COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF FLOW THROUGH HIGH SPEED TURBINE USING FLUENT

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE

REQUIREMENTS FOR THE DEGREE OF

Bachelor of Technology

In

Mechanical Engineering

BY

BIDHAN KUMAR PRADHAN

Under the Guidance of

PROF. SUNIL KUMAR SARANGI

NATIONAL INSTITUTE OF TECHNOLOGY
ROURKELA

Department of Mechanical Engineering

National Institute of Technology

Rourkela

National Institute of Technology

Rourkela

CERTIFICATE

This is to certify that the thesis entitled "**CFD ANALYSIS OF FLOW THROUGH HIGH SPEED TURBINE USING FLUENT**" submitted by **Bidhan Kumar Pradhan** in partial fulfillment of the requirements for the award of Bachelor of technology Degree in Mechanical Engineering at the National Institute of Technology, Rourkela (Deemed University) is an authentic work carried out by him under my supervision and guidance.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other University / Institute for the award of any Degree or Diploma.

Prof. Sunil Kumar. Sarangi

Date

National Institute of Technology

Rourkela-769008

ACKNOWLEDGEMENT

I would like to express my deep sense of gratitude and respect to my mentor Prof. Sunil Kumar Sarangi, for his excellent guidance and suggestions. He has been a motivating factor for me. I feel myself extremely lucky to get a chance to work under the guidance of such a dynamic personality.

I am thankful to Prof P. Rath for building up my concepts in Computational Fluid Dynamics.

Besides I would like to thank Mr. Banjara and Mr. Subrato Ghosh for their help.

Bidhan Kumar Pradhan

Roll No- 10503054

8^{th} Semester B. TECH

Mechanical Engineering Department

National Institute of Technology, Rourkela

LIST OF CONTENTS

LIST OF FIGURES

<u>**LIST OF TABLES**</u>

ABSTRACT

This project deals with the computational fluid dynamics analysis of flow in high speed turbine. This involves with the three dimensional analysis of flow through of a high turbine having radial inlet and axial outlet. The software used for this purpose are GAMBIT and FLUENT. The 3 D model of the parts of the turbine are made by GAMBIT and analysis are to be carried out by FLUENT. The models are first generated using the data and then are meshed and then various velocity and pressure contours are to be drawn and graphed in this paper to analyze the flow through the cryogenic turbine. Various graphs indicating the variation of velocity, pressure and temperature along the stream length of the turbine are given.

Keywords: Radial Inlet, Axial Outlet, Gambit, Fluent, Stream length

CHAPTER 1

INTRODUCTION

INTRODUCTION-

Oxygen, Nitrogen, Helium, Argon etc are industrially important gases. Nature has provided us an abundant supply of these gases in atmosphere and under the earth crust. Oxygen and Nitrogen are available from atmosphere and helium, argon etc are available from the earth crust. The main aim is to harness these gases and use it for important purpose. The production and utilization of these gases form the major part of economy. This can be said as a indicator of technological improvement. These gases have various uses. Oxygen is used for steel manufacturing, rocket propulsion and medical applications. Argon is used in TIG welding and high temperature furnaces. Helium finds its use in superconductivity, nuclear reactors etc. Hydrogen is used as fuel in rocket propulsion systems. Nitrogen is a major input to the fertilizer industry. It is also used in cryosurgery and semiconductor industry. Nitrogen is used as a blanket gas in most chemical processes, and serves as the basic raw material in production of ammonia based fertilizers and chemicals. High purity nitrogen is used as a carrier gas in the electronic industry; and liquid nitrogen provides the most effective cooling medium for many low temperature processes – from shrink fitting to cryosurgery.

These important gases are first trapped from the various sources and a low temperature process known as air separation is used to separate them from each other. This air separation is carried out using expanding turbines. Air separation using turbo expanders has several benefits over high pressure (Linde) process. These benefits include low capital cost, better product mix and high operational flexibility. While room temperature processes, based on adsorption and membrane separation, are finding increasing application, particularly for low purity products, cryogenic distillation still remains the predominant method of producing bulk industrial gases. The cryogenic distillation process, operating at temperatures below 100K, offers several advantages over its room temperature counterparts. This process is economical in large scale, delivers both gaseous and liquid products, produces argon and rare gases (such as neon, krypton and xenon), and can respond to variation in demand in product mix.

In petrochemical industries, turbo expanders are used for separation of propane and heavier hydrocarbons from natural gas stream. Turbo expanders generate low temperature necessary for recovery of ethane and do it less expensively.

Expansion turbines are also used for power recovery applications as in refrigeration and high pressure wellhead gas, in power cycles using geothermal heat, in energy recovery in pressure let down, in Organic Rankine Cycle used in cryogenic process plants in order to achieve total utility consumption and in paper and other industries for waste gas energy recovery.

The expansion turbines can operate continuously for years and are more reliable than the other forms of reciprocating expanders used widely earlier. This is made possible by use of gas lubricated bearings, which use the process gas as the lubricant. While larger machines use axial flow geometry, cryogenic turbines universally adopt mixed flow, radial inlet and axial discharge, configuration. Multi-staging is difficult to achieve with radial or mixed flow geometry. Therefore, cryogenic turbines always adopt single stage expansion, irrespective of the expansion ratio. In addition to their role in producing liquid cryogens, turbo-expanders provide refrigeration in a variety of other applications, at both cryogenic and normal temperatures. Closed cycle cryo-coolers based on the Reversed Brayton cycle are used in cooling of radiation detectors and superconducting magnets.

CHAPTER 2

HISTORY REVIEW

HISTORY REVIEW-

The concept that a turbine can be used as a refrigerant machine was first introduced by Lord Rayleigh. In a letter of 26th June 1898 to Nature, he suggested the use of turbine instead of a piston expander for air liquefaction because of practical difficulties being encountered with the low temperature reciprocating machines. In this letter, Rayleigh emphasized the most important function of and cryogenic expander, which is to production of the cold, rather than the power produced. This followed a series of early patents on cryogenic expansion turbine. In 1898 The British engineer Edgar C Thrupp patented a simple liquefying system using an expansion turbine. Thrupp's expander was a double flow machine entering the center and dividing into two oppositely flowing streams. Each end of the rotor consists of 7 discs on each of which were from two to four row of blades parallel with the rotor axis. Airflow was from the center outward through the moving blades on each disk and intervening fixed blades on the turbine casing. The casing was so shaped internally as to bring air discharge from periphery of each rotor disk back to hub of the succeeding disk for further expansion.

Contemporaneously with Thrupp, an American engineer Joseph E Johnson patented an apparatus for liquefying gases. A fraction of air to be liquefied was to be condensed in the turbine nozzle and fall to the bottom of the liquefaction chamber for collection, and run off upon exhausting from the turbine. A refrigerative expansion turbine with a tangential inward flow pattern was patented by the Americans Charles F and Orrin J Crommett in 1914. Gas was to be admitted to the turbine wheel by a pair of nozzles, but it was specified that any desired numbers of nozzle could be used. The turbine blades were curved to present slightly concave faces to the jet from the nozzle. These blades were comparatively short, not exceeding very close to the rotor hub.

In 1922, the American engineer and teacher Harvey N Davis had patented an expansion turbine of unusual thermodynamic concept. This turbine was intended to have several nozzle blocks each receiving a stream of gas from different temperature level of high pressure side of the main heat exchanger of a liquefaction apparatus. Davis pointed out that if the supply pressure were sufficiently high all streams would be expanded into the two phase region and so although achieving varying degrees of wetness, would reach the same terminal temperature.

Successful commercial application of an expansion turbine for gas liquefaction does not have been made until the early 1930's. This was done in Linde Works in Germany. The turbine used was an axial flow single stage impulse machine. Later in the year 1936 it was replaced by an inward radial flow turbine based on a patent by an Italian inventor, Guido Zerkowitz. One feature was a reversing chamber fitted inside the turbine wheel to give a second admission of the gas to the moving blades. In this way, velocity compounding could be achieved with a consequent reduction in the wheel speed. The patent specification set forth may details of turbine construction to keep refrigerate and piping losses minimum. The shaft bearings were to be entirely outside the turbine housing being within the casing of the machine driven and so removed from the cold zone.

Peter Kapitza, a well known Russian in cryogenics in the year 1939 came out with a break through paper. It contains of two useful conclusions:

1. In this Kapitza compared the thermodynamics of the liquefiers operating on high and low pressure cycles and concluded that a low pressure liquefier is better than high pressure liquefier.
2. Secondly Kapitza undertook to show by analysis and experimental results that an inward radial flow turbine would preferable to an axial impulse machine.

After the works of Kapitza one of the first well documented air liquefaction turbines to be built and operated was that designed by Elliot company and constructed by Sharples company which was done in 1942 and the machine was described as Swearingen. The turbine was a radial inflow reaction type with designed speed of 22000rpm. The turbine was supported by ball bearings.

Work on the small gas bearing turbo expander commenced in the early fifties by Sixsmith at Reading University on a machine for a small air liquefaction plant. In 1958, the United Kingdom Atomic Energy Authority developed a radial inward flow turbine for a nitrogen production plant. During 1958 to 1961 Stratos Division of Fairchild Aircraft Co. built blower loaded turbo expanders, mostly for air separation service. Voth et. developed a high speed turbine expander as a part of a cold moderator refrigerator for the Argonne National Laboratory (ANL). The first commercial turbine using helium was operated in 1964 in a refrigerator that produced 73 W at 3 K for the Rutherford helium bubble chamber. A high speed turbo alternator was developed by General Electric Company, New York in 1968, which ran on a practical gas bearing system capable of operating at cryogenic temperature with low loss.

National Bureau of Standards at Boulder, Colorado developed a turbine of shaft diameter of 8 mm. The turbine operated at a speed of 600,000 rpm at 30 K inlet temperature. In 1974, Sulzer Brothers, Switzerland developed a turbo expander for cryogenic plants with self acting gas bearings. In 1981, Cryostar, Switzerland started a development program together with a magnetic bearing manufacturer to develop a cryogenic turbo expander incorporating active magnetic bearing in both radial and axial direction. In 1984, the prototype turbo expander of medium size underwent extensive experimental testing in a nitrogen liquefier. Izumi at Hitachi, Ltd., Japan developed a micro turbo expander for a small helium refrigerator based on Claude cycle. The turbo expander consisted of a radial inward flow reaction turbine and a centrifugal brake fan on the lower and upper ends of a shaft supported by self acting gas bearings. The diameter of the turbine wheel was 6mm and the shaft diameter was 8mm. The rotational speeds of the 1st and 2nd stage turbo expander were 816,000 and 519,000 rpm respectively.

