iJRASET
International Journal For Research in Applied Science and Engineering Technology

# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

**Volume:** 7 **Issue:** IX **Month of publication:** September 2019

**DOI:** http://doi.org/10.22214/ijraset.2019.9024

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
*ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177*
*Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com*

# Design and Material Suggestion for Radial Inflow Turbine (RIT) Impeller of Turbocharger by using Open FOAM and Ansys

Kiran R. Gerange[1], Dr. H. P. Khairnar[2]
*[1, 2]Department of Mechanical Engineering, Veermata Jijabai Technological Institute, Mumbai –400 019, Maharashtra, India*

***Abstract: Downsizing the internal combustion engines has been a major topic of study, recently. Among different techniques and technologies introduced through the time, using turbochargers is considered quite a demanding strategy. However, various considerations about the application of turbocharger and matching of the system have to be investigated. Nowadays, numerical modeling of turbochargers is a popular tool to provide precise and useful study criteria to investigate the performance and dynamic phenomena inside a system of engine/turbocharger. Through this research various characteristics and aspects of turbocharger has been studied numerically. The first part of the project is about the study of RIT impeller of turbocharger in steady state operational condition. A pressure-based Computational Fluid Dynamics (CFD) solver in OpenFOAM platform has been progressed and used. In this project we are analyzing the pressure acting on the turbocharger impeller turbine wheel with the help of pressure-based CFD solver in OpenFOAM. Further this pressure values acting on the turbocharger impeller turbine wheel are used for second part of the research. In the second part of the research, to explore the analysis of a turbocharger turbine wheel with design and material optimization. The study deals with structural and modal analysis, carried out considering different materials for RIT impeller wheel using Ansys software. In this project we are analyzing the pressure acting on the turbocharger impeller turbine wheel by using CFD and this pressure value is used for the FEM analysis on the three materials namely Inconel alloy 740, Inconel alloy 783 and Inconel alloy NO6230. A structural analysis is used to investigate the stresses, strains and displacements of the turbine impeller of turbocharger. A modal analysis is used to investigate the frequency and deflection of the turbine impeller. The turbine impeller of a turbocharger will be recommend based on the better material results.***
***Keywords: CFD, Open FOAM, MRF, Simple Foam, AMI, ICEM CFD, FEA, Static Structural and Modal Analysis***

## I. INTRODUCTION

In fluid dynamics, an analytical approach to solve the flow equations is hard to achieve with good precision due to the complexity. Therefore, many of the researchers use CFD (Computational Fluid Dynamics) to simulate the flow and compare to experimental data. In CFD, setting up the boundary conditions correctly is important to simulate the flow as accurate as possible. However, this is not a simple task to do. A large number of things can be changed such as the turbulence model used number of cells that make up the mesh and the solver. OpenFOAM is an open source CFD and is the modelling program used in this dissertation. This program is a "free, open source CFD software package, to solve complex fluid flows" (OpenFOAM user guide, 2018). A fundamental understanding of the flow behavior of a radial turbine is necessary to be able to understand the results from the CFD simulation. For the simulation of the radial turbine computer aided calculations are needed. The calculations are divided into two parts, structural analysis and fluid dynamics. The structural analysis is calculated with the FEA approach while the CFD method uses the Finite Volume Method (FVM) for fluid dynamics.

Simpson [2] performed a CFD analysis of existing test turbine geometries in both vaned and vaneless configurations. A total of six geometries were analysed and the results compared with measured turbine performance data. "Steady state predictions showed good agreement with the experimental trends confirming the vaneless stators to yield higher efficiencies across the full operating range" [2].According to the author, vaned stators lead to a higher level of losses because of the wake detaching from vane trailing edges, boundary layer growth and secondary flows. Spence et al. [3] tested three pairs of vaneless and corresponding vaned stators within a range of pressure ratios and flow rates. For each pair of stators the rotor was the same and the operating conditions were identical. "The vaneless volutes delivered consistent and significant efficiency advantages over the vaned stators over the complete range of pressure ratios tested. At the design operating conditions, the efficiency advantage was between 2% and 3.5%" (Spence et al, [3]). Padzillah et al. [1] compared nozzled and nozzleless turbines under pulsating flow and found that "the differences in flow angle distribution between increasing and decreasing pressure instances during pulsating flow operation is more prominent in the

nozzleless volute than in its nozzled counterpart", suggesting that the addition of a nozzle ring leads the turbine to more stable flow angle configurations at off-design points. On the other hand Baines & Lavy [7] claimed that the advantage of the vaned configuration consists of its highest peak efficiency at design point, as at off-design this efficiency drops dramatically. In three articles Whitfield et al. ([4], [5], [6]) give a comprehensive treatment of vaneless volutes for radial inflow turbines, which is taken as main reference for volute design in this thesis. Firstly they developed a practical method, suitable for implementation in a program code, based on simple hypotheses of incompressible flow and free vortex law. Since the outflow angle is specified as input parameter the aim of Whitfield's method is to design distributions of centroid radius and cross section area which ensure the flow to be delivered uniformly and at the appropriate angle to the impeller. Siggeirsson & Gunnarsson [8] report that "when the diffusion angle [(defined as the slope of the wall of the divergent pipe)] is large the diffusion rate is rapid and can cause boundary layer separation resulting in flow mixing and stagnation pressure losses. On the other hand if the diffusion rate is too low the required length of the diffuser will be very large and the fluid friction losses increase". This opinion is shared by other authors such as Dixon [9], who sets to 7° - 8° the value of the diffusion angle which gives an optimal rate of diffusion. V.R.S.M. Ajjarapu Kishore1, K.V.P.P. Chandu, D.M. Mohanthy Babu, et.al, "Design and analysis of the impeller of a turbocharger". In this study turbochargers are a class of turbo machinery intended to increase the power of internal Combustion engines. For Compressor the minimum vonmises stress (32.981 MPA) is obtained for the material Inconel alloy 909 and the maximum frequency (482.61 HZ) is obtained for the material Inconel alloy 909. For Turbine the minimum vonmises stress (171.01 MPA) is obtained for the material Inconel alloy 740 and in the frequency comparing to the compressor maximum frequency (482.61 HZ) for Inconel alloy 909. [10]
M Sai Vastav, Amrita Vishwa Vidyapeetham University, Coimbatore (2015), **"**Automotive design and analysis of turbo charger with eleven and twelve blades". This project we are designed the 3D model of the turbocharger turbine wheel by using pro-e software and the analysis taken by different materials and the analysis taken by the ansys software. This project we are analyzing the pressure acting on the turbocharger impeller turbine wheel by the three materials namely Inconel alloy 740, Inconel alloy 783 and wrought aluminium 2219. Comparing the three materials 12 plates turbine impeller Inconel alloy740 turbine wheel has the low values of total deformation and also the heat flux values high over Inconel alloy 783 and low compared to wrought aluminum 2219. Wrought aluminium 2219 has high heat flux values and deformation values. Because of its high deformation values it cannot be used for turbine. Hence it is concluded that for 12 plates turbine impeller Inconel alloy740 material can be used for turbine wheel. [11] In this work, a CFD software OpenFoam is employed considering a steady state method, for simulating the three-dimensional turbulent exhaust gas flow through a radial turbine, used in turbocharger of automobile diesel engine. The flow characteristics and performance level will be predicted. Then further pressure distribution on each blade is mapped from CFD software OpenFoam as input for structural analysis, which determines the maximum stress and maximum deformation of turbine impeller. A modal analysis is used to investigate the frequency and deflection of the turbine impeller. The turbine impeller of a turbocharger will be recommend based on the better material results.

