machines and for calculating their characteristics [6].The concept of specific speed was first introduced for classifying hydraulic machines. Balje [7] introduced this parameter in design of gas turbines and compressors. PhD dissertation of Ghosh [8] explains the detailed summary of technical features and experimental analysis of cryogenic turboexpander. S.K. Ghosh, R.K. Sahoo, S.K. Sarangi in 2005 gave a computational approach to the design of a cryogenic turbine blade profile[9].

3. COMPUTATIONAL FLUID FLOW ANALYSIS:

The computational analysis of turbine is majorly done in following three steps. creo element/pro-E is used to create the model of turbine using available data of hub, shroud and blade profile. ANSYS FLUENT 14.5 is used to mesh the model, to specify the simulation settings and physical parameters required to describe the flow through turbo expander at inlet and to examining and analyzing the output results. The present design procedure is available in literature [8].

4. Designing of Turbine Model:

Designing of model is done in Pro-E by using data available for blade profile, hub, and shroud. The hub and the tip streamlines are taken from the literature [8, 9]. The surface of turbine model which had generated is considered as the mean surface within a blade. The blade faces will be merged where they are tangent to one another. Here while developing the model there is a need to create a stage fluid zone body for the flow passage, Create fluid zone property is selected and an enclosure feature to subtract the blade body. This resulting enclosure can be used for a CFD analysis of the blade passage. Create all blades this property is used to create All the blades are created using create all blades property by specifying the number of blades in the Pro-E model. Here we are using ten numbers of blades.
In summary, the major dimensions for prototype turbine have been computed by S.K. Ghosh[8] as follows:

Rotational speed: N = 22910rad/sec = 218790rpm

Wheel diameter: D2 = 16.0 mm

Eye tip diameter: Dtip = 10.8mm

Eye hub diameter: Dhub = 4.6mm

Numbers of blades: Ztr = 10

Thickness of blades: ttr = 0.6mm

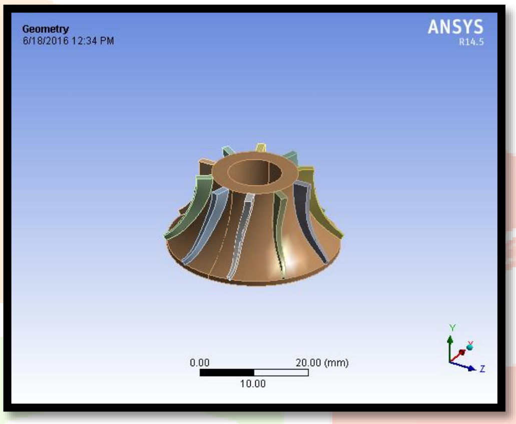


Figure4.1 Sample Model designed with using blade angle **15^0**

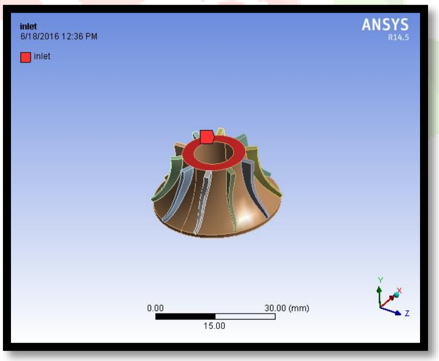


Figure4.2 Inlet fluid passage

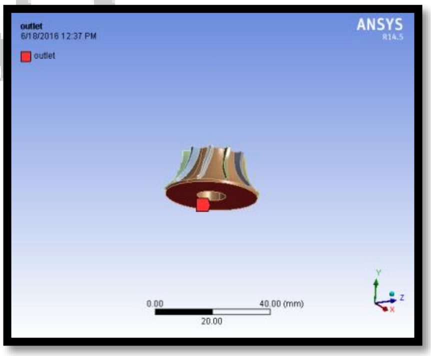


Figure4.3 Outlet fluid passage

5. Material for Turbine Wheel:

Material selection and manufacturing of the turbine wheel are critical issues because of the thin blading, surface finish, and tolerance requirements. Aluminium is the ideal material for turbine impellers or blades because of its excellent low temperature properties, high strength to weight ratio and adaptability to various fabricating techniques . Aluminium alloy with the following composition is usually selected for turbine wheel.