A simple method sufficient for the design of a high efficiency expansion turbine is outlined by Kun. A study was initiated in 1979 to survey operating plants and generates the cost factors relating to turbine developed by Kun & Sentz. Sixsmith in collaboration with Goddard Space Flight Centre of NASA, developed miniature turbines for Brayton Cycle cryo coolers. They have developed of a turbine, 1.5 mm in diameter rotating at a speed of approximately one million rpm. Yang developed a two stage miniature expansion turbine made for 1.5 L/hr helium liquefier at the Cryogenic Engineering Laboratory of the Chinese Academy of Sciences. The turbines rotated at more than 500,000 rpm. The design of a small, high speed turbo expander was taken up by the National Bureau of Standards (NBS) USA. The first expander operated at 600,000 rpm in externally pressurized gas bearings. The turbo expander developed by

Kate was with variable flow capacity mechanism (an adjustable turbine), which had the capacity of controlling the refrigerating power by using the variable nozzle vane height.

A wet type helium turbo expander with expected adiabatic efficiency of 70% was developed by the Naka Fusion Research Centre affiliated to the Japan Atomic Energy Institute. The turbo expander consists of a 40 mm shaft, 59 mm impeller diameter and self acting gas journal and thrust bearings. Ino developed a high expansion ratio radial inflow turbine for a helium liquefier of 100 L/hr capacity for use with a 70 MW superconductive generator. Davydenkov developed a new turbo expander with foil bearings for a cryogenic helium plants in Moscow, Russia. The maximum rotational speed of the rotor was 240,000 rpm with the shaft diameter of 16 mm. The turbo expander third stage was designed and manufactured in 1991, for the gas expansion machine regime, by “Cryogenmash”. Each stage of the turbo expander design was similar, differing from each other by dimensions only produced by “Heliummash”.

The ACD Company incorporated gas lubricated hydrodynamic foil bearings into a TC–3000 turbo expander. Several Cryogenic Industries has been involved with this technology for many years including Mafi-Trench. Agahi have explained the design process of the turbo expander utilizing modern technology, such as Computational Fluid Dynamic software, Computer Numerical Control Technology and Holographic Techniques to further improve an already impressive turbo expander efficiency performance. Improvements in analytical techniques, bearing technology and design features have made turbo expanders to be designed and operated at more favorable conditions such as higher rotational speeds. A Sulzer dry turbo expander, Creare wet turbo expander and IHI centrifugal cold compressor were installed and operated for about 8000 hrs in the Fermi National Accelerator Laboratory, USA. This Accelerator Division/Cryogenics department is responsible for the maintenance and operation of both the Central Helium Liquefier (CHL) and the system of 24 satellite refrigerators which provide 4.5 K refrigeration to the magnets of the Tevatron Synchrotron. Theses expanders have achieved 70% efficiency and are well integrated with the existing system. Sixsmith at Creare Inc., USA developed a small wet turbine for a helium liquefier set up at the particle accelerator of Fermi National laboratory. The expander shaft was supported in pressurized gas bearings and had a 4.76 mm turbine rotor at the cold end and a 12.7 mm brake compressor at the warm end. The expander had a design speed of 384,000 rpm and a design cooling capacity of 444 Watts. Xiong at the institute of cryogenic Engineering, China developed a cryogenic turbo expander with a rotor of 103 mm long and weighing 0.9 N, which had a working speed up to 230,000 rpm. The turbo expander was experimented with two types of gas lubricated foil journal bearings. The L'Air liquid company of France has been manufacturing cryogenic expansion turbines for 30 years and more than 350 turbo expanders are operating worldwide, installed on both industrial plants and research institutes. These turbines are characterized by the use of hydrostatic gas bearings, providing unique reliability with a measured Mean Time between failures of 45,000 hours. Atlas Copco has manufactured turbo expanders with active magnetic bearings as an alternative to conventional oil bearing system for many applications.

India has been lagging behind the rest of the world in this field of research and development. Still, significant progress has been made during the past two decades. In CMERI Durgapur, Jadeja developed an inward flow radial turbine supported on gas bearings for cryogenic plants. The device gave stable rotation at about 40,000 rpm. The programme was, however, discontinued before any significant progress could be achieved. Another programme at IIT Kharagpur developed a turbo expander unit by using aerostatic thrust and journal bearings which had a working speed up to 80,000 rpm. Recently Cryogenic Technology Division, BARC developed Helium refrigerator capable of producing 1 kW at 20K temperature

CHAPTER 3

THEORY

THEORY

3.1 GENERAL DESCRIPTION OF A CRYOGENIC TURBINE EXPANDER-

A cryogenic turbo expander consists of the following components-

- ✓ Nozzle
- ✓ Turbine Wheel
- ✓ Diffuser
- ✓ Shaft
- ✓ Brake Compressor
- ✓ Radial Bearing
- ✓ Thrust Bearing
- ✓ Housing
- ✓ Plumbing an Instrumentation

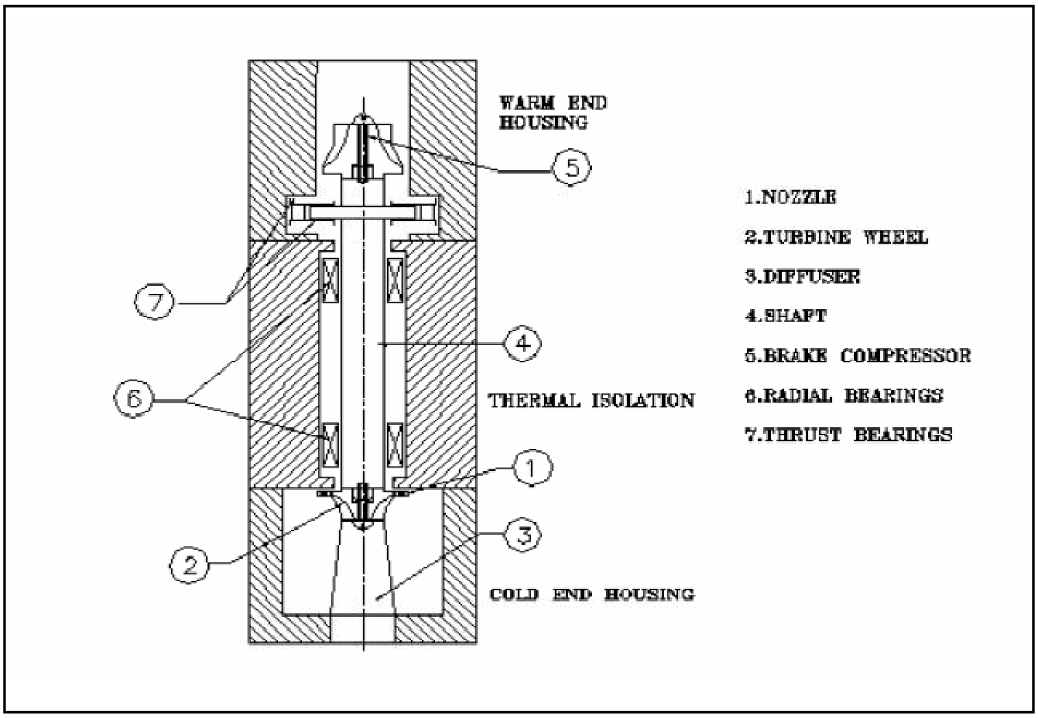


FIGURE 1: SCHEMATIC OF A CRYOGENIC TURBOEXPANDER

3.2 DESIGN AND OVERALL GEOMETRY-

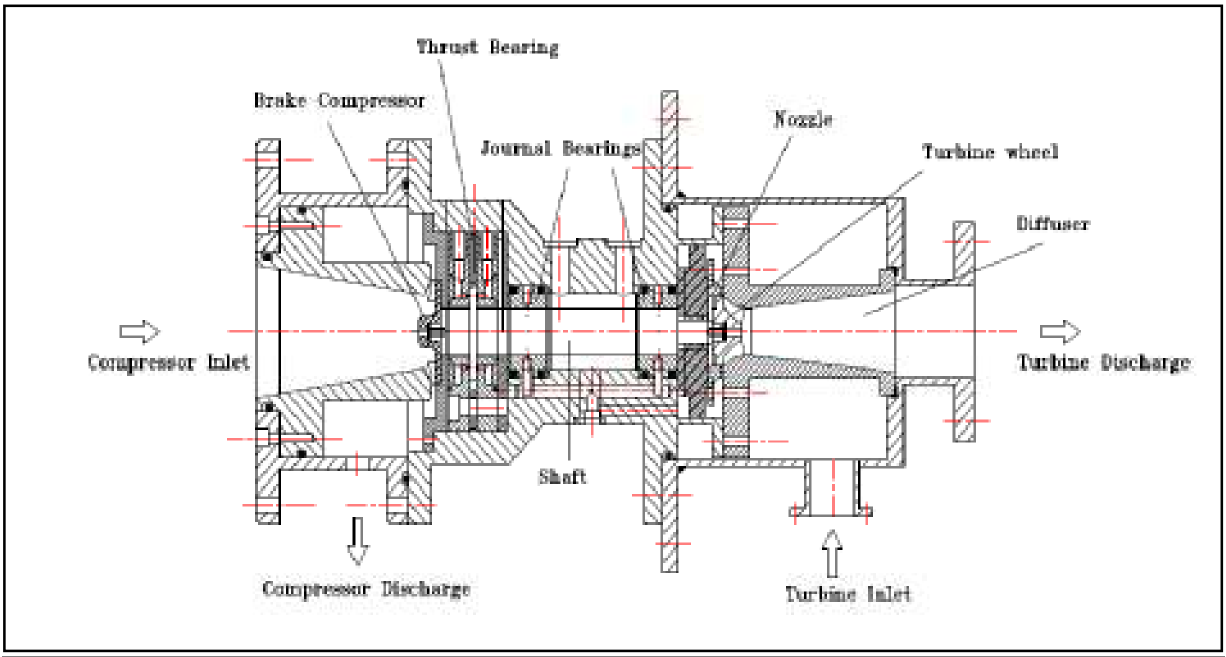


FIGURE 2: SECTION OF THE TURBINE DISPLAYING ITS COMPONENTS

A cryogenic turbo-expander is a 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 recompressor 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. 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. The various components are described below-

ROTOR-

The rotor is mounted vertically. The rotor consists of the shaft with a collar integrally machined on it to provide thrust bearing surfaces, the turbine wheel and the brake compressor mounted on opposite ends. The impellers are mounted at the extreme ends of the shaft while the bearings are in the middle.

NOZZLE-

The nozzles expand the inlet gas isentropically to high velocity and direct the flow on to the wheel at the correct angle to ensue smooth, impact free incidence on the wheel blades. A set of static nozzles must be provided around the turbine wheel to generate the required inlet velocity and swirl. The flow is subsonic, the absolute Mach number being around 0.95. Filippi has derived the effect of nozzle geometry on stage efficiency by a comparative discussion of three nozzle styles: fixed nozzles, adjustable nozzles with a centre pivot and adjustable nozzles with a trailing edge pivot. At design point operation, fixed nozzles yield the best overall efficiency. Nozzles should be located at the optimal radial location from the wheel to minimize vaneless space loss and the effect of nozzle wakes on impeller performance. Fixed nozzle shapes can be optimized by rounding the noses of nozzle vanes and are directionally oriented for minimal incidence angle loss.