## II. MEAN LINE DESIGN PROCEDURE

First simulations of turbocharger were launched in history with a 'hand-made' housing geometry. As there was no precise geometry references from the turbocharger producer, for handmade geometry results were obviously not enough accurate. For turbocharger applications, no data were available on the web due to confidential reasons. In this thesis mean line design is carried out to get geometric parameters of turbocharger parts then further CAD geometry has been made with the help of different software.
The design process used in this work is comprised of three steps: the meanline (one-dimensional) design, geometry generation and performance and structural analysis using CFD and FEA tools. Turbine design is an iterative process, where iterations are needed between CFD results and preliminary design results and between the FEA results and geometry details. Figure 1 summarizes the design process that is used in this work. This section briefly describes the design steps and provides details for the meanline design process, CFD and FEA setup are given in the following sub sections.

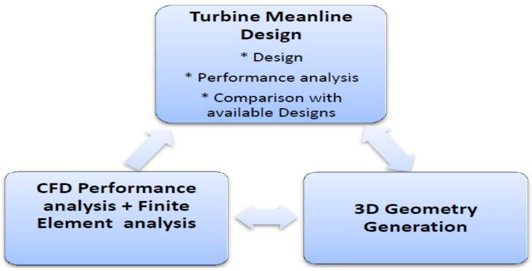



Fig. 1 Turbine Design Process diagrams. [12]

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
*ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177*
*Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com*

A mean line or one-dimensional design approach is the application of basic equations and empirical relations to calculate the overall design parameters. Mean line design is a good approach to get the first estimate for the radial turbomachine design and analysis as it is a fast method and needs a small amount of information regarding the turbomachine geometry. Also, a number of design options can be checked before moving to advanced stages of system design including three dimensional geometry construction and computational analysis software.

To start the mean line design calculation; two sets of data need to be provided to the design program. The first is taken from the turbine cycle analysis; the constraints or parameters needed for design of volute, impeller and diffuser of turbocharger. The second set comprises empirical relations that are combination of performance and dimensionless ratios available. In the next sub sections those sets of data would be discussed in more details. Based on the data gained from cycle analysis and literature, the velocity triangle stagnation and static temperature and pressure and rotor inlet diameter and blade height can be calculated which will further helps to find out geometric parameters of volute casing and diffuser part of turbocharger.

The radial turbine is a work-producing device that consists of moving and stationary parts. Figure 2 shows a schematic diagram for a radial turbine and its components. The turbine has a stationary component (stator) and rotating component. The stationary component is comprised of an annular ring called the volute and set of nozzle guide vanes. The rotating part which is the heart of the turbine called the rotor. In this work design of radial turbine is carried out without nozzle ring taken into consideration, as in case of turbocharger for reducing complexity and weight, size mostly these nozzle rings are avoided by proper volute design. Also it seen that the implementation of a fixed nozzle ring in a radial turbine does not guarantee higher efficiency through all the engine combustion cycle.

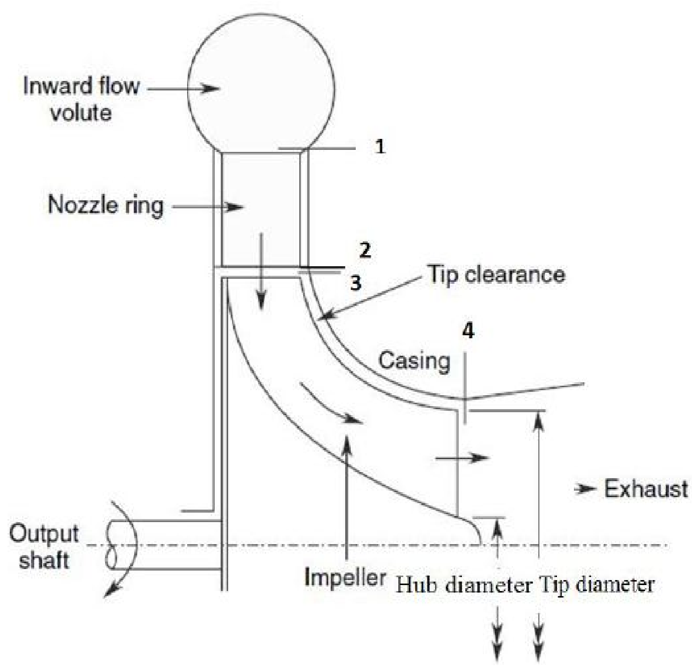



Fig. 2 Radial Turbine Components. [12]

### A. *Mean Line Design Of Volute Casing*

Volute casing design should be in such a way that to decrease the static pressure and to increase its speed, to meet desired velocity and flow angle at rotor inlet. Volute must distribute the flow uniformly along the azimuth direction and perform the energy conversion as efficiently as possible and should works with a minimum loss in stagnation pressure. The constraints on the volute design are the following (values of parameters as per engine exhaust and volute outlet consideration shown in table 1):

1) Radius at outlet ($R_2$) and passage width ($b_2$). These are set by the rotor geometry and define the volute discharge area ($A_2$).
2) Mach number ($M_2$) and absolute flow angle ($\alpha_2$) at outlet. These requirements are imposed by the designed performance of the rotor.
3) Thermodynamic flow conditions at volute inlet ($P_{01}$, $T_{01}$, $\gamma$). These parameters are set by the working point of the engine.
4) Mach number ($M_1$) at inlet, linked to the velocity of the exhaust gases.
5) Based on the turbine impeller vanes inlet diameter and width the geometry of the volute were calculated.

Table 1 Meanline design parameters for volute

| Parameter | Volute Without Nozzle Ring |
|---|---|
| Mach at volute inlet | $M_1$=0.21 |
| Mach at volute outlet | $M_2$=0.44 |
| Flow angle at volute outlet | $\alpha_2$=68 ° |
| Radius at volute outlet | $R_2$=72.27 mm |
| Outlet passage width | h=13mm |
| Flow properties at inlet | P01=2.56 bar<br>T01=846 K<br>$\dot{m}_1$=0.3322 kg/s<br>$\gamma=\gamma_{air\|T01}$=1.34<br>$\rho_1$=1.037 kg/m$^3$ |
| Additional constraints | $R_1$= 43 mm<br>$A_1$=3306.7 mm$^2$ |

On the basis of the above parameters and equations of meanline design theory MATLAB program is implemented. A volute with a circular cross section methodology was also implemented in the program. The program calculates the area and the centerline coordinate for each circular cross section of the volute at different angle starting from 0° to 360°. The volute dimension can be found in table 2. Volute geometry was generated using SolidWorks based on the preliminary design data (see table 2). Figure 3 shows the designed volute geometry.