Alloying elements	%
Copper	0.10
Iron	0.6
Magnesium	0.4 to 1.4
Manganese	0.4 to 1.0
Chromium	0.3
Silicon	0.6 to 1.3
aluminium	rest

Table 5.1 Material for turbine wheel

6. MESHING OF TURBINE MODEL:

Ansys fluent 14.5 is used for meshing of turbine model. High quality hexahedral meshes are created in rotating turbine rotor for fluid flow analysis. Geometry information regarding rotating turbine rotor is imported from Pro-e in .iges format. Ansys uses this pro-e file to the length unit, number of blades, axis of rotation and other parameters.

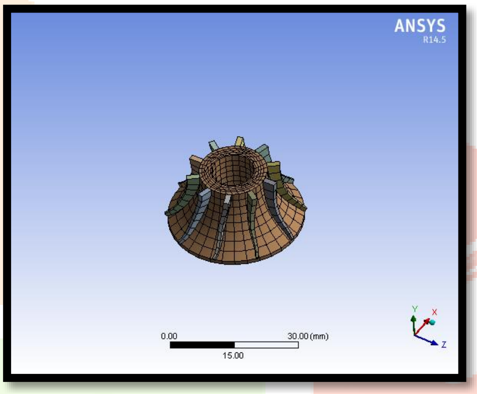


Figure 6.1 Meshed turbine model.

7. Input conditions for rotating turbine simulation:

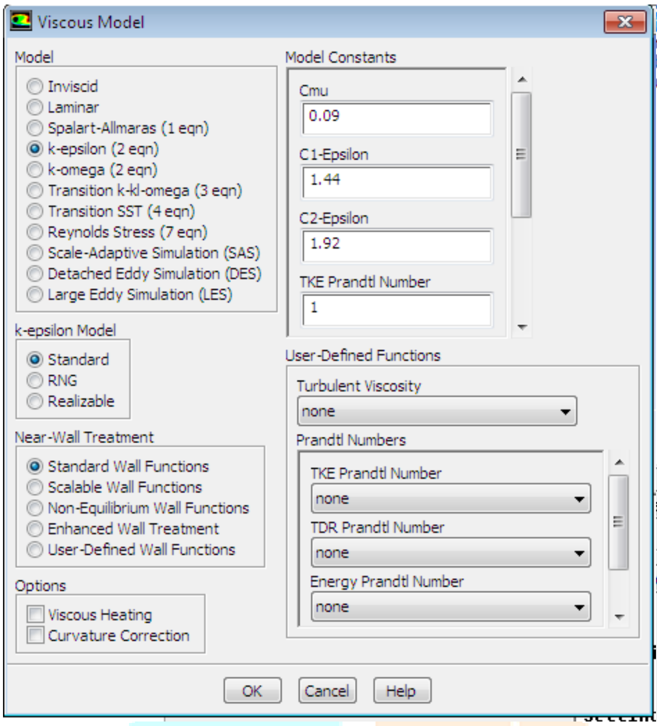
CFD simulatation for rotating turbine is done in Ansys fluent 14.5.In this the model is taken from pro-e which is converted from 2D to 3D and the later is it meshed by fixing the nodes, in software general conditions are provide for pressure based and energy equation with fluid density and viscosity. In following figure we can see the boundary conditions and fluid condition while inlet of the rotating turbine.

Mass flow rate=0.024 kg/sec

Temperature: 120 K

Pressure= 30 bar

Here we are giving the no .of iteration is 20.