The throat of the nozzle has an important influence on turbine performance and must be sized to pass the required mass flow rate at design conditions. Converging–diverging nozzles, giving supersonic flow are not generally recommended for radial turbines. The exit flow angle and exit velocity from nozzle are determined by the angular momentum required at rotor inlet and by the continuity equation. The throat velocity should be similar to the stator exit velocity and this determines the throat area by continuity. Turbine nozzles designed for subsonic and slightly supersonic flow are drilled and reamed for straight holes inclined at proper nozzle outlet angle. In small turbines, there is little space for drilling holes; therefore two dimensional passages of appropriate geometry are milled on a nozzle ring. The nozzle inlet is rounded off to reduce frictional losses.

An important forcing mechanism leading to fatigue of the wheel is the nozzle excitation frequency. As the wheel blades pass under the jets emanating from the stationary nozzles, there is periodic excitation of the wheel. The number of blades in the nozzle and that in the wheel should be mutually prime in order to raise this excitation frequency well beyond the operating speed and to reduce the overall magnitude of the peak force. The number of vanes as 17 in the nozzle for 7 in the wheel has been chosen.

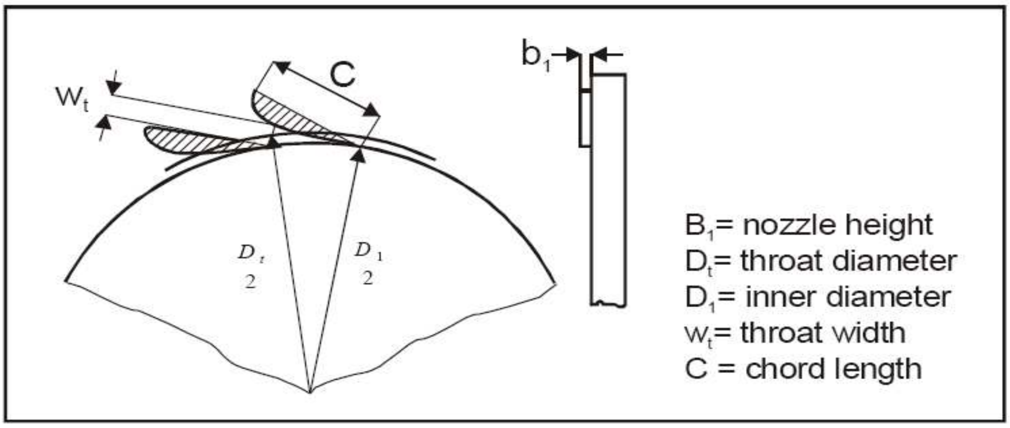


FIGURE 3: MAJOR DIMENSIONS OF NOZZLE

DIFFUSER-

The diffuser acts as a compressor, converting most of the kinetic energy in the gas leaving the rotor to potential energy in the form of pressure rise. The expansion ratio in the rotor is thereby increased with a corresponding gain in efficiency. The efficiency of a diffuser may be defined as the fraction of the inlet kinetic energy that gets converted to gain in static pressure. The Reynolds number based on the inlet diameter normally remains around 105. The efficiency of a conical diffuser with regular inlet conditions is about 90% and is obtained for a semi cone angle of around 5° to 6°. According to Shepherd, the optimum semi cone angle lies in the range of 3°-5° [24]. A higher cone angle leads to a shorter diffuser and hence lower frictional loss, but enhances the chance of flow separation.

The diffuser can be seen as an assembly of three separate sections operating in series – a converging section or shroud, a short parallel section and finally the diverging section where the pressure recovery takes place. The converging portion of the diffuser acts as a casing to the turbine. A clearance is provided to cover the tolerances of form, position and profile of the wheel, diffuser and the assembly. In addition, it covers the radial deflection of the wheel due to centrifugal stresses. The differential contraction between the wheel and the diffuser at low temperature usually acts to enhance this clearance.

SHAFT-

The force acting on the turbine shaft due to the revolution of its mass center and around its geometrical center constitutes the major inertia force. A restoring force equivalent to a spring force for small displacements, and viscous forces between the gas and the shaft surface, act as spring and damper to the rotating system. The film stiffness depends on the relative position of the shaft with respect to the bearing and is symmetrical with the center-to-center vector.

BRAKE COMPRESSOR-

The power developed in the expanders may be absorbed by a geared generator, oil pump, viscous oil brake or blower wheel. Where relatively large amounts of power are involved, the generator provides the most effective means of recovery. Induction motors running at slightly above their synchronous speed have been successfully used for this service. This does not permit speed variation which may be desirable during plant start up or part load operation. A popular loading device at lower power levels is the centrifugal compressor. Because of its simplicity and ease of control the centrifugal compressor is ideally suited for the loading of small turbines. It has the additional advantage that it can operate at high speeds. For small turbines whose work output exceeds the capacity of a centrifugal gas compressor, an electrical or oil brake may be used. The electrical device may be an eddy current brake or permanent magnet alternator, the latter having the advantage that heat is generated in an external load. The power generated by the turbine is absorbed by means of a centrifugal blower which acts as a brake. The helium gas in the brake circuit is circulated by the blower through a water cooled heat exchanger and a throttle valve. The throttle valve is used to adjust the load on the blower and the corresponding speed of the shaft. The blower is over-designed so that when the throttle is fully open the shaft speed is less than the optimum value. The heat exchanger removes the heat energy equivalent of the shaft work generated by the turbine from the system. Thus the turbine removes heat from the process gas and transfers it to the cooling water.

The turbo expander brake assembly is designed in the form of a centrifugal wheel of diameter 11.5 mm with a control valve at the inlet which provides for variation in rotor speed within 20%. The heat of friction is removed by the flow of lubricant through the static gas bearings thereby ensuring constant temperature of the parts supporting the rotor.

Various other components that form the turbo expander include bearings(thrust bearing, journal bearing etc) and seals. These also form an integral part of the turbo expander. The bearings are meant for proper support to the rotors. Seals are meant to minimize the heat leakage between warm and cold ends due to flow of gas along the shaft.

3.3 PARAMATERS OF TURBINE WHEEL-

The various thermodynamic parameters that at inlet and the exit are listed below. The turbine assembly include the nozzle, turbine wheel and diffuser. The phrase "inlet state" depicts the total or stagnation condition at the inlet, whereas "exit states" refer to the static conditions at the exit of the diffuser (state *ex*). The various that are enlisted below include the pressure, temperature, density, enthalpy, entropy. The inlet, actual exit along with ideal (isentropic) exit state is tabulated below. This table indicates the operating conditions of the turbine.

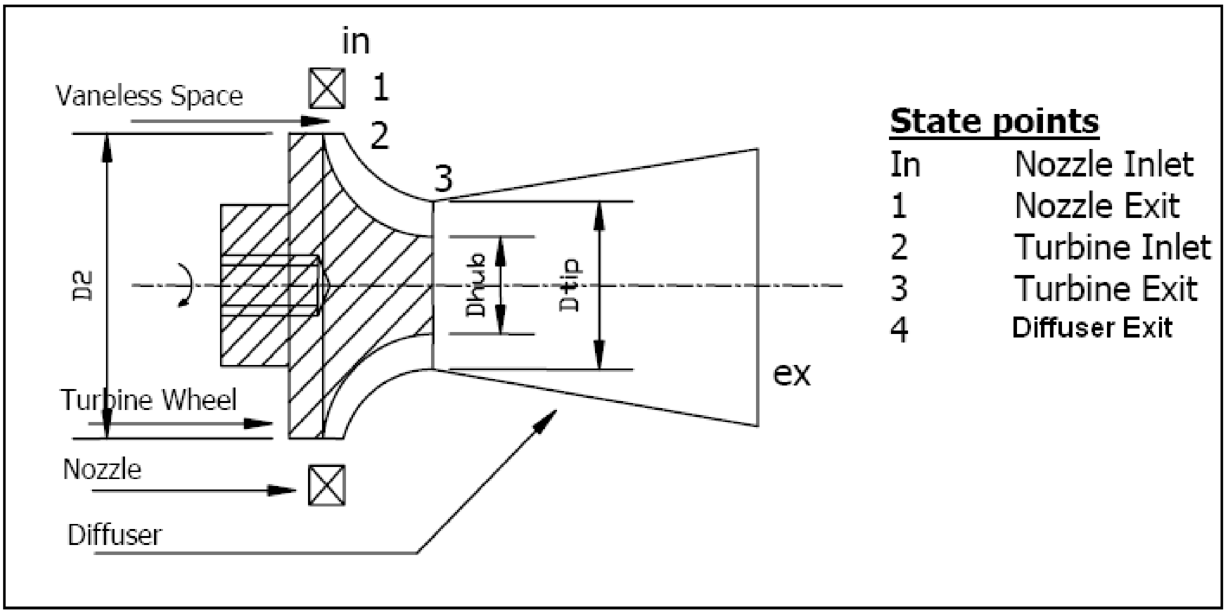


FIGURE 4: STATE POINTS OF TURBO EXPANDER

	Inlet (State 1)	Ideal (isentropic) exit state (*ex,s*)	Actual exit state (*ex*) (η = 75%)
Pressure (bar)	6.00	1.50	1.50
Temperature (K)	120.46	80.84	88.74
Density (kg/m^3)	18.06	6.64	5.95
Enthalpy (kJ/kg)	117.36	79.25	88.78
Entropy (kJ/kg.K)	5.325	5.325	5.437

TABLE 1: OPERATING CONDITIONS OF TURBINE

CHAPTER 4

OBJECTIVE AND ORGANISATION OF THE THESIS

OBJECTIVE AND ORGANISATION OF THE THESIS-

This thesis deals with the study of flow of a cryogenic turbo expander. Computational fluid analysis is carried out to determine the velocity profile and the temperature profile. Computational fluid analysis is carried using two software- Gambit 2.3 and Fluent 6.3

Gambit is used to build the model and mesh it and Fluent is used to carry out the velocity, temperature and pressure analysis. This total analysis is known as Computational fluid dynamics analysis. Before doing the analysis it is important to have an overview of what fluent is and how does it work.

4.1 WHAT IS CFD?

CFD or computational fluid dynamics is predicting what will happen, quantitatively, when fluids flow, often with the complications of simultaneous flow of heat, mass transfer (eg perspiration, dissolution), phase change (eg melting, freezing, boiling), chemical reaction (eg combustion, rusting), mechanical movement (eg of pistons, fans, rudders), stresses in and displacement of immersed or surrounding solids. Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. Even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. Ongoing research, however, may yield software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is often performed using a wind tunnel with the final validation coming in flight test.