Fig. 3 3D CAD models of the volute casing

Table 2 Final Design dimensions of the turbine volute from meanline design.

| No. | Azimuth Angle [deg.] | Radius of Center [mm] | Radius of Section [mm] |
|---|---|---|---|
| 1 | 30 | 44.76 | 1.76 |
| 2 | 60 | 46.14 | 3.14 |
| 3 | 90 | 47.62 | 4.62 |
| 4 | 120 | 49.25 | 6.25 |
| 5 | 150 | 51.36 | 8.36 |
| 6 | 180 | 53.86 | 10.86 |
| 7 | 210 | 56.46 | 13.46 |
| 8 | 240 | 59.38 | 16.38 |
| 9 | 270 | 61.96 | 18.96 |
| 10 | 300 | 64.98 | 21.98 |
| 11 | 330 | 68.91 | 25.91 |
| 12 | 360 | 72.27 | 29.27 |

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com

*B. Meanline design of Turbine Impeller*

The rotor is the main component of a radial turbine, being the only one which extracts work from the flow. Apart from the requirement of matching with the volute, which sets the dimensions of the diameter and the height of the blade at inlet, there are no constraints on the design of the rotor: the goal is to minimize all sources of losses so to achieve the maximum power output under designed flow conditions. As illustrated in the introductory chapter, nominal design condition is associated with a minimum in incidence losses. Therefore the rotor blade must be designed so that for all spanwise locations (from hub to shroud) the flow enters radially ($\beta_2$=0) and leaves the rotor axially ($\alpha_3$=0). Based on the data gained from cycle analysis and literature, the velocity triangle, stagnation and static temperature and pressure and rotor inlet diameter and blade height can be calculated by importing this data in Ansys Bladegen tool which already having meanline design equations on background. The majority of the internal rotor losses are functions of the fluid velocity, and to improve the efficiency of the rotor the velocity at rotor inlet should be reduced as much as possible to reduce the incidence loss occurring at the rotor inlet.

The turbine impeller single blades were generated using ANSYS baldegen with the dimensions as shown in table 3 and the angles were taken from the preliminary design results. Further single blade converted into symmetric 12 blade rotor geometry by using Solidworks as shown in fig. 4 (model designed for CFD).

Table 3 Dimensions of turbine impeller blade from Meanline Design

| Parameters | Value |
|---|---|
| Rotational speed (rpm) | $n$=80000 |
| Inlet tip radius | $r_2$=40 $mm$ |
| Inlet blade height | $b_2$=13 $mm$ |
| Exit tip radius | $r_{3t}$=32 $mm$ |
| Exit hub radius | $r_{3h}$=11.1 $mm$ |
| Axial length | $k_a$=23.9 $mm$ |
| Number of blades | $Z_b$=12 |

Figure 5 shows an actual 3D model of turbine impeller and which required for FEA, applying the boundary conditions from CFD analysis as well as material properties in FEA it will be simpler to suggest best material for turbine impeller amongst the other materials used for FEA.

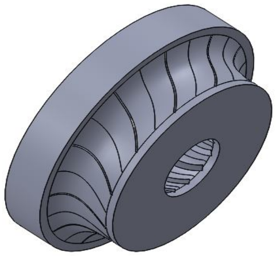


Fig. 4 CFD Model of Turbine Impeller of Turbocharger

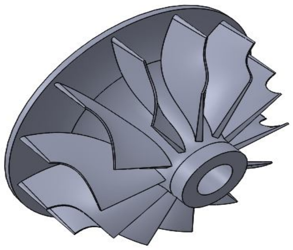


Fig. 5 Turbine Impeller/ Rotor 3D Model For FEM Analysis

*C. Meanline design of Diffuser*

The diffuser must convert part of the kinetic energy of the flow into pressure in order to reach the condition $P_4>P_{atm}$ at the outlet of the turbine and allow the flow to be discharged. The constraints on the design of the diffuser are

*1)* Inner radius ($r_{3h}$) at diffuser inlet, which must be equal to the hub radius at rotor outlet.
*2)* Outer radius ($r_{3t}$) at diffuser inlet, which must be equal to the shroud radius at rotor outlet.

As per literature review for design diffuser optimal rate of diffusion required which obtained by 7° - 8° the value of the diffusion angle. Also length of diffuser should be within range to decrease friction losses. From the meanline design equations given in the literature and outlet parameters of turbine impeller the dimensions of diffuser are shown in the table 4 and 3D model for diffuser made by using Solidworks shown in fig. 6.

Table 4 Dimensios Of Diffuser From Meanline Design

| Parameters | Value |
|---|---|
| Outer Radius At Diffuser Outlet | $r_{4t}$=35 Mm |
| Inner Radius At Diffuser Outlet | $r_{4h}$=11.1 Mm |
| Axial Length | $l$=70 Mm |

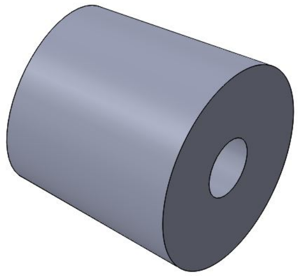


Fig. 6 3D CAD model of Diffuser used for CFD

The pressure inlet specification is unreliable when the inlet boundary is too much closed to the turbine wheel, region in which the fluid has more vorticity and difficult to be solved. Therefore to avoid more vorticity and to form continuity in the flow at inlet and outlet, when there is total pressure inlet, static pressure outlet conditions are used for CFD the inlet and outlet of the geometry should be to farthest from impeller wheel inlet and outlet respectfully as shown in fig. 7

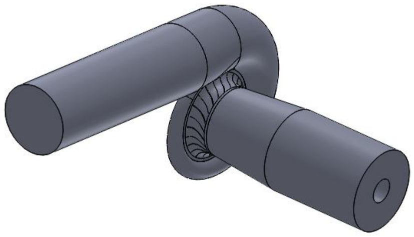


Fig. 7 Complete Assemble 3D CAD CFD Model Of Turbocharger Without Nozzle Ring

## III.MESHING AND BOUNDARY CONDITIONS

In this thesis, Mesh generating is one of the most important steps to simulate CFD problems. Fundamentally, CFD simulations are affected by the quality of the mesh. A high-quality mesh makes CFD simulations accurately and converges quickly whereas the poor quality of the mesh can approach poor results. In order to generate mesh in this work, we used ICEM CFD software to create mesh for CFD, especially for complex geometries like the geometry consisting multiple parts like turbocharger assemblies, propeller. After generating mesh in ICEM CFD, it will be exported to OpenFOAM that makes a folder called polyMesh that defines the geometries and characteristics of the grids. More specifically, after exporting the mesh to OpenFOAM and creating the case, to check the validity of the mesh, we can use "checkMesh'' command. It gives required information about the quality of the mesh such as number of the cells, domain size, the boundary conditions and mesh error and, etc. As this problem solved in OpenFoam, turbocharger splitted into three separate zones (stationary and rotary zones) for consideration of multiple reference zone (MRF) method. While solving this problem Arbitrary Mesh Interface (AMI) technique used in this project. Meshing of these three different zones was carried out separately to make complete conformal patch interfaces between each other. After that separate part wise meshing merged into one project of ICEM to create volume meshing on complete assembly of turbocharger used for CFD. Complete mesh consisting three differ zones with conformal patches between each other as shown below fig. 8 and fig. 9

1) *Volute:* Stationary zone
2) *Impeller Wheel:* rotary zone
3) *Diffuser:* Stationary zone

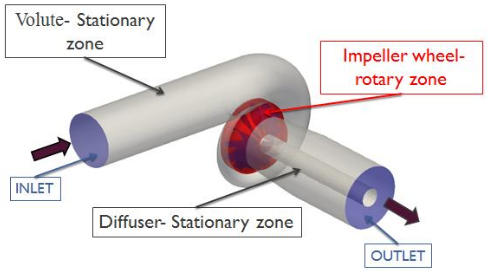



Fig. 8 Turbocharger assembly splitted into stationary and rotary zones

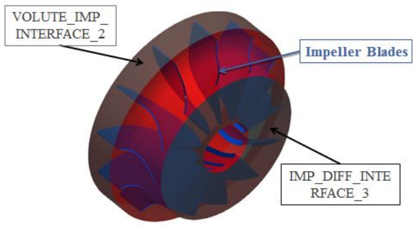



Fig. 9 Rotary zone turbine impeller of turbocharger

### A. *Volume Mesh Of Turbocharger And It's Quality*

The mesh was done using ANSYS® ICEM CFD, according to solver the overall mesh quality which includes the effect of several factors such as smoothness, Aspect ratio, cell shapes, node density, determinant etc. Mesh size should be in such way that it will provide high accuracy with minimum computation power and time for solution completion. Table 5 shows the total mesh grid size for CFD analysis using OpenFoam and fig. 10 shows the complete volume meshing of turbocharger assembly for CFD and cut-sections of volume meshing at mid plane of volute and diffuser respectively. Fig. 11 depicts the overall mesh quality for various grid configurations. The mesh quality reduces as the grid elements increase. The volumetric mesh quality should be above 0.2. Mesh should be made as equilateral as possible and the difference in the shape of the mesh from its equilateral form is defined by its skewness. Universally 0.95 is the maximum allowable skewness ratio and the average skewness can be up to 0.33.