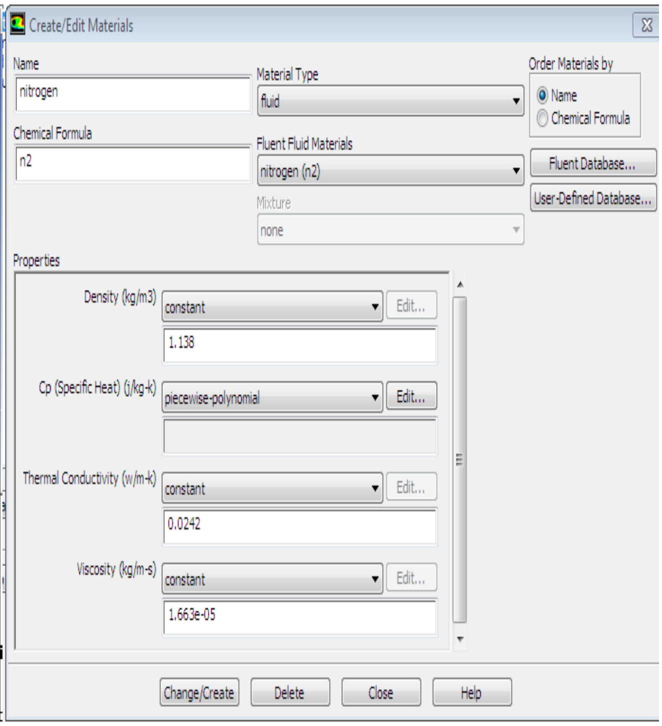


Figure 7.1Inlet condition for turbine in Ansys

Figure 7.2 Fluid conditions for nitrogen

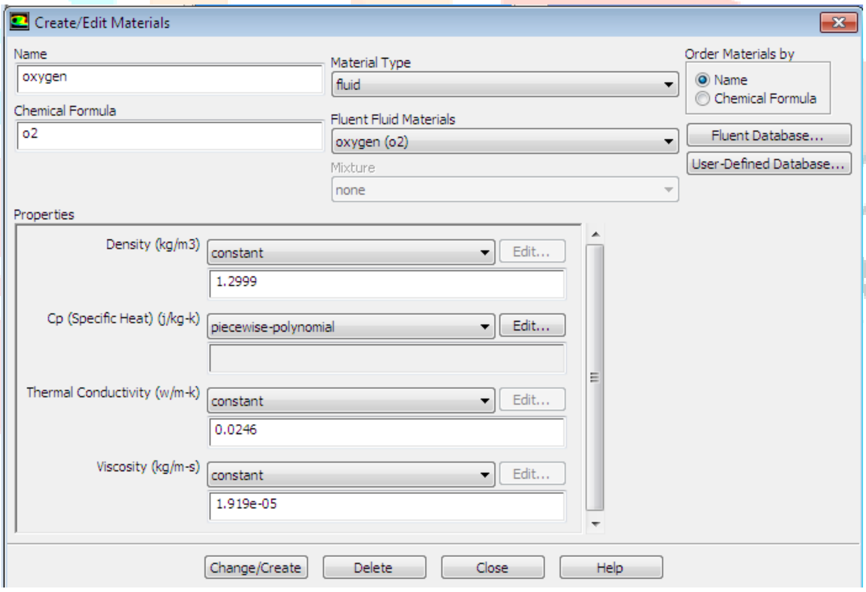


Figure 7.3 Fluid conditions for oxygen

8. Obtained Results:

Ansys fluent 14.5 is used to allow easy visualization and quantitative analysis of results of CFD simulations. It can create user defined scalar and vector variables it also includes automatic report.