The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for inviscid and Navier-Stokes equations for viscous flow). In addition, such a mesh can be either irregular (for instance consisting of triangles in 2D, or pyramidal solids in 3D) or regular; the distinguishing characteristic of the former is that each cell must be stored separately in memory. Where shocks or discontinuities are present, high resolution schemes such as Total Variation Diminishing (TVD), Flux Corrected Transport (FCT), Essentially Non Oscillatory (ENO), or MUSCL schemes are needed to avoid spurious oscillations (Gibbs phenomenon) in the solution. If one chooses not to proceed with a mesh-based method, a number of alternatives exist, notably Smoothed particle hydrodynamics (SPH), a Lagrangian method of solving fluid problems, Spectral methods, a technique where the equations are projected onto basis functions like the spherical harmonics and Chebyshev polynomials, Lattice Boltzmann methods (LBM), which simulate an equivalent mesoscopic system on a Cartesian grid, instead of solving the macroscopic system (or the real microscopic physics). It is possible to directly solve the Navier-Stokes equations for laminar flows and for turbulent flows when all of the relevant length scales can be resolved by the grid (a direct numerical simulation). In general however, the range of length scales appropriate to the problem is larger than even today's massively parallel computers can model. In these cases, turbulent flow simulations require the introduction of a turbulence model. Large eddy simulations (LES) and the Reynolds-averaged Navier-Stokes equations (RANS) formulation, with the k-ε model or the Reynolds stress model, are two techniques for dealing with these scales. In many instances, other equations are solved simultaneously with the Navier-Stokes equations. These other equations can include those describing species concentration (mass transfer), chemical reactions, heat transfer, etc. More advanced codes allow the simulation of more complex cases involving multi-phase flows (e.g. liquid/gas, solid/gas, liquid/solid), non-Newtonian fluids (such as blood), or chemically reacting flows (such as combustion).

4.2 DISCRETIZATION METHODS IN CFD

The stability of the chosen discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier-Stokes equations both admit shocks, and contact surfaces.

Some of the discretization methods being used are:

- Finite volume method (FVM). This is the "classical" or standard approach used most often in commercial software and research codes. The governing equations are solved on discrete control volumes. FVM recasts the PDE's (Partial Differential Equations) of the N-S equation in the conservative form and then discretize this equation. This guarantees the conservation of fluxes through a particular control volume. Though the overall solution will be conservative in nature there is no guarantee that it is the actual solution. Moreover this method is sensitive to distorted elements which can prevent convergence if such elements are in critical flow regions. This integration approach yields a method that is inherently conservative (i.e. quantities such as density remain physically meaningful

- Finite element method (FEM). This method is popular for structural analysis of solids, but is also applicable to fluids. The FEM formulation requires, however, special care to ensure a conservative solution. The FEM formulation has been adapted for use with the Navier-Stokes equations. Although in FEM conservation has to be taken care of, it is much more stable than the FVM approach. Subsequently it is the new direction in which CFD is moving. Generally stability/robustness of the solution is better in FEM though for some cases it might take more memory than FVM methods.

- Finite difference method. This method has historical importance and is simple to program. It is currently only used in few specialized codes. Modern finite difference codes make use of an embedded boundary for handling complex geometries making these codes highly efficient and accurate. Other ways to handle geometries are using overlapping-grids, where the solution is interpolated across each grid.

- Boundary element method. The boundary occupied by the fluid is divided into surface mesh.
- High-resolution schemes are used where shocks or discontinuities are present. To capture sharp changes in the solution requires the use of second or higher order numerical schemes that do not introduce spurious oscillations. This usually necessitates the application of flux limiters to ensure that the solution is total variation diminishing.

4.3 HOW IS THE WORKING DONE IN CFD

Working in CFD is done by writing down the CFD codes. CFD codes are structured around the numerical algorithms that can be tackle fluid problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces input problem parameters and to examine the results. Hence all codes contain three main elements:

1. Pre-processing.

2. Solver

3. Post - processing.

PRE-PROCESSING

Preprocessor consists of input of a flow problem by means of an operator friendly interface and subsequent transformation of this input into form of suitable for the use by the solver.

The user activities at the Pre-processing stage involve:

1) Definition of the geometry of the region*:* The computational domain. Grid generation is the subdivision of the domain into a number of smaller, no overlapping sub domains (or control volumes or elements Selection of physical or chemical phenomena that need to be modeled).

2) Definition of fluid properties*:* Specification of appropriate boundary conditions at cells, which coincide with or touch the boundary. The solution of a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of CFD solutions is governed by number of cells in the grid. In general, the larger numbers of cells better the solution accuracy. Both the accuracy of the solution & its cost in terms of necessary computer hardware & calculation time are dependent on the fineness of the grid. Efforts are underway to develop CFD codes with a (self) adaptive meshing capability. Ultimately such programs will automatically refine the grid in areas of rapid variation.

SOLVER

These are three distinct streams of numerical solutions techniques: finite difference, finite volume& finite element methods. In outline the numerical methods that form the basis of solver performs the following steps:

1) The approximation of unknown flow variables are by means of simple functions

2) Discretization by substitution of the approximation into the governing flow equations & subsequent mathematical manipulations.

POST-PROCESSING

As in the pre-processing huge amount of development work has recently has taken place in the post processing field. Owing to increased popularity of engineering work stations, many of which has outstanding graphics capabilities, the leading CFD are now equipped with versatile data visualization tools.

These include:

1) Domain geometry & Grid display

2) Vector plots

3) Line & shaded contour plots

4) 2D & 3D surface plots

5) Particle tracking

6) View manipulation (translation, rotation, scaling etc.)

CHAPTER 5

GAMBIT DESIGNING OF THE MODEL

GAMBIT DESIGNING OF THE MODEL-

5.1 OVERVIEW OF GAMBIT

The model development is carried out on gambit. It is the pre processing process where the model development is done and meshing of model is followed for further analysis. GAMBIT is a software package designed to help analysts and designers build and mesh models for computational fluid dynamics (CFD) and other scientific applications. GAMBIT receives user input by means of its graphical user interface (GUI). The GAMBIT GUI makes the basic steps of building, meshing, and assigning zone types to a model simple and intuitive, yet it is versatile enough to accommodate a wide range of modeling applications.

The various components of the turbine are designed using Gambit. The various components include blade, blade passage, nozzle and diffuser. The components are individually made and all are assembled. The profiles are generated with the help of coordinates available which have been generated. A single blade profile is made and then a group of seven blades are arranged to make the blade passage. After the formation of the blade passage the nozzle arrangement is done followed by the designing of the diffuser using the coordinate. All these components are finally assembled and meshing is done.

5.2 MODELLING OF THE COMPONENTS-

MODELLING THE BLADE PROFILE-

The hub and the tip streamlines are available in the table below. A ruled surface is created by joining the hub and tip streamlines with a set of tie lines. The surface so generated is considered as the mean surface within a blade. The suction and pressure surfaces of two adjacent channels are computed by translating the mean surface in the positive and negative Φ directions through half the blade thickness. Coordinates of all the blade surfaces are computed by further rotating the pair of surfaces over an angle $2\pi / Z$, i.e. 51.43 degrees for $Z = 7$. Non Uniform Rational B Splines are used to develop the solid surface.

TABLE 2: COORDINATES FOR GENERATION OF BLADE PROFILE-1

Tip Camberline			*Hub Camberline*		
z (mm.)	r (mm.)	phi (deg.)	z (mm.)	r (mm.)	phi (deg.)
-1.910	9.046	0.000	0.000	3.166	0.000
-1.439	9.121	5.502	0.426	3.383	5.502
-0.950	9.129	10.610	0.835	3.666	10.610
-0.469	9.134	15.347	1.251	3.954	15.347
0.002	9.143	19.732	1.675	4.243	19.732
0.463	9.158	23.786	2.107	4.533	23.786
0.914	9.181	27.524	2.546	4.826	27.524
1.353	9.213	30.961	2.990	5.125	30.961
1.782	9.256	34.111	3.437	5.432	34.111
2.201	9.311	36.987	3.884	5.752	36.987
2.611	9.381	39.602	4.328	6.085	39.602
3.010	9.466	41.966	4.766	6.436	41.966
3.398	9.570	44.091	5.193	6.806	44.091
3.777	9.693	45.989	5.606	7.196	45.989
4.143	9.839	47.672	6.001	7.608	47.672
4.497	10.009	49.150	6.372	8.042	49.150
4.837	10.206	50.435	6.716	8.496	50.435
5.160	10.431	51.540	7.028	8.969	51.540
5.464	10.686	52.475	7.305	9.457	52.475
5.747	10.970	53.253	7.546	9.958	53.253
6.007	11.283	53.891	7.749	10.468	53.891
6.240	11.624	54.380	7.916	10.982	54.380
6.445	11.990	54.752	8.048	11.496	54.752
6.623	12.379	55.007	8.148	12.009	55.007
6.774	12.787	55.154	8.219	12.518	55.154
6.899	13.116	55.202	8.267	13.116	55.202

z= axial length

r= radius

phi= angle of deflection measured in clockwise direction

TABLE 3: COORDINATES FOR GENERATION OF BLADE PROFILE -2

$z_{pressure}$(mm)	$r_{pressure}$(mm)	$\theta_{pressure}$(radian)	$z_{suction}$ (mm)	$r_{suction}$(mm)	$\theta_{suction}$(radian)
0	3.85	0.055	0	3.85	-0.055
0.45	3.92	0.166	0.45	3.92	0.068
0.91	3.99	0.26	0.91	3.99	0.172
1.36	4.07	0.339	1.36	4.07	0.261
1.82	4.14	0.404	1.82	4.14	0.336
2.27	4.22	0.458	2.27	4.22	0.4
2.72	4.31	0.502	2.72	4.31	0.453
3.17	4.41	0.537	3.17	4.41	0.497
3.62	4.53	0.566	3.62	4.53	0.533
4.06	4.66	0.588	4.06	4.66	0.562
4.49	4.82	0.605	4.49	4.82	0.585
4.91	5	0.617	4.91	5	0.603
5.32	5.21	0.627	5.32	5.21	0.616
5.71	5.46	0.633	5.71	5.46	0.626
6.08	5.74	0.637	6.08	5.74	0.633
6.42	6.05	0.64	6.42	6.05	0.637
6.72	6.39	0.641	6.72	6.39	0.64
6.99	6.77	0.642	6.99	6.77	0.641
7.23	7.16	0.642	7.23	7.16	0.642
7.43	7.58	0.642	7.43	7.58	0.642
7.59	8.01	0.642	7.59	8.01	0.642