Table 5 Grid Size for CFD analysis

| Element Type | Volute Mesh | Impeller Mesh | Diffuser Mesh | Total Elements |
|---|---|---|---|---|
| Tetra_4 | 5, 50, 407 | 17, 50, 007 | 95, 622 | 23, 96, 036 |
| Tri_3 | 65, 784 | 2, 00, 906 | 13, 770 | 2, 80, 460 |
| Total Nodes | | 4,72,310 | | 26, 76, 496 |

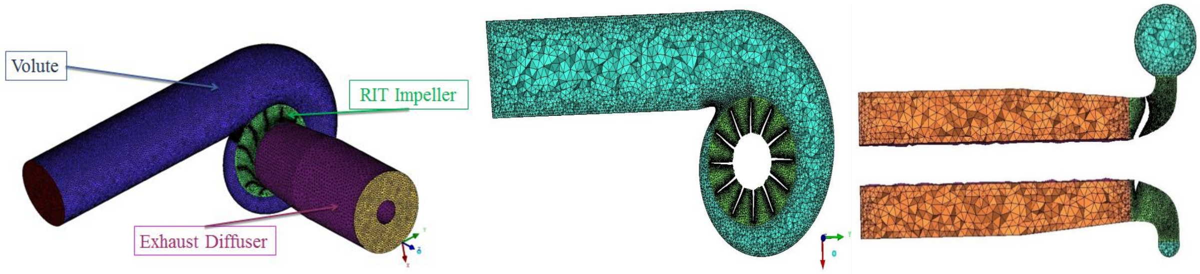



Fig. 10 Complete Volume Meshing Of Turbocharger And Its Cut-Sections

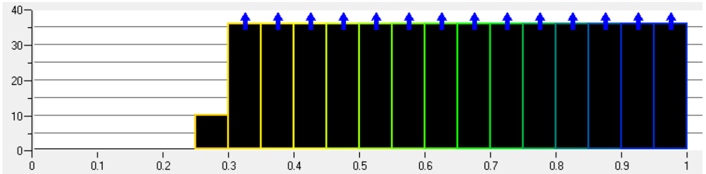



Fig. 11 overall mesh quality

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com

FEM analysis is conducted by using Ansys® software, in which meshing of RIT impeller wheel generated in Ansys® Mechanical. As in case of FEM multiple simulations are carried out for different material applications the mesh size should be in such way that it will provide high accuracy with minimum computation power and time for solution completion. On the basis of this impeller is divided into 13 parts, 12 blades and impeller hub and by providing bonded contact between hub and blades it will be simpler to get high quality mesh with optimum size. Hexahedral meshing is generated on blades and tetrahedral mesh is generated on impeller hub body, possible just because of separation of blades from impeller hub by using Boolean operation. Advantage of more number of hexahedral elements than tetrahedral elements is less number of elements generated which takes minimum time and minimum computational power, provides more accurate results. Fig. 12 shows the RIT impeller wheel meshing for FEM. Due to hexahedral mesh this meshing also shows the mesh quality above 0.3.

Element Types:

1. Hexahedral Element 2. Tetrahedral Element

Total Number of Elements: 4, 75,020 Total Number of Nodes: 13, 91,833

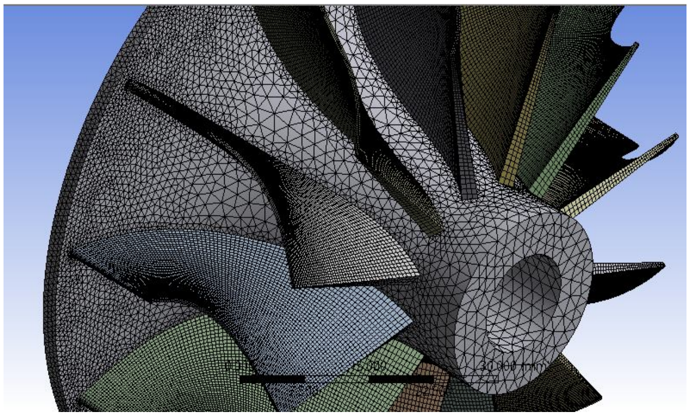


Fig.12 RIT Impeller Wheel Meshing For FEM

*B. Boundary Conditions used for CFD in OpenFoam*

This study uses OpenFOAM Open source CFD tool to compute the incompressible flow field around the RIT impeller wheel. OpenFOAM (Open source Field Operation and Manipulation) is a free, open source computational fluid dynamic (CFD) software package that provides different solvers, libraries and different utilities for various CFD problems released by OpenCFD. The initial purpose of OpenFOAM was to develop a more robust and flexible simulation platform than FORTRAN. $C^{++}$ as a programming language was a good choice because its highest modularity and being object oriented.

In this study SimpleFoam used for MRF technique. SimpleFoam is a steady-state solver for incompressible fluids with MRF regions. As sliding grid technique is more common two solve steady state as well as transient or dynamic nature problems, AMI method of sliding grid technique for MRF has been used in this work.In OpenFoam for MRF approach fluid flow is computed using different reference frames. Rotating zone is solved in the rotating frame and stationary zone is solved in the stationary frame. This approach accounts the rotation of the rotor without having to physically rotate any part of the mesh. The approach is also called the frozen rotor technique. In MRF approach cells in the rotating region should be kept in a separate cellZones. In this case mesh is generated in two regions and merged together using mergeMeshes utility. In this case AMI (Arbitrary Mesh Interface) is used. AMI operates by projecting one of the patch faces on the other patch face, on the basis of contact between two patched surfaces weighting factors will be calculated, to determine how an AMI boundary cell should couple to the AMI boundary cells on the other side of the interface. These weighting factors should be close to 1 always. Following lines of programme while running SimpleFoam solvers, shows weighing factor is equal to 1 means AMI Interface shows that stationary and rotating patches are perfectly conformal for MRF condition.