- WITH BLADE ANGLE 15^0
- WORKING FLUID = NITROGEN

8.1.1 PRESSURE

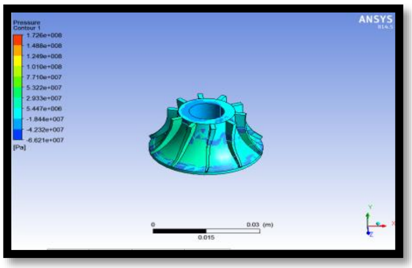


Figure 8.1.1 Result of pressure flow inside the turbine

8.1.2 VELOCITY

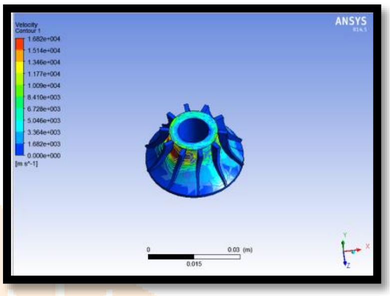


Figure 8.2.1 Results of velocity flow inside the turbine

8.1.3 TEMPERATURE

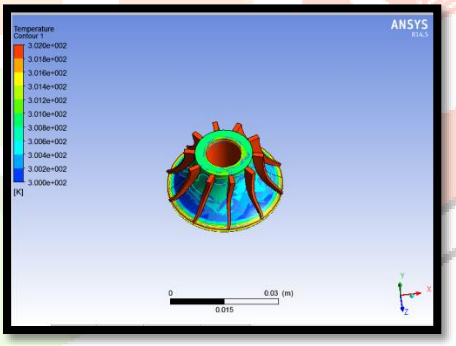


Figure 8.13 Results of Temperature flow inside the turbine

8.1.4 ENTROPY

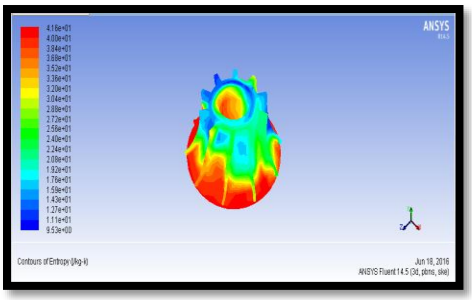


Figure 8.1.4 Results of Entropy flow inside the turbine

- WITH BLADE ANGLE= 15^0
- WORKING FLUID = OXYGEN

8.2.1 PRESSURE

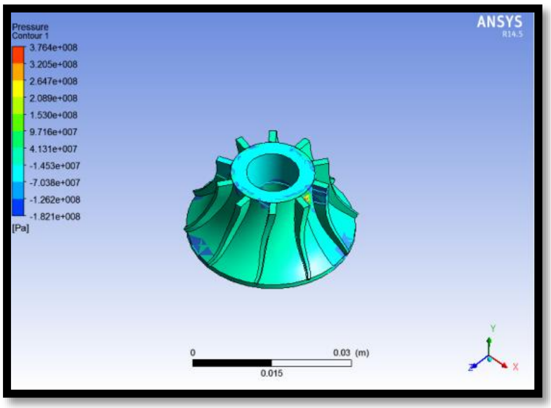


Figure 8.2.1 Results of pressure flow inside the turbine

8.2.2 VELOCITY

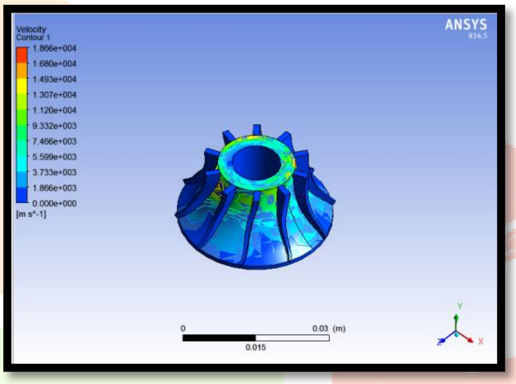


Figure 8.2.2 Results of velocity flow inside the turbine

8.2.3 TEMPERATURE

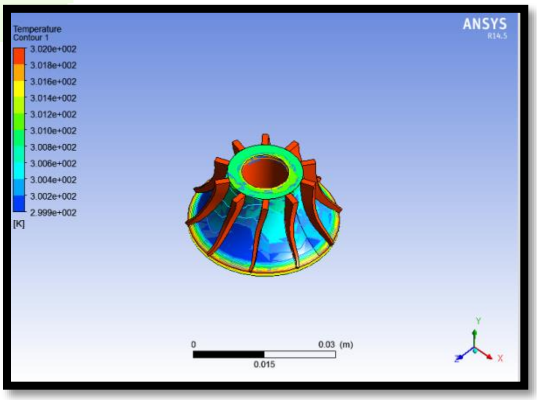
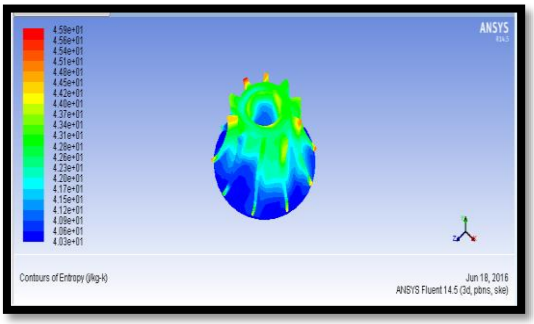