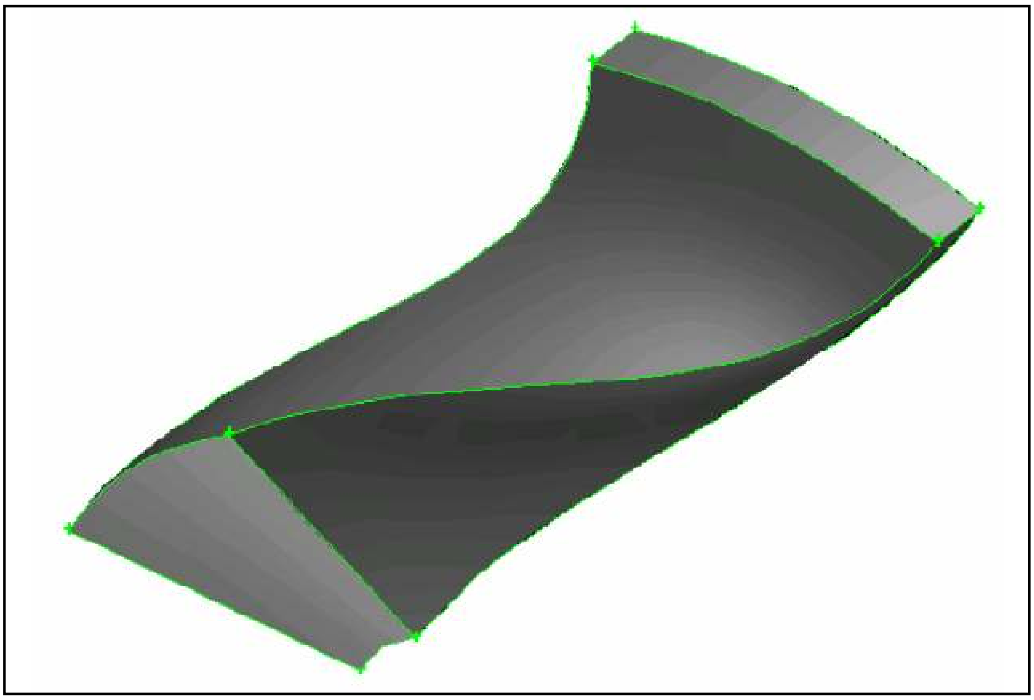


FIGURE 5: BLADE PROFILE GENERATED IN GAMBIT

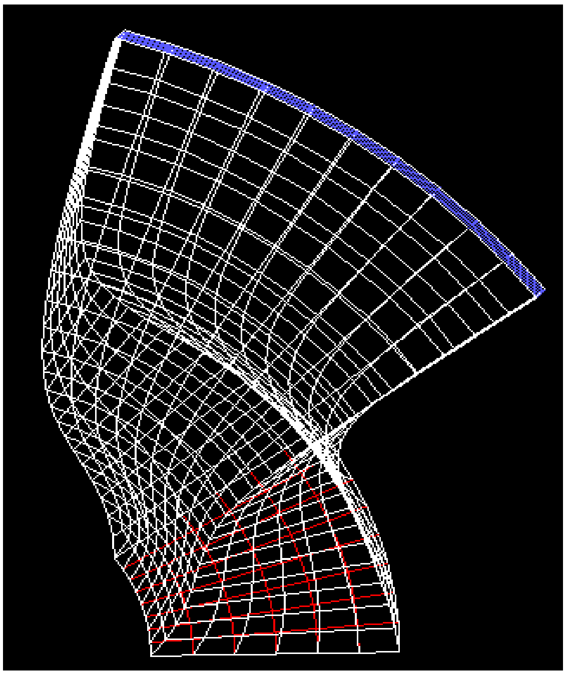


FIGURE 6: MESHED MODEL OF BLADE PROFILE

BLADE PASSAGE MODELLING-

A single blade profile which created is aligned to create the blade passage. Seven blades are aligned about a circle to form the blade passage profile.

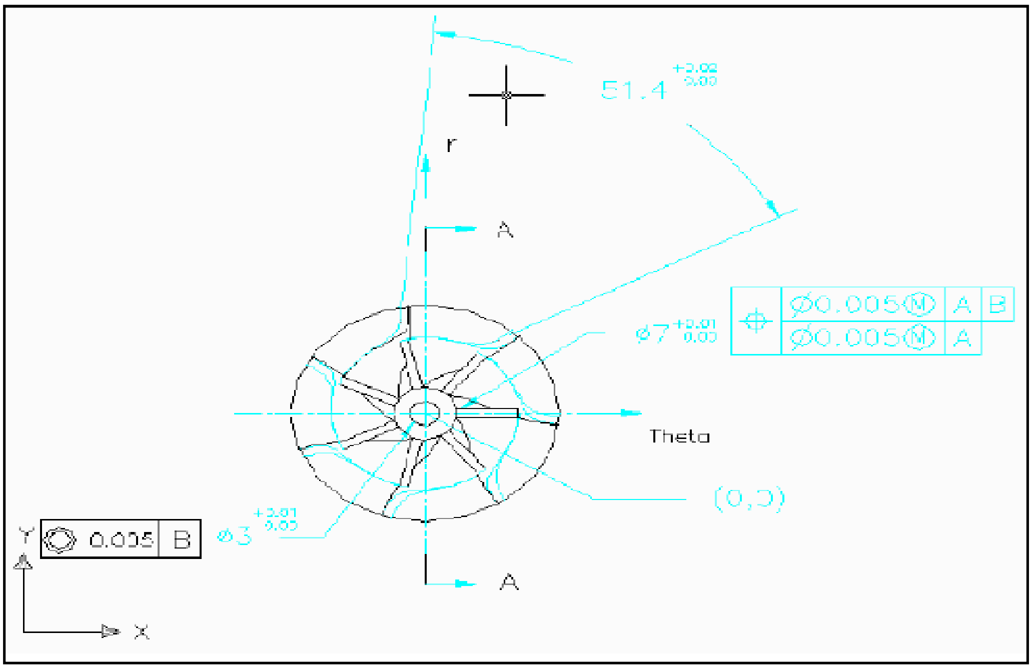


FIGURE 7: MODEL REPRESENTING BLADE PASSAGE

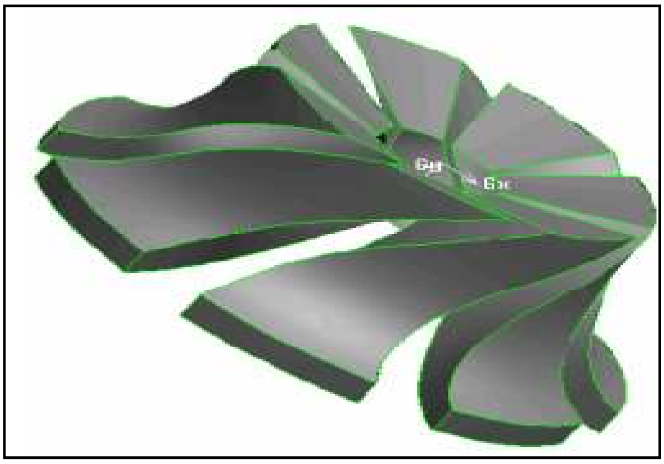


FIGURE 8: BLADE PASSAGE GENERATED IN GAMBIT

NOZZLE AND ITS ARRANGEMENT-

The number blades in the nozzle and the blades passage should be mutually prime. The number blades taken in blade passage are 7 and the number of blades in the nozzle is taken to be 17. The nozzle profile is done as per the autocad profile and the extrusions of the vanes are done to develop the nozzle arrangement 3D profile.

Point	X coordinate	Y coordinate
A	3.81	12.38
B	-0.43	17.06
C_1	0.00	0.00
C_2	0.702	16.00
C_3	-0.97	16.31
D	-0.68	14.48

Coordinates of a single vane.
[Other coordinates are generated by arraying in NC Part Programming]