AMI: Creating addressing and weights between 3142 source faces and 3142 target faces
AMI: Patch source sum(weights) min/max/average = 1, 1, 1
AMI: Patch target sum(weights) min/max/average = 1, 1, 1
AMI: Creating addressing and weights between 2224 source faces and 2224 target faces
AMI: Patch source sum(weights) min/max/average = 1, 1, 1
AMI: Patch target sum(weights) min/max/average = 1, 1, 1

*ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177*
*Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com*

For run the simulation following commands are used,

```
Fluent3DMeshToFoam                         // import mesh into solver
splitMeshRegions -cellZones –overwrite     // to split stationary and rotary zones from each other
checkMesh                                  // validity of mesh
decomposePar                               // partition of mesh for parallel computing
mpirun -np 3 simpleFoam -parallel > & log.simpleFoam &
gnuPlot residue.txt                        //To observe convergence
paraview                                   // Post processing tool
```

The boundary conditions on the CFD fluid model consisted of a total pressure of exhaust gases at the inlet, perpendicular to the inlet in all studied cases and static pressure at outlet of fluid domain. Also, cyclic AMI conditions are used for conformal interface patches between volute-impeller and impeller-diffuser shown in fig. 8, 9. No slip is also used when the wall facing viscous flow. In boundary conditions walls* notation is used for the entire all those are stationary other than inlet, outlet and interfaces. Similarly rotatingwall* notation is used for all the walls of rotary zone-impeller and interfaces between two zones.

Table 6 Boundary Conditions

| Patch | Type | U | p |
|---|---|---|---|
| Inlet | Patch | fixed Value uniform (0 0 0) | Fixed Value uniform 256360 |
| Outlet | Patch | fixed Value uniform (0 0 0) | Fixed Value uniform 112770 |
| Walls* | Wall | No slip | zeroGradient |
| Rotatingwall* | Symmetry | No slip | zeroGradient |
| Ami | interface | cyclicAMI | cyclicAMI |

Table 7 Operating Conditions of Turbocharger

| Exhaust Gas (for T = 575°C) | $\rho$ | Kg/m$^3$ | 0.548 |
|---|---|---|---|
| kinematic viscosity | $\nu$ | m$^2$/s | 1E-06 |
| Angular velocity | $\omega$ | rad/s | 8377.58 |
| Total pressure at inlet | $P_t$ | Pa | 256360 |
| Static pressure at outlet | $P_s$ | Pa | 112770 |

### *C. Boundary Conditions used for Finite Element Analysis*

A Finite Element Analysis was conducted in order to assess the stress distribution for the turbine wheel. The FEA was performed using ANSYS Workbench. A static structural analysis was carried out. This is necessary because of the high rotational speed and high flow temperature experienced by the impeller. The material alloy Inconel was selected because of its high-quality mechanical properties under extreme temperatures. In this study different grades of Inconel alloy like Inconel alloy 740, Inconel alloy 783 and Inconel alloy N06230 was used to carry out FEM analysis and on the basis of FEM it will be simple to suggest better material for radial turbine impeller of turbocharger. Therefore while carried out FEM analysis some initial boundary conditions are required like material properties, geometry constraints and magnitude of load applied as shown in fig. 13 and table 8. To ensure safe behaviour under working conditions, the Von Mises yield criterion needs to be satisfied. This criterion states that a material is said to start yielding when its Von Mises stress reaches a critical value known as the Yield strength. The Yield strength is defined as the stress at which a material begins to deform plastically.

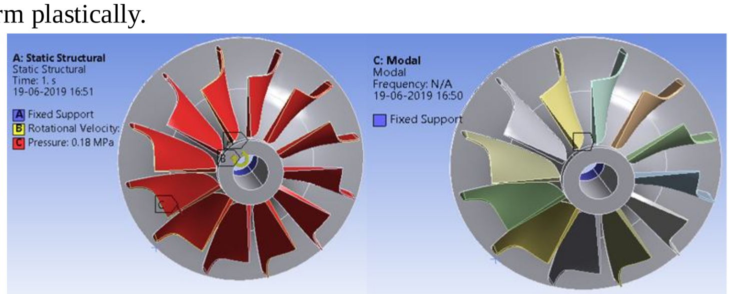



Fig. 13 Boundary conditions applied on RIT impeller for FEM analysis.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com

Table 8 Material properties for FEM analysis

| Material | Density (ρ) In $Kg/m^3$ | Young's Modulus(E) in MPa | Poisson Ratio(ν) (Dimensionless) | Thermal Conductivity(K) in $W/m^0C$ |
|---|---|---|---|---|
| Inconel Alloy 740 | 8050 | 218000 | 0.32 | 10.2 |
| Inconel Alloy 783 | 8220 | 158579 | 0.291 | 10.1 |
| Inconel Alloy NO6230 | 8910 | 211500 | 0.34 | 12.3 |

## IV. CFD AND FEM RESULTS

In this chapter the results from the simulations are presented, and the outputs in terms of pressure variation and axial pressure load acting on the blades of impeller used as input to finite element analysis. First, the CFD results for turbocharger in different advance numbers for AMI model are investigated. Then results of CFD are used as inlet conditions for FEM analysis, for better material recommendation structural as well as modal analysis is carried out on the Ansys® Workbench.

### *A. CFD Results*

In this project total pressure inlet and static pressure outlet boundary conditions are used and for these boundary conditions rotary velocity of impeller is taken. Results can get through streamlines, vector plot and velocity and pressure contour. The assumptions consider while carry the CFD simulation are as follows:

1) The walls of the casing were assumed to be smooth hence any disturbances in flow due to roughness of the surface were neglected.
2) The friction co-efficient for all surfaces were set to 0, hence friction between the walls and fluid was neglected.
3) Steady state conditions and incompressible fluid flow.

Solution parameter used while carries the CFD simulation is as follows:

a) 3-D double precision solver used to solve for simulation.
b) Multiple reference frame technique used to simulate the turbine performance.
c) Exhaust gas (air having high temperature properties) is taken as working fluid.
d) The OpenFOAM toolbox already provides a solver called simpleFoam is used for solving the steady-state Reynolds-Averaged Navier-Stokes equations with turbulence Standard K-omega (SST) modeling.
e) Convergence criteria for continuity, velocity and turbulence parameters were set to $10^{-04}$.
f) Second order scheme is used for pressure correction as well as for solving momentum, turbulent kinetic energy and turbulence dissipation rate.
g) SIMPLEC scheme is used for pressure velocity coupling.

Pressure and Velocity distribution in turbocharger turbine impeller at 80000 rpm rotation are as shown in the following fig. 14, 15, and 16.