Figure 8.2.3 Results of Temperature flow inside the turbine

8.2.4 ENTROPY



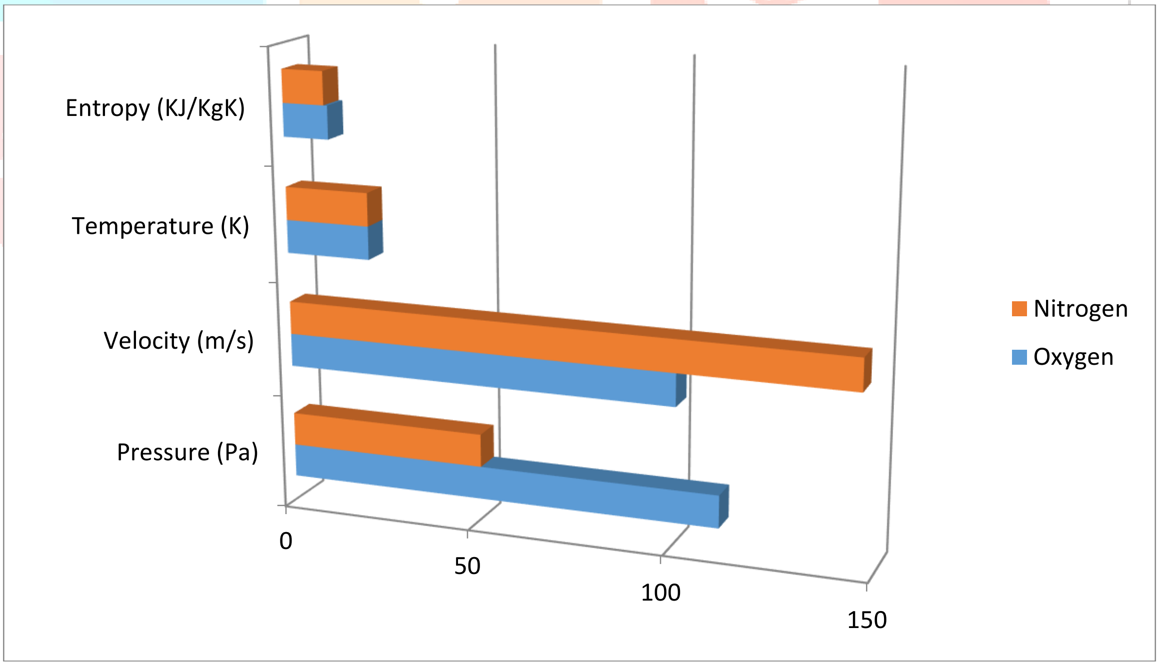
8.2.4 Results of Entropy flow inside the turbine

9. Results and Discussions:

Graph and table are available from generated results.
In this the input parameters for nitrogen and oxygen fluid provided were same. the input pressure was 30bar but if we see the variation in output pressures, for oxygen fluid output pressure is directly double from the nitrogen fluid. Same in the case of output velocities in some extent we can say.

Properties at blade angle 15	Pressure (pa)	Velocity(m/s)	Temperature (k)	Entropy(kj/kgk)
Nitrogen	51	101	22	12
Oxygen	112	146	22	11

9.1 table for output resluts



9.2 Graph shows variation in output results for nitrogen and oxygen.

10. Conculsion:

This work is a modest attempt at flow analyzing inside a cryogenic rotating turbine with blade angle 15^0 . A model of rotating turbine expander has been designed, meshed and simulated. The design procedure covers the designing of turbine in pro-E. ANSYS FLUENT 14.5 is used to mesh the model, to specify the simulation settings and physical parameters required to describe the flow through turbine at inlet and to examining and analyzing the output results. Graph indicating the variations of mass flow rate, pressure, velocity and entropy inside the turbine. There is a lot of variation in results when fluid is change, for this again the redesigning of turbine model is necessary by changing its blade angle in future work the remodelling and analysing of rotating turbine can be done by using different blade angle.

11. References:

[1] L.C. Kun, T.C. Hanson, High efficiency turboexpander in aN2 liquefier AICHE Spring meeting, Houston, Texas (1985).
[2] L. C. Kun, Expansion turbines and refrigeration for gas separation and liquefaction Advances in Cryogenic Engineering (1987), V33, 963-973
[3] R. R. Aghai, M. C. Lin, B. Ershaghi, High Performance cryogenic turboexpanders Advances in Cryogenic Engineering (1996), V41, 941-947
[4] R. R. Aghai, M. C. Lin, B. Ershaghi, Improvements of the efficiency of the turboexpanders in cryogenic applications Advances in Cryogenic Engineering (1996), V41, 933-940
[5] Von der Nuell , W. T. Single - stage radial turbine for gaseous substances with high rotative and low specific speed Trans ASME (1952), V74, 499-51
[6] O. E. Balje, A contribution to the problem of designing radial turbomachines Trans ASME (1952), V74, 451-472
[7] O. E. Balje, A study on design criteria and matching of turbomachines: Part-A—similarity relations and design criteria of turbines Trans ASME J Eng Power (1972), 83-101
[8] Ghosh, S.k. "Experimental and Computational Studies on Cryogenic Turboexpander" Ph.D dissertation, NIT Rourkela.
[9] Ghosh, S.K., Seshaiah, N., Sahoo, R.K., Sarangi, S.K. Design of Turboexpander for Cryogenic applications, Indian Journalof Cryogenics, Special Issue - Vol.2, 75-81
[10] Computational Fluid Flow Analysis of High Speed Cryogenic Turbine Using CFXSushant Upadhyay1, Shreya Srivastava2, Siddharth Sagar3, Surabhi Singh4 and Hitesh Dimri5 Asian Journal of Engineering and Applied TechnologyISSN: 2249-068X Vol. 4 No. 1, 2015, pp.44-52