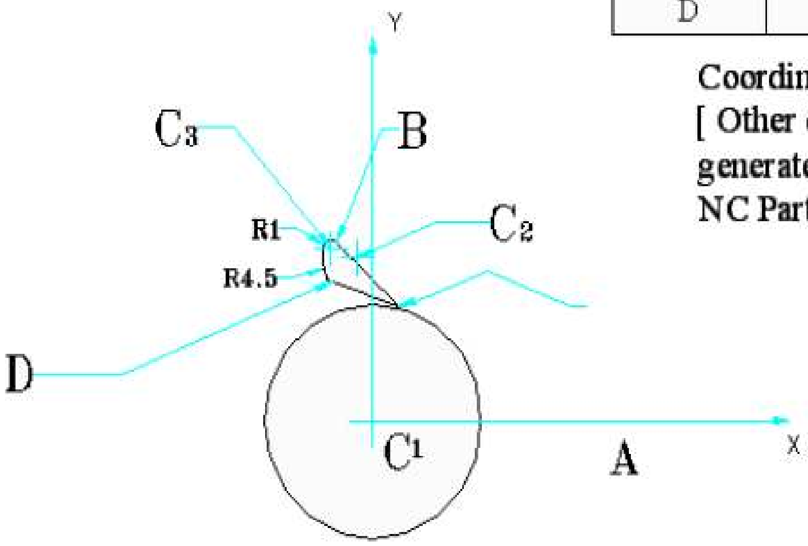


FIGURE 9: COORDINATES OF A SINGLE VANE AND ITS REPRESENTATION

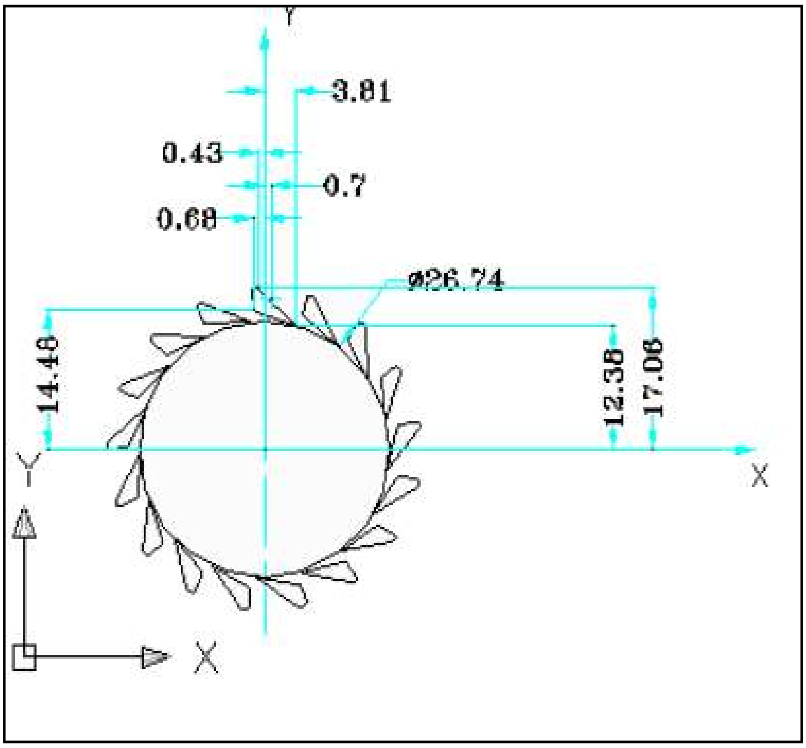


FIGURE 10: NOZZLE ARRANGEMENT REPRESENTATION

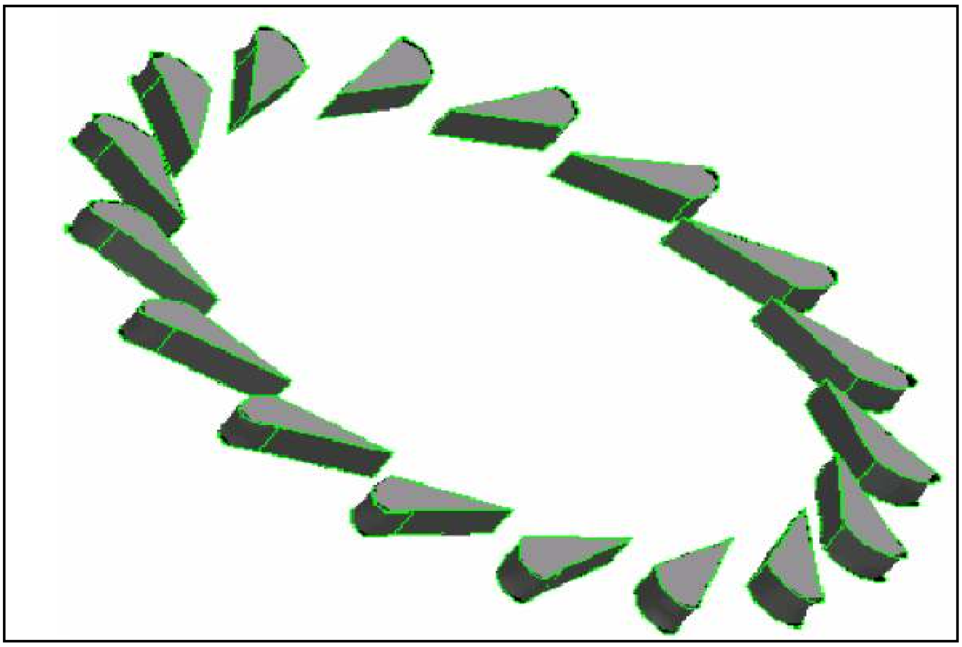
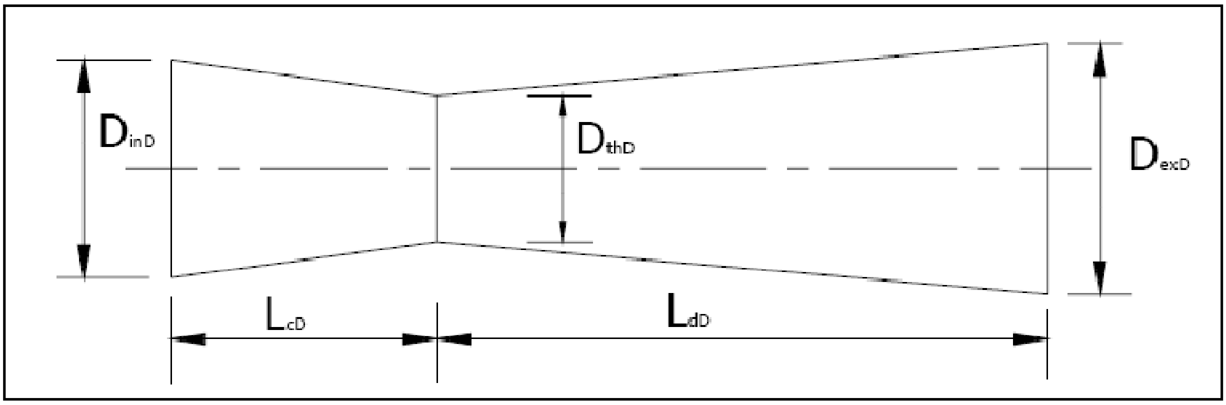


FIGURE 11: NOZZLE ARRANGEMENT

MODELLING THE DIFFUSER-

The coordinates are provided for the development of the 2D model of the diffuser. The model is then rotated about 360 degrees to get the 3D profile. For design purposes, the diffuser can be seen as an assembly of three separate sections operating in series – a converging section or shroud, a short parallel section and finally the diverging section. The converging portion of the diffuser acts as a casing to the turbine. The straight portion of the diffuser helps in reducing the non-uniformity of flow, and in the diverging section, the pressure recovery takes place. The geometrical specifications of the diffuser have chosen somewhat arbitrarily. Diameter of diffuser inlet is equal to diameter of the turbine inlet. Diameter of throat of diffuser is depending on the shroud clearance. The recommended clearance is 2% of the exit radius, which is approximately 0.2 mm for wheel. The differential contraction between the wheel and the diffuser at low temperature usually acts to enhance this clearance. The profile of the convergent section has been obtained by offsetting the turbine tip profile by o.2 mm radially. For diameter of diffuser exhaust, suggested exit velocity of the diffuser should be maintained near about 20 m/s with a half cone angle of 5.50.



D_{inD} = Inlet Diameter Diffuser, D_{thD} = Throat Diameter Diffuser, D_{exD} = Exit Diameter Diffuser, L_{cD} = Length Convergent Section, L_{dD} = Length Divergent Section

FIGURE 12: DIFFUSER NOMECLATURE

TABLE 4: COORDINATES FOR GENERATION OF DIFFUSER

Z(in mm)	X(in mm)	Z(in mm)	X(in mm)	Z(in mm)	X(in mm)
0	26.632	2.4	20.3325	7.4	18.6645
0.1	26.1098	2.5	20.2364	7.6	18.6604
0.2	25.5823	2.6	20.1404	7.8	18.6552
0.3	25.0383	2.7	20.0444	8	18.6486
0.4	24.5761	2.8	19.9646	8.2	18.6421
0.5	24.1616	2.9	19.8848	8.4	18.5836
0.6	23.7879	3	19.805	8.6	18.5204
0.7	23.4373	4.5	19.0485	8.8	18.492
0.8	23.1446	4.6	19.015	9	18.492
0.9	22.8519	4.7	18.9888	9.2	18.492
1	22.6023	4.8	18.9625	9.4	18.492
1.1	22.3606	4.9	18.9363	9.6	18.492
1.2	22.1343	5	18.9097	9.8	18.492
1.3	21.9335	5.2	18.8699	10	18.492
1.4	21.7328	5.4	18.8299	10.5	18.492
1.5	21.5547	5.6	18.7993	11	18.492
1.6	21.3874	5.8	18.7701	11.5	18.492
1.7	21.2202	6	18.7474	12	18.492
1.8	21.0742	6.2	18.7266	12.5	18.492
1.9	20.9349	6.4	18.7096	13	18.492
2	20.7956	6.6	18.6966	13.5	18.492
2.1	20.672	6.8	18.6845	14	18.492
2.2	20.5561	7	18.6769	14.7	18.492
2.3	20.4402	7.2	18.6693		

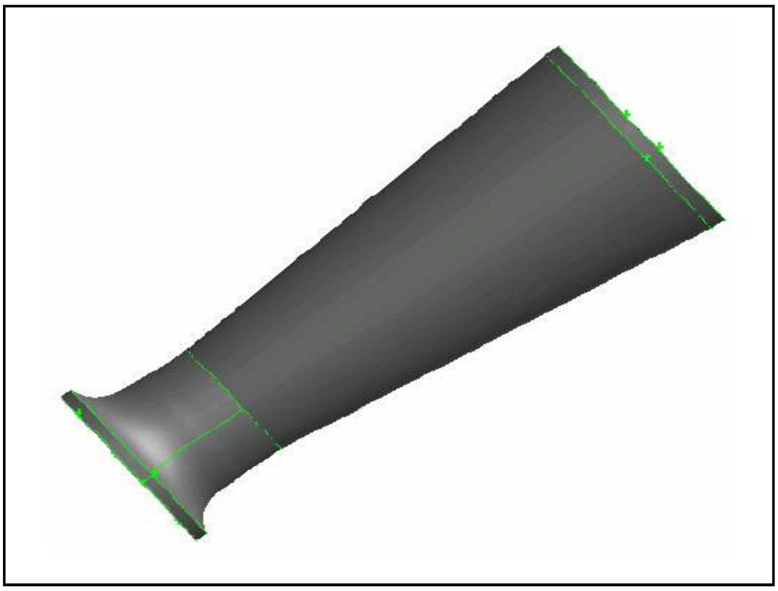


FIGURE 13: DIFFUSER GENERATED IN GAMBIT

5.3 ASSEMBLYING-

The blades passage, nozzle and diffuser are assembled to give the turbine assembly.

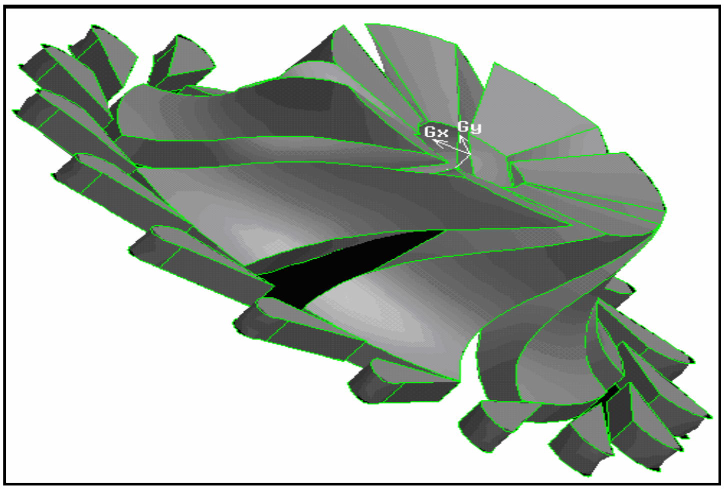


FIGURE 14: BLADE AND NOZZLE ASSEMBLY

5.4 MESHING AND DEFINING BOUNDARY CONDITIONS -

The assembly is meshed using tetrahedral elements of T-grid scheme type. The boundary conditions are defined as given. Nozzle inlet is taken as mass flow inlet and diffuser is taken as the pressure outlet. Here mixing planes are also defined at the interface of nozzle outlet and blade passage inlet as pressure outlet and inlet respectively and the interfaces at the blade passage outlet and diffuser inlet as pressure and inlet respectively. The nozzle, blades and diffuser flow path are defined as fluid and the nozzles and the blades are taken as solid. File is then saved for analysis in fluent.