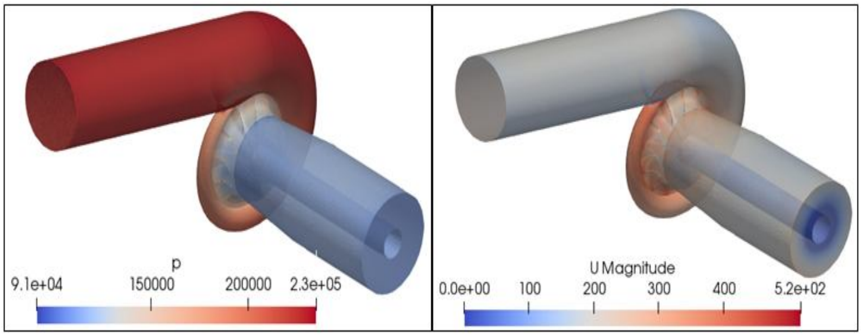



Fig. 14 Pressure and Velocity distribution in turbocharger

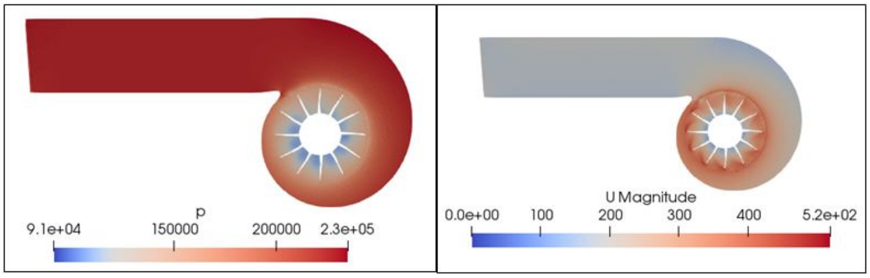



Fig. 15 Pressure and Velocity distribution at vertical mid cut plane of volute

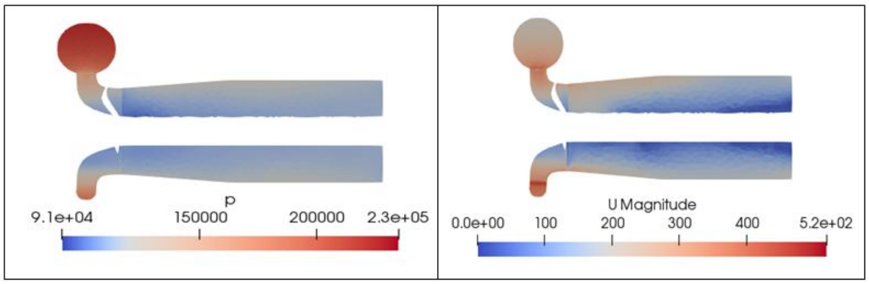



Fig. 16 Pressure and Velocity distribution at X-axis cut plane of turbocharger

From the above CFD results it concludes that pressure drop from inlet to outlet is varies between total pressure at inlet to static pressure at outlet as per applied boundary conditions, and the axial thrust or pressure force acting on impeller blades varies as shown in fig. 17. The final converged axial force or pressure force acting on impeller blades lies between 406-411 Newton. As isothermal expansion inside the turbine is not possible practically, isentropic expansion of gases takes place inside the turbine means there is pressure as well as temperature loss takes place and which is responsible for final work output. In practical cases minimum temperature loss takes place in turbocharger radial inflow turbine approximately is $60^0$C-$70^0$C. Therefore, the maximum total-static isentropic efficiency of the turbine $\eta_T$ as per the standard equation is 66% and which lies between the general efficiencies values 65 and 70 % at the design point of the turbine performance map [8–10].

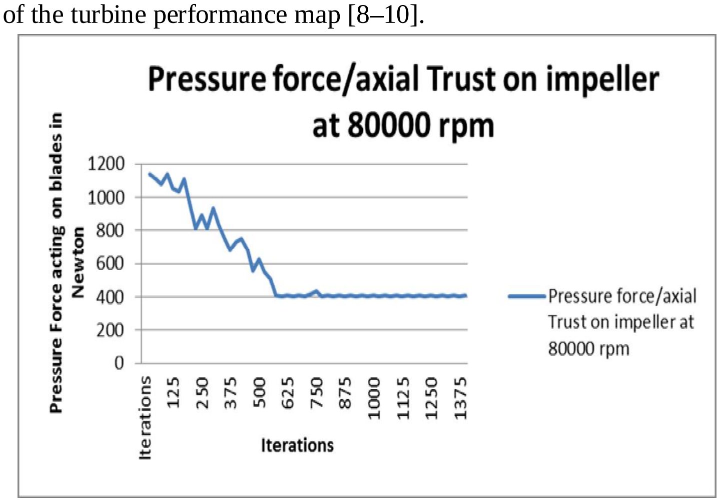



Fig. 17 Pressure force/axial Trust acting on impeller blades at 80000 rpm

### *B. FEM Results*

Modal analysis results are achieved through analysing the first three mode deformations and natural frequencies. Therefore by comparing results between three materials, best material always lies with higher natural frequencies and lesser deformation for all three modes. In structure analysis results are achieved through analysing the von misses stress, von misses strain and total deformation. The validation of structural results lies on the basis criteria of material is safe; if the maximum von misses stress on the impeller is less than the yield strength of the material then the material used for turbine impeller is safe and the material which shows lesser value of maximum von misses stress among the three materials will be the best recommended material for radial turbine impeller of turbocharger. Assumptions used while carried out structural analysis are as follows:

*1)* Number of r/m is constant that is 80000 rpm.
*2)* Linear analysis is being done.
*3)* Static structure analysis is being done.

Structural and Modal Analysis Results for Turbine impeller of Inconel alloy 740 is as shown in the following figure 18, 19 and 20. Structural and Modal analysis results for other material are directly shown in tabular and graphical form.

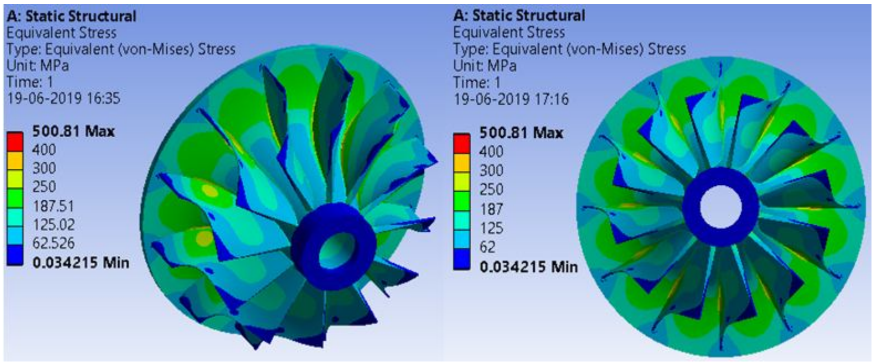



Fig.18 Von misses stresses acting on turbine impeller of Inconel alloy 740

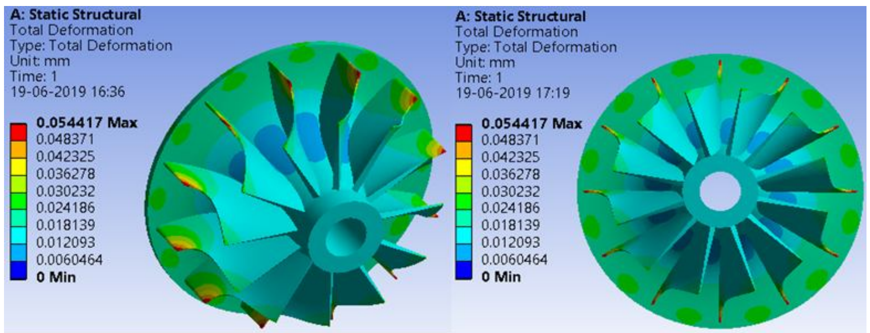



Fig. 19 Total deformation of turbine impeller of Inconel alloy 740

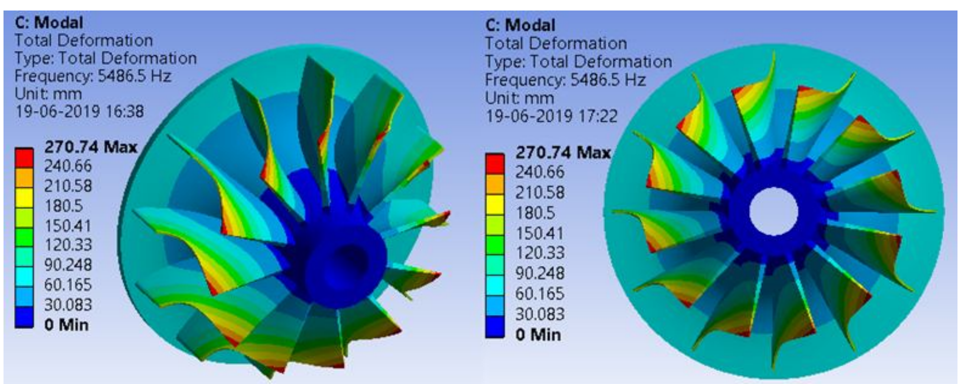



Fig. 20 Total deformation and natural frequency at 3rd mode of turbine impeller of Inconel alloy 740

The Comparison of von misses stresses with respect to turbine materials, it can be seen that the maximum von misses stresses are induced in Inconel alloy NO6232, when compared with Inconel alloy 740 and Inconel alloy 783. Where a maximum value of von misses stresses 563.23Mpa was noticed for Inconel alloy NO6232 and minimum value 500.81Mpa was noticed for Inconel alloy 740.