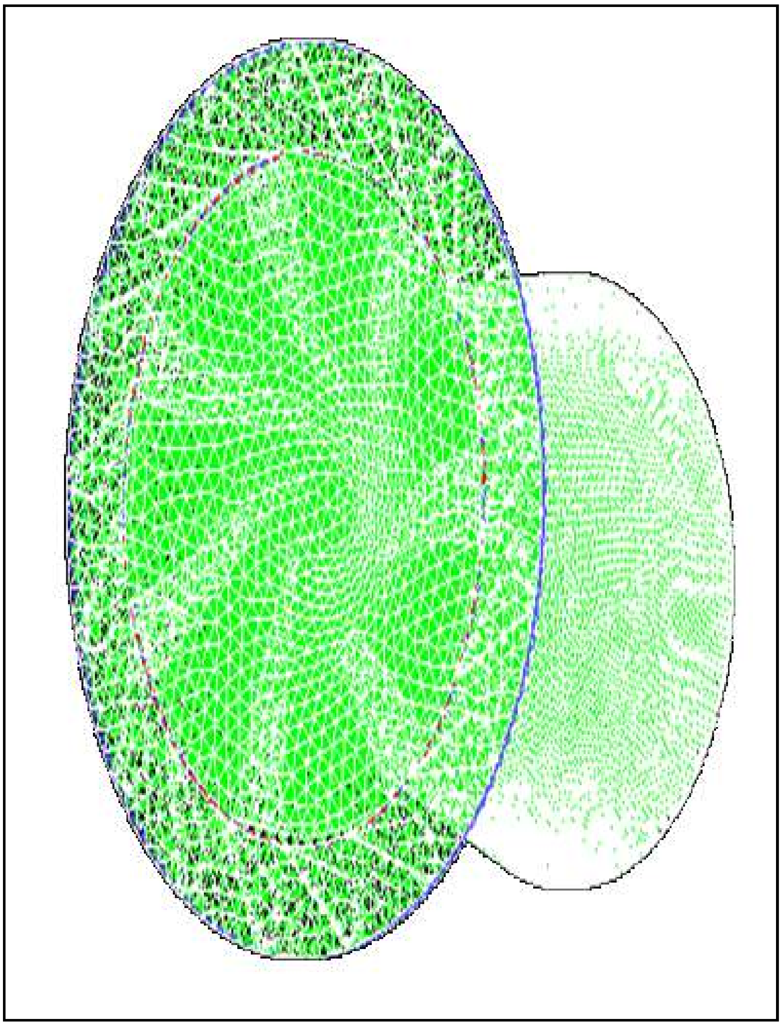


FIGURE 15: MESHED MODEL OF THE ASSEMBLY

CHAPTER 6

FLUENT ANALYSIS

FLUENT ANALYSIS-

6.1 OVERVIEW OF FLUENT-

FLUENT is the software used for modeling fluid flow and heat transfer in complex geometries. It provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex geometries with relative ease. It is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible. All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface. The basic procedural steps for solving a problem in FLUENT include:

1) Define the modeling goals.

2) Create the model geometry and grid.

3) Set up the solver and physical models.

4) Compute and monitor the solution.

5) Examine and save the results

6) Consider revisions to the numerical or physical model parameters, if necessary

FLUENT can model flow involving moving reference frames and moving cell zones, using several different approaches, and flow in moving and deforming domains (dynamic meshes). Solving flows in moving reference frames requires the use of moving cell zones. Problems that can be addressed include flow in a (single) rotating frame and flow in multiple rotating and/or translating reference frames. However in case of turbo machinery fluent uses multiple reference frame model, mixing plane model and sliding mesh model.

6.2 ANALYSIS-

The analysis is carried in fluent by importing the meshed file saved in gambit. The steps that are followed are given below which include all the conditions and the boundaries values for the problem statement.

Checking of mesh and Scaling-

The fluent solver is opened where 3D is selected and then the importing of the meshed file is done. The meshed file is then undergoes a checking where 51620 number of grids are found. After this grid check is done following which smoothing and swapping of grid is done. For this skewness method is selected and minimum skewness is set as 0.8 and the iterations are set as 8. The model is then smoothed and swapped where the number of faces comes to be 2696.

Following this the scaling is done. Scale is scaled to mm. Grid created was changed to mm. After this defining of various parameters are done.

Solver and Material Selection and Operating Condition Defining-

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. First the analysis is carried using laminar flow and then the k-epsilon is considered. 2 results are to be found out.

The selection of material is done. Material selected is nitrogen gas. The properties of nitrogen is taken as follows-

Density = 1.138 kg/m3
Cp (specific heat capacity) = 1040.67 J/kg K
Thermal conductivity = 0.0242 W/m K

Viscosity = 1.663 x 10-5 kg/m s

The analysis is carried out under operating conditions of 101325 Pascal. Gravity is not taken into consideration.

Mixing Plane Selection-

The mixing plane model was set up. Two mixing planes were needed, one at the interface between the pressure outlet of the Upstream nozzle outlet region and the pressure inlet at the adjacent face of the Blades passage Region. It was defined as radial mixing plane geometry. Similarly, the second mixing plane was defined at the pressure outlet of Blades passage and the pressure inlet to the Downstream Diffuser inlet region. It was defined as Axial mixing plane geometry. Under relaxation value was set as 1.

Boundary Conditions-

Nozzle Inlet-

Mass flow inlet was taken for the nozzle inlet and the value of mass flow rate was taken as 0.0606kg/s. Initial gauge pressure was taken as 500000 Pascal. Temperature was taken as 120K.

Mixing Planes-

Mixing planes were defined as defined earlier in gambit.

Blades and Flow Path-

For blades solid material taken was Aluminium and rotation axis origin was taken as (0,0,0) and rotation axis direction was set as (0,0,-1) and motion type selected was moving mesh. The rotational velocity was taken as 10400rad/s. Translational velocity was set as (0,0,0).

Outlet-

The diffuser was set as pressure outlet and the temperature was set to 80K and the pressure was set to 0 Pascal.

The similar analysis was also carried out for turbulent flow, using k-epsilon method. Here turbulence energy and dissipation rate are taken into consideration.

Controls Set Up-

The solutions controls are set as listed below.

The **under relaxation factor** was set as given-

Pressure-0.2

Density-1

Body forces-1

Momentum-0.5

Energy-1

Pressure Velocity Coupling was taken as **SIMPLE**

Discretization Equation are selected as given-

Pressure- PRESTO (Pressure Staggering Option)

Momentum- Second Order Upwind

Energy- Second Order Upwind (For turbulent flow Power Law was taken into consideration)

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.

CHAPTER 7

RESULTS

RESULTS-

The velocity profiles are drawn for both the laminar and turbulent (k-epsilon) flow. In case of laminar flow the velocity increases from the nozzle inlet (blue region) to reach a maximum at the nozzle outlet (red region). In case of the turbulent flow higher velocity range are to be seen. The velocity profile is quiet random in case of turbulent flow. However it is to be noted that the convergence is reached in turbulent flow much quicker in comparison to laminar flow.