The comparison of displacement with respect to turbine materials, it can be seen that the maximum displacement are induced in Inconel alloy 783, when compared with Inconel alloy 740 and Inconel alloy NO6230. Where a maximum value of displacement 0.07580 mm was noticed for Inconel alloy 783 and minimum value 0.054 mm was noticed for Inconel alloy 740. Following table 9 comparing the maximum von misses stress; maximum von misses strain and maximum displacement of turbine impeller for different three materials.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com

Table 9 Structural analysis for turbine impeller

| Properties | Inconel alloy 740 | Inconel alloy 783 | Inconel alloy N06230 |
|---|---|---|---|
| Max. von mises stress (N/mm2) | 500.81 | 560.07 | 563.23 |
| Max. Displacement (mm) | 0.054 | 0.07580 | 0.06270 |
| Max. von mises strain (mm/mm) | 0.002824 | 0.004373 | 0.003316 |

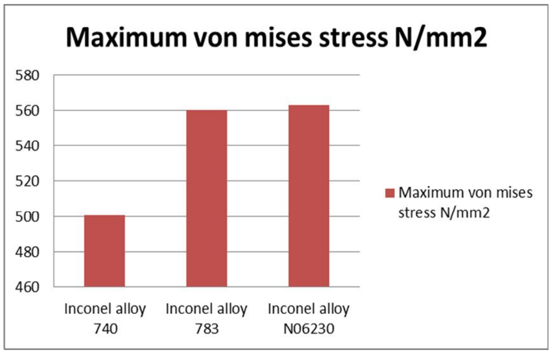



Fig. 21 Stress distribution vs. materials

Comparison of frequency deflection with respect to compressor materials, it can be seen that the maximum natural frequency and minimum deflection is obtained in Inconel alloy 740 when compared with Inconel alloy 783 and Inconel alloy NO6230. Following table 10 and bar graph shows the comparative analysis between the three materials for natural frequency deflections up to 3 mode shapes obtained in turbine impeller.

Table 10 Modal analysis for turbine impeller

| Type of Material | Frequency (Hz) | | | Deflection(mm) | | |
|---|---|---|---|---|---|---|
| | 1 | 2 | 3 | 1 | 2 | 3 |
| Inconel alloy 740 | 4900 | 4901.1 | 5486.5 | 201.02 | 203.07 | 270.74 |
| Inconel alloy 783 | 4114.4 | 4119.9 | 4664.7 | 203.09 | 205 | 285 |
| Inconel alloy N06230 | 4556.9 | 4557.9 | 5113.6 | 191.79 | 193.73 | 260.2 |

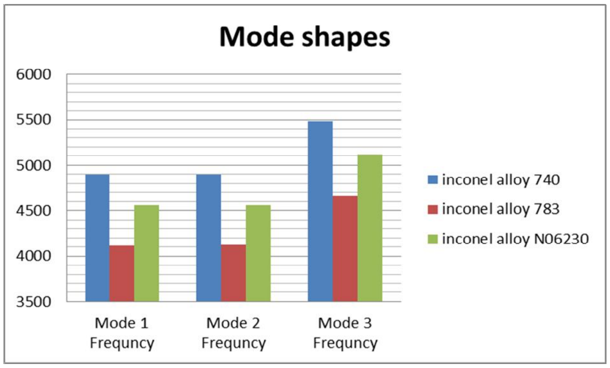



Fig. 22 Natural frequency vs. Materials

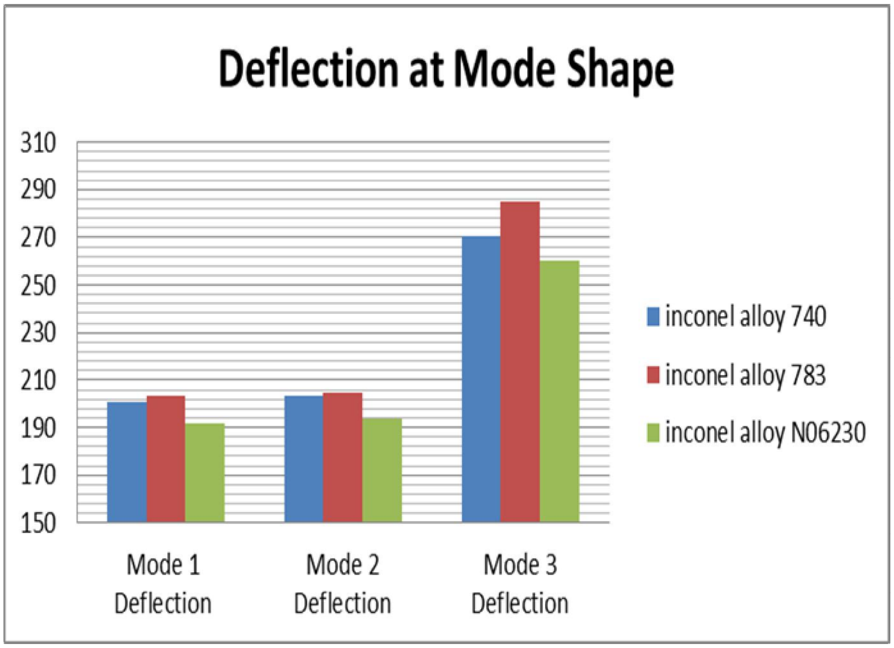



Fig. 23 Mode shapes deflection vs. Materials

## V. CONCLUSIONS

The main purpose of this thesis was to investigate the performance of rotating machinery and approach the radial inflow turbine performance through numerical modelling. We have investigated the feasibility study on the MRF technique, to predict the flow of exhaust gas around the radial inflow turbine of turbocharger. From CFD results it concludes that pressure drop from inlet to outlet is varies between total pressure at inlet to static pressure at outlet as per applied boundary conditions, and the axial thrust or pressure force acting on impeller blades decreases as the convergence occur. Therefore, on the basis of CFD results maximum application of pressure load as well as maximum pressure acting on blades is obtained and which required as input condition while simulating FEM. As per the CFD results the maximum total-static isentropic efficiency $\eta_T$ of the turbocharger turbine is 66% and which lies between the general efficiencies values 65 and 70 % at the design point of the turbine performance map and also axial thrust or pressure force acting on the blades varies between 406 newton to 411 newton.

The main conclusions that can be drawn about the CFD method, those are as follow:

*A.* CFD method has the very highest computational cost
*B.* Might have long setup times
*C.* Gives the possibility to visualize problem areas.

Second part of this thesis FEM analysis concludes that Inconel alloy 740 material is best material recommendation for radial inflow turbine blades among the all three materials as it shows higher final factor of safety (FOS $\approx$ 1.5) than other two materials. Also for Turbine impeller the minimum von mises stress and maximum frequency is obtained for the material Inconel alloy 740. So we conclude that among these three different materials Inconel alloy 740 is the best material.