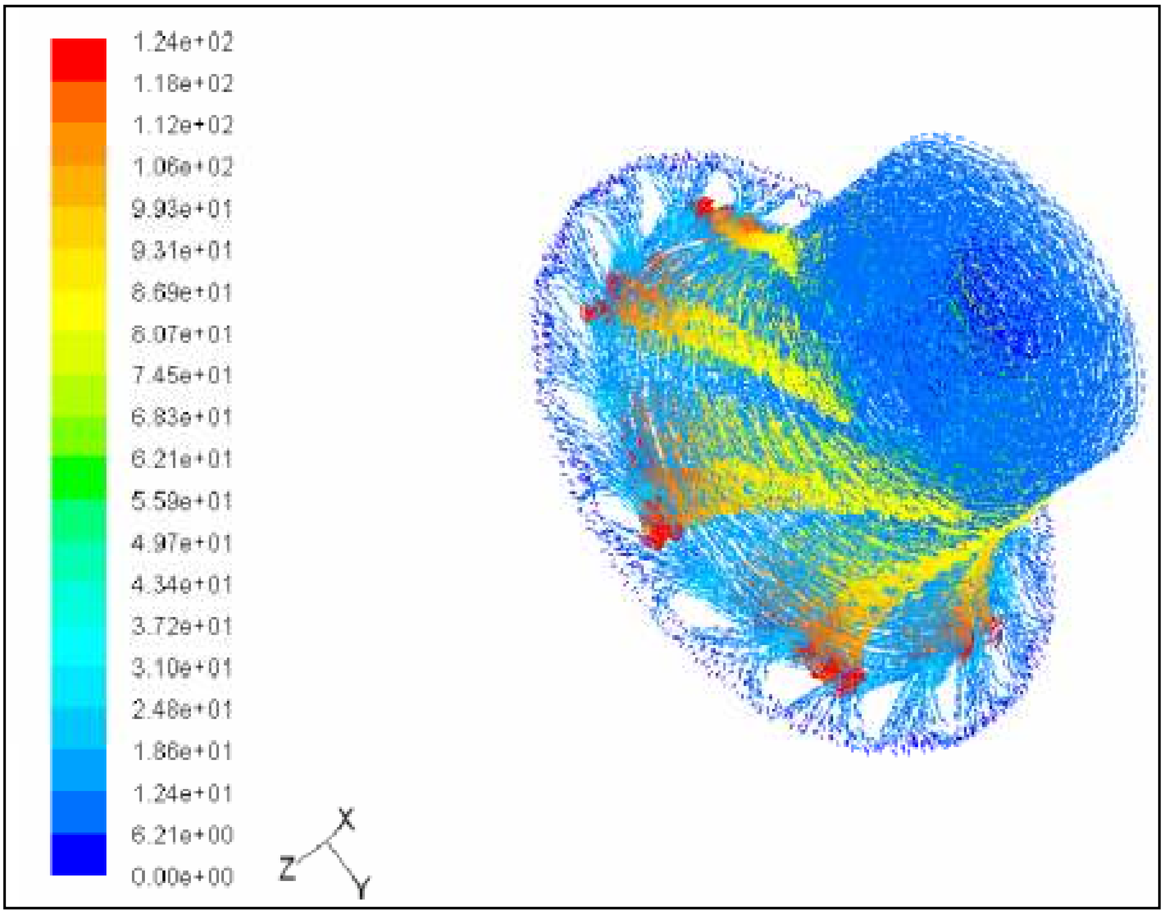


FIGURE 16: VELOCITY CONTOURS FOR LAMINAR FLOW IN TURBINE

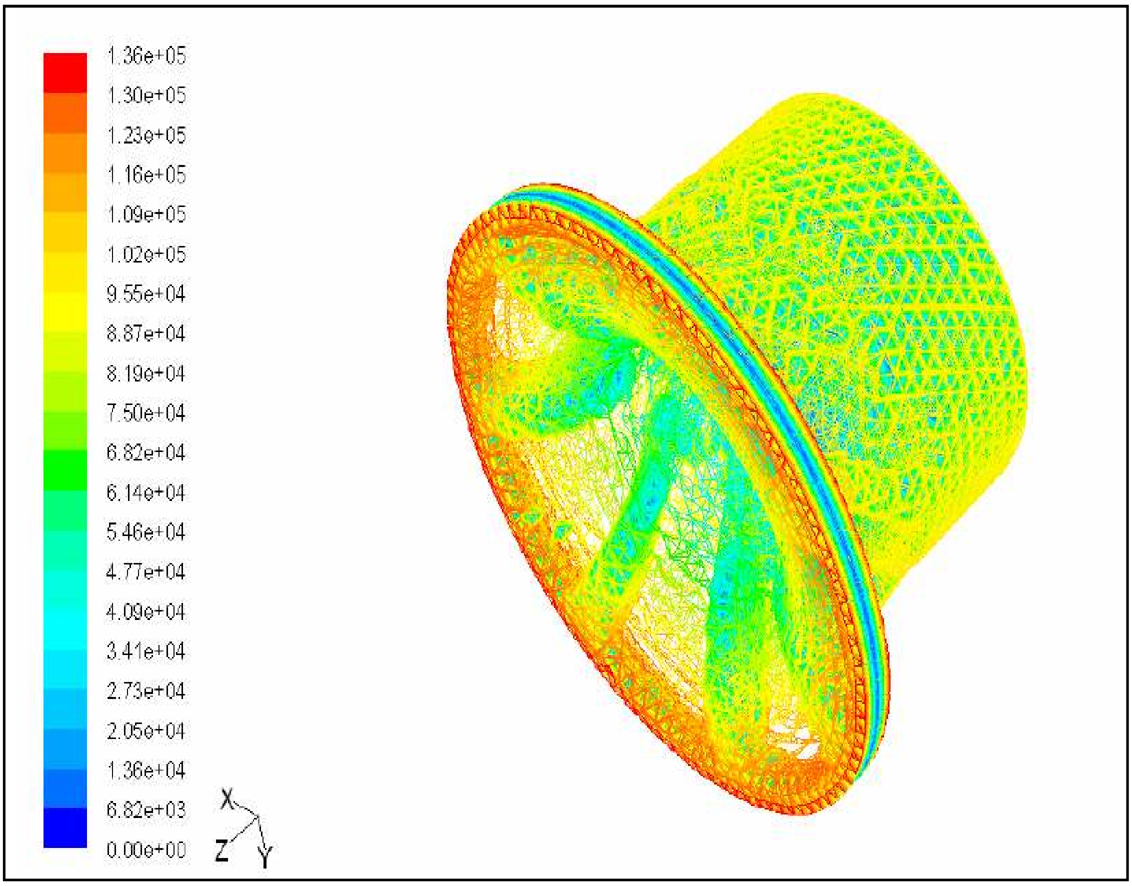


FIGURE 17: VELOCITY CONTOURS FOR TURBULENT FLOW IN TURBINE

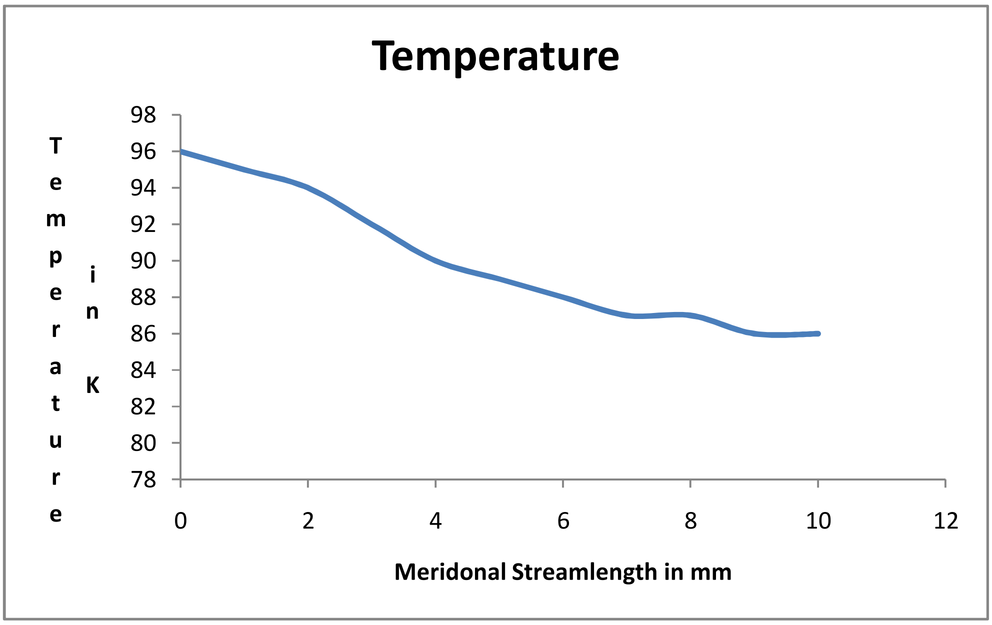


FIGURE 18: TEMPERATURE VARIATION ALONG MERIDONAL STREAMLENGTH IN TURBINE WHEEL

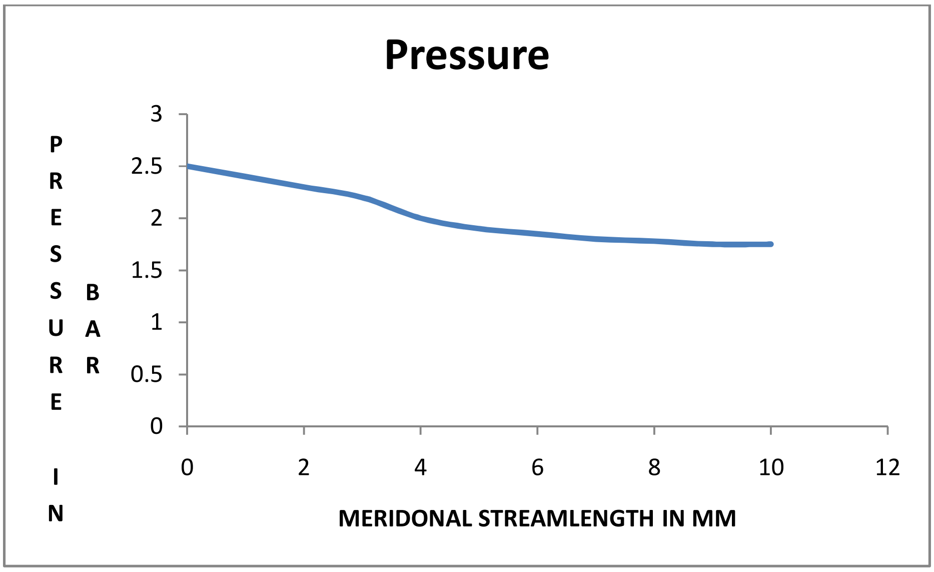


FIGURE 19: PRESSURE VARIATION ALONG MERIDONAL STREAMLENGTH IN TURBINE WHEEL

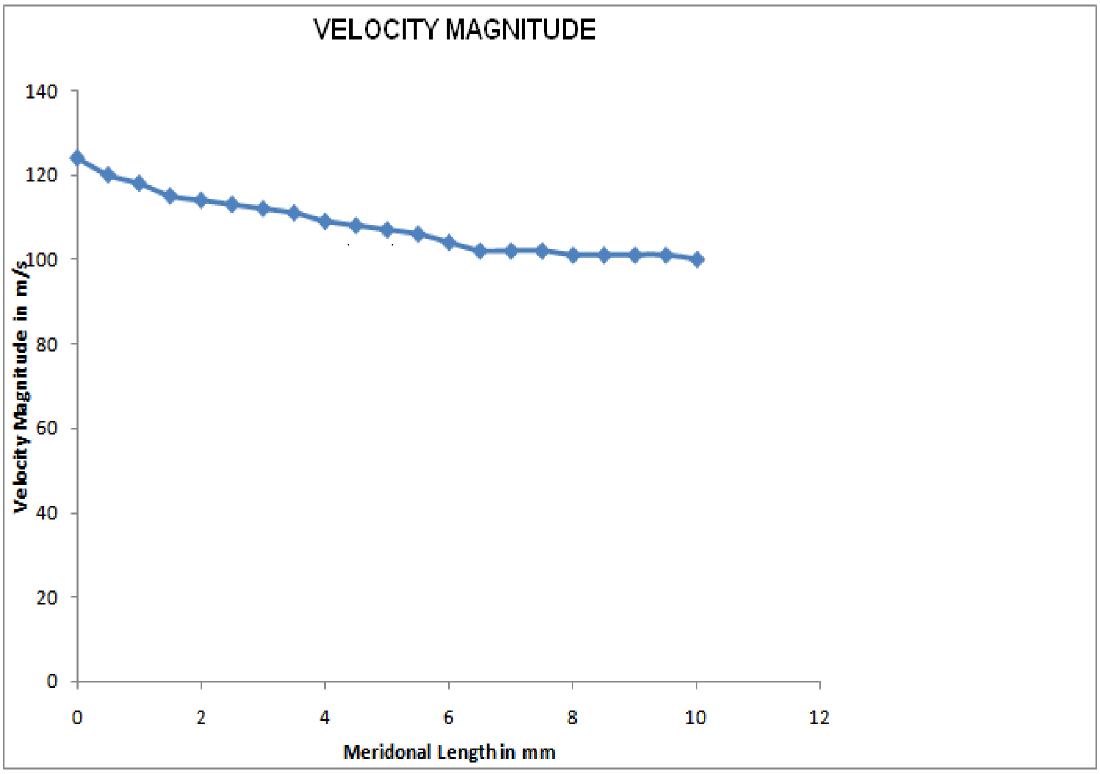


FIGURE 20: VARIATION OF VELOCITY ALONG THE MERIDONAL STREAMLENGTH IN TURBINE

Average Velocity magnitude has been found out. The average velocity is found out using

$$\frac{\int v.da}{\int da} = 86\text{m/s for laminar flow.}$$

CHAPTER 8

CONCLUSION

CONCLUSION-

Cryogenic turbine has great utility in industrial collection of gases. They also provide a wonderful medium for refrigeration. The modeling of the various parts of the turbine is done using Gambit and the computational fluid dynamics analysis is done using fluent software. The velocity contours and graphs indicating the variation of temperature, pressure and velocity along the meridonal streamlength is given. However the results are not accurate as the leakage are not taken into consideration. Moreover the original material used to construct the turbine is also not taken into consideration.

REFERENCES:

1) **Patankar Suhas V** Numerical Heat Transfer and Fluid Flow
2) **Ghosh, P and Sarangi, S** Thesis on “Analytical and Experimental Studies on Turboexpander” , IIT Kharagpur (2002)
3) **Arkharov, A., Marfenina, I. and Mikulin, Ye. (Trans. Kuznetsov, B.V.)** Theory and Design of Cryogenic Systems, Mir Publishers, Moscow
4) **Barron, R. F.** Cryogenic Systems Oxford University Press(1985)
5) **Blackford, J. E., Halford P. and Tantam, D. H.** Expander and pumps in G G Haselden (Ed) Cryogenic Fundamentals Academic Press London (1971)
6) **Flynn, T.M.** Cryogenic Engineering, Marcel-Dekkar Inc.
7) **Rama S. R. Gorla and Aijaz A. Khan** Turbomachinery Design and Theory
8) **Logan Jr, E**. Turbomachinery: Basic Theory and Applications **Marcel Dekker Inc.**