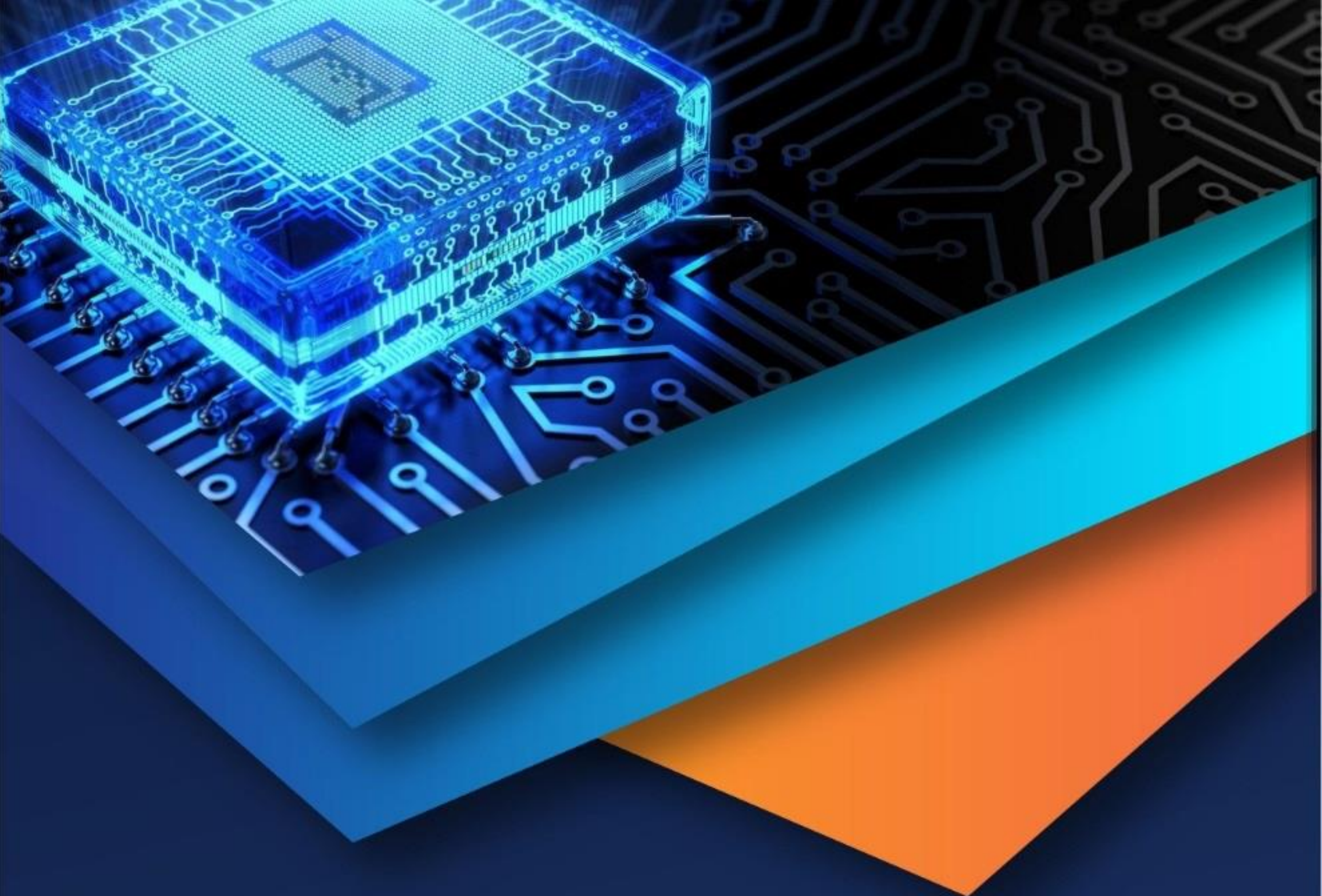
## VI. FUTURE WORK

Inputs for possible future works are listed below:

*A.* Results presented in this thesis solely rely on a numerical model, and numerical methods have some uncertainties or losses and Limitations. In order to assess the validity of the results the latter should be compared with experimental data.
*B.* In this study only steady conditions have been considered, and it is implicitly assumed that the flow has enough time to adapt to variations of pressure and temperature in the exhaust gases. This may not be true in general, especially when the engine operates at high rpm, hence a further study should model the unsteadiness of the flow.

## REFERENCES

[1] M. H. Padzillah, M. Yang, W. Zhuge, R. F. Martinez-Botas, "Numerical and Experimental Investigation of Pulsating Flow Effect on a Nozzled and Nozzleless Mixed Flow Turbine for an Automotive Turbocharger", ASME Turbo Expo 2014: Turbine Technical Conference and Exposition, 2014.
[2] A. T. Simpson, "Aerodynamic Investigation of Different Stator Designs for a Radial Inflow Turbine", Queen's University Belfast, 2007.
[3] S. W. T. Spence, R. S. E. Rosborough, D. W. Artt, G. Mccullough, "A Direct Performance Comparison of Vaned and Vaneless Stators for Radial Turbines", ASME Journal of Turbomachinery, Vol. 129, 2007
[4] S. A. MacGregor, A. Whitfield, A. B. Mohd Noor, "Design and Performance of Vaneless Volutes for Radial Inflow Turbines. Part 3: Experimental Investigation of the Internal Flow Power and Energy, June 1994
[5] A. Whitfield, A. B. Mohd Noor, "Design and Performance of Vaneless Volutes for Radial Inflow Turbines. Part1:Non- Dimensional Conceptual Design Considerations, Proceedings of the Institution of Mechanical Engineers Part A Journal of Power and Structure", Proceedings of the Institution of Mechanical Engineers Part A Journal of Energy, June 1994
[6] A. Whitfield, S. A. MacGregor, A. B. Mohd Noor, "Design and Performance of Vaneless Volutes for Radial Inflow Turbines. Part 2: Experimental Investigation of the Mean Line Performance – Assessment of Empirical Design Parameters", Proceedings of the Institution of Mechanical Engineers Part A Journal of Power and Energy, June 1994
[7] N. C. Baines, M. Lavy, "Flow in Vaned and Vaneless Stators of Radial Inflow Turbocharger Turbines", Proceedings Institution Mechanical Engineers, Turbochargers and Turbocharging Conference, 1969
[8] E. M. V. Siggeirsson, S. Gunnarsson, "Conceptual Design Tool for Radial Turbines", MSc Thesis, Chalmers University of Technology, Department of Applied Mechanics, Gothenburg, 2015
[9] S. L. Dixon, C. A. Hall, "Fluid Mechanics and Thermodynamics of Turbomachinery", 6th Edition, Butterworth-Heinemann, 2010
[10] Ajjarapu Kishore, V.R.S.M., K.V.P.P. Chand, D.M. Mohanthy, Babuet.al, "Design and analysis of the impeller of a turbocharger", pp. 257-266, 1998.
[11] M Sai Vastav, Amrita Vishwa Vidyapeetham University, Coimbatore (2015), "Automotive design and analysis of turbo charger with eleven and twelve blades". International Journal of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 3, Issue-11, Nov.-2015.
[12] Mahmoud Khader, "Optimised Radial Turbine Design D1.8", School of Mathematics, Computer Science and Engineering, London, October 2014



10.22214/IJRASET

INDEX COPERNICUS

45.98

IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY