Impact of Material Selection on the Vibration and Structural Integrity of Centrifugal Pump Impeller Using FEA

M.S.Rao[1], Vasupalli Raghuram[2]

[1]*Assistant Professor, Dept. of Mechanical, Sanketika Vidya Parishad Engineering College, Vizag, A.P, India*
[2]*M.TECH (final year),Dept. of Mechanical, Sanketika Vidya Parishad Engineering College, Vizag, A.P, India*

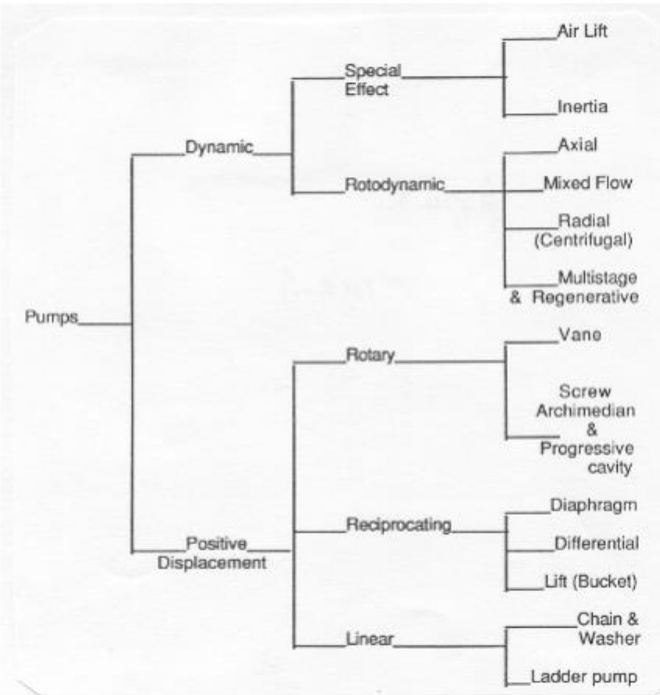
---***---

Abstract –
In this study, vibration and fluid-structure interface analyses were conducted on a centrifugal pump impeller using Finite Element Analysis (FEA). The impeller, a key component in centrifugal pumps, is responsible for raising the pressure head of the fluid. The 3D model of the impeller was initially obtained from KUMAR PUMPS.Industries and then created in CATIA software. This model served as the foundation for performing simulations in ANSYS software, including Computational Fluid Dynamics (CFD), structural analysis, and vibration analysis. These analyses were critical in understanding how different materials perform under operational conditions and their ability to withstand forces generated during pump operation. The results from these simulations helped in evaluating the overall behavior of the impeller, providing insights into how the material choice impacts both fluid dynamics and structural integrity.Five different materials were selected for the impeller to assess their load-bearing capabilities, cost efficiency, and strength. Vibration analysis, in particular, is a key technique for monitoring the performance of mechanical components, as it helps in identifying potential issues such as fractures or fatigue failure. In vibration analysis,
frequency is a critical factor, as it is directly related to the material. stiffness the higher the frequency, the stiffer the material. Therefore, the goal is to improve the materials stiffness and subsequently increase its frequency to enhance performance and prevent failure. Based on the results of the vibration and structural analysis, material selection was guided by the ability to achieve optimal stiffness and frequency, ensuring that the impeller can efficiently handle the forces it experiences without undergoing structural damage or fracture.

Keywords: Centrifugal pump, Impeller design, Vibration analysis, Fluid-structure interaction and Material selection

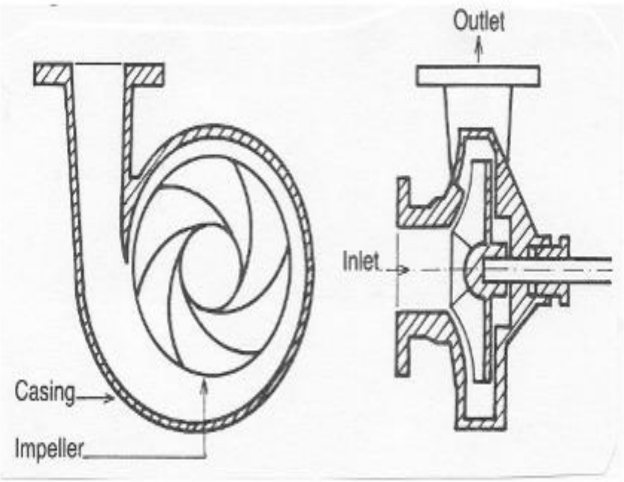
1. INTRODUCTION

Water pumps can be divided into three types: displacement, impulse and other types. Positive displacement pumps can be of reciprocating and rotary type. In either case liquid is displaced from the low pressure suction side to the high pressure discharge side (the term positive refers to the direction of flow displacement related to the pressure gradient). The geometry of the pump is changed periodically and determines the flow in both supply and delivery system. In a positive displacement pump there is no direct communication between the suction and discharge circuit. As a rule, a positive displacement pump is self-priming



Centrifugal Pumps

Centrifugal (or rotodynamic) pumps are based on the principle of imparting kinetic energy to the water. In these pumps water enters axially and is discharged by the rotor into a discharge pipe. They have an impeller which rotates in a casing of a special shape. The impeller vanes accelerate the water, which is thrown out by the centrifugal force. The shape of the casing is designed to effectively build up a high pressure at the pump outlet. It is this pressure level that lifts the water against the pumping head. In Figure 5, a single stage of a centrifugal pump is shown. This type of pumps are typically driven by an electric motor or combustion engine and installed above ground level.



Impeller and casing of a centrifugal pump stage.

Computer numerical controlled (CNC) lathes are rapidly replacing the older production lathes (multispindle, etc.) due to their ease of setting, operation, repeatability and accuracy. They are designed to use modern carbide tooling and fully use modern processes. The part may be designed and the tool paths programmed by the CAD/CAM process or manually by the programmer, and the resulting file uploaded to the machine, and once set and trialled the machine will continue to turn out parts under the occasional supervision of an operator.

The machine is controlled electronically via a computer menu style interface, the program may be modified and displayed at the machine, along with a simulated view of the process. The setter/operator needs a high level of skill to perform the 7 process, however the knowledge base is broader compared to the older production machines where intimate knowledge of each machine was considered essential. These machines are often set and operated by the same person, where the operator will supervise a small number of machines (cell).

OBJECTIVE OF THE WORK:

The objective of the present work is to determine the optimum cutting parameters and also which factor has major influence on C45 steel during CNC Turning. RSM is used to determine the effect of factors on each response variable and to create a model for each response variable. Further fuzzy goal programming is used to convert a multiobjectve optimization problem to a single objective optimization problem. Finally genetic algorithm is used to optimize the values of response variables and determine the corresponding values of design variables.

OBJECTIVES:

1. Gather dimensions from a real-time impeller model and develop a detailed 3D geometry for accurate visualization and drafting. This will serve as the foundation for further simulations and analysis.

2. Select different materials based on cost, strength, and performance criteria. Perform simulations to evaluate their behavior under operational conditions like hydrodynamic loading and centrifugal forces.

3. Conduct vibration analysis to assess the dynamic behavior of the impeller at different frequencies. Identify the resonance frequencies and ensure that the material and design avoid critical resonant conditions.

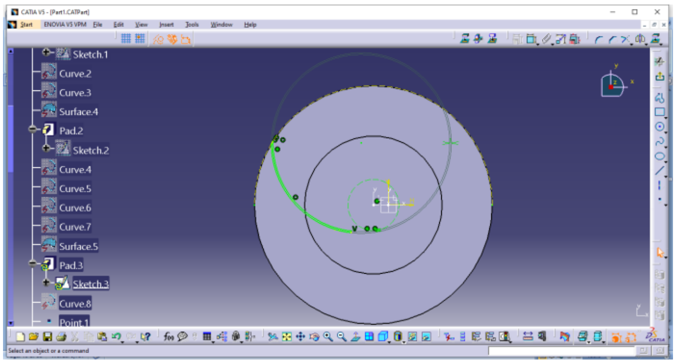
4. Perform fatigue analysis to evaluate the impeller's durability under cyclic loading. Determine the materials' resistance to fatigue and their ability to withstand long-term operational stresses.

5. Compare the results from vibration, fatigue, and structural analyses to identify the optimal material and impeller model. Select the best-performing design to ensure efficiency, safety, and longevity under operational conditions.

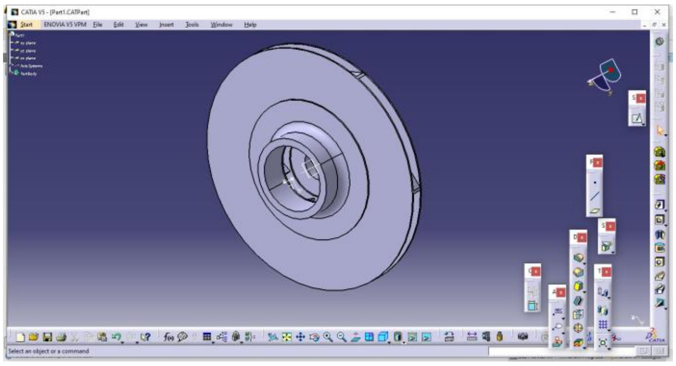
2.COMPUTER AIDED MODELING:

CFD (Computational Fluid Dynamics) is a branch of fluid mechanics that uses numerical methods and algorithms to analyze and simulate fluid flow, heat transfer, and related phenomena within a system. In CFD, engineers can model the behavior of fluids—such as air or water—through complex geometries and operating conditions, such as turbulence, pressure drops, and thermal effects. This simulation technique is widely used in industries for optimizing designs, such as in heat exchangers, pumps, or combustion systems, as it helps predict performance without the need for costly physical prototypes. By visualizing fluid flow patterns and temperature distributions, CFD enables engineers to identify inefficiencies and refine designs to improve performance, reduce energy consumption, and enhance overall system efficiency.

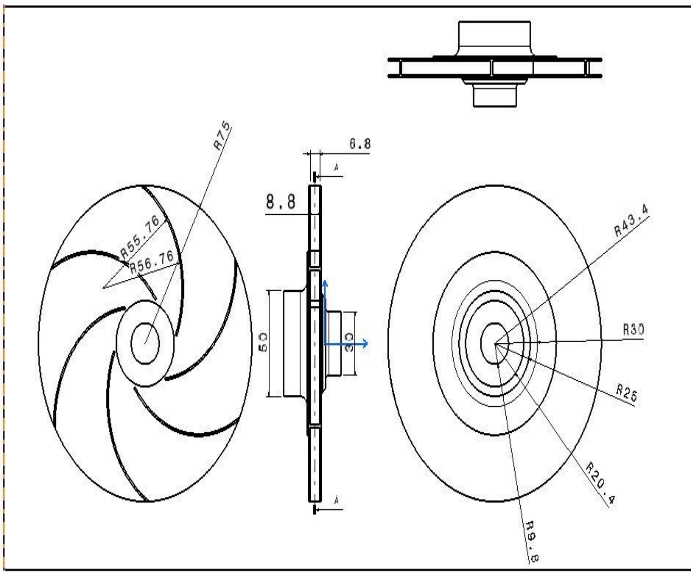
CATIA Software Overview:



blade cross section created in Catia



3D model of impeller



Detail view of impeller.

Parameter	value in mm
Impeller outer Radius	75
Hub inner radius	9.8
hub Outer radius	20.4
Suction inner radius	25
Suction outer radius	30
Wane inner radius	55.76
Wane outer radius	56.76
Thickess	1
Total width	8.8

Table: design parameters.

3.ANALYSIS OF IMPELLER:

Navier-Stokes Equation:

The Navier-Stokes equations are fundamental to fluid dynamics and describe the motion of viscous fluid substances. These equations are a set of nonlinear partial differential equations that express the balance of momentum in a fluid flow. They are derived from the principles of conservation of mass, momentum, and energy. The Navier-Stokes equation can be written as:

$$\rho\left(\frac{\partial u}{\partial t} + u.\,\nabla u\right) = -\nabla p + \mu\,\nabla^2\,u + f$$

Energy Equation:

The energy equation in fluid dynamics describes how thermal energy is transported and transformed in a fluid flow. It accounts for conduction, convection, and possibly even radiation heat transfer. The energy equation is based on the first law of thermodynamics and can be expressed as:

$$\rho\left(\frac{\partial h}{\partial t} + u.\,\nabla h\right) = \nabla.(k\,\nabla\,T) + \Phi + Q$$

Pre-Processing in CFD

The methodology in Computational Fluid Dynamics (CFD) begins with the pre-processing phase, which involves setting up the geometry, creating the computational domain, and meshing the model. First, the geometry of the object or system to be simulated, such as a heat exchanger, vehicle, or fluid channel, is created or

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 11 | Nov - 2024 SJIF Rating: 8.448 ISSN: 2582-3930

imported into the software. This step often involves simplification or idealization of the geometry to reduce complexity and computational cost. The next step is creating the mesh, where the entire domain is divided into smaller elements or cells. A high-quality mesh is crucial for accurate simulations, and techniques such as structured, unstructured, or hybrid meshing are used depending on the geometry and problem type. The mesh size is an important factor, as finer meshes provide better accuracy but require more computational resources.

Solving the Governing Equations:

Once the geometry and mesh are ready, the next step is solving the governing equations, including the Navier-Stokes equations for fluid flow and the energy equation for heat transfer. These equations are discretized using numerical methods such as finite volume or finite element methods, which convert the continuous equations into algebraic form. In this phase, boundary conditions (e.g., velocity, temperature, pressure) are applied to the domain, and the solver iterates through the equations to calculate the flow field, temperature distribution, and other relevant physical parameters. Depending on the problem, turbulence models (e.g., k-ε, k-ω) may be employed to model the effects of turbulence, and depending on the complexity, models for heat transfer, chemical reactions, or phase changes may also be included. The solver calculates the variables at each grid point across time steps or steady-state conditions.

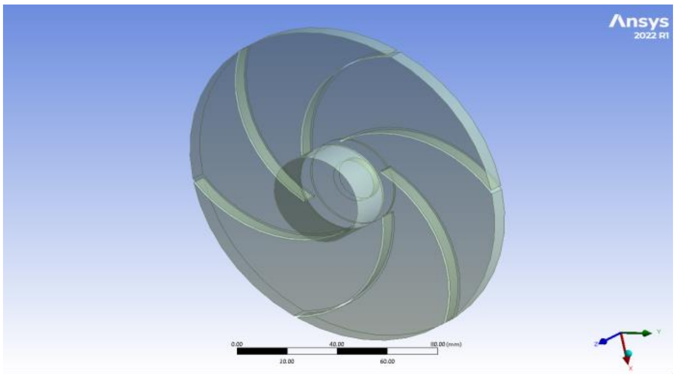
Post-Processing and Analysis:

After the solver has completed the simulations, post-processing is used to analyze and visualize the results. In this phase, the results are extracted, interpreted, and presented in a meaningful way. Common post-processing tools include contour plots, vector plots, and streamlines to visualize flow patterns, temperature distribution, velocity profiles, pressure variations, and other important metrics. Additionally, performance indicators such as heat transfer coefficients, pressure drops, and Nusselt numbers are calculated to assess the efficiency of the system. Sensitivity analyses may also be performed to understand the impact of input parameters on the solution's accuracy. The final analysis helps validate the model, refine the design, and guide engineering decisions for optimizing the system's performance.

Geometry import

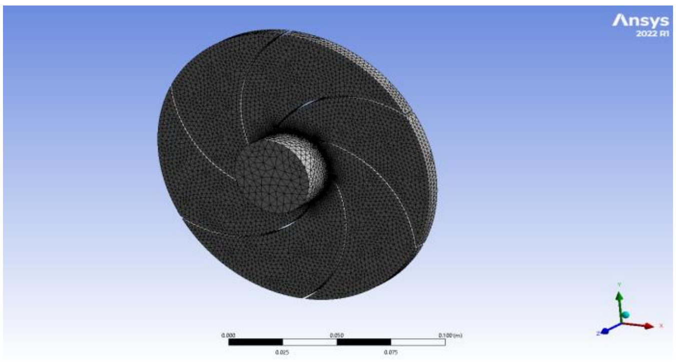
In ANSYS, the geometry of a centrifugal impeller created in CATIA can be imported using the STEP (Standard for the Exchange of Product model data) file format, which facilitates seamless data transfer between CAD software. The STEP format preserves the 3D geometry, ensuring that all critical details—such as dimensions, curves, and surfaces—are accurately transferred. To import the centrifugal impeller into ANSYS Workbench, the user exports it as a STEP file from CATIA and then imports it into ANSYS. This method maintains the integrity of the impeller model, allowing complex features like blade curvature and hub geometry to be retained. Once imported, users can leverage ANSYS's advanced simulation tools to analyze the impeller's performance under various conditions, including fluid dynamics to study flow patterns, structural analysis for stress and deformation, and thermal analysis to evaluate heat transfer. This process enables a comprehensive assessment of the impeller’s functionality and efficiency within a centrifugal pump system.

The basic techniques of the GAs are designed to simulate processes in natural systems necessary for evolution, specially those follow the principles first laid down by Charles Darwin of "survival of the fittest.". Since in nature, competition among individuals for scanty resources results in the fittest individuals dominating over the weaker ones



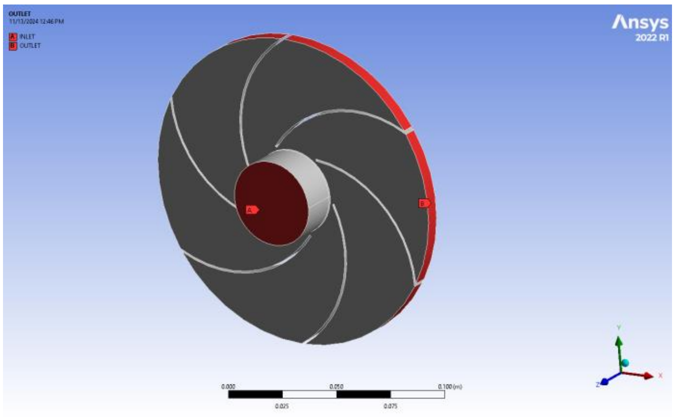
Fluid domain created in ANSYS

Meshing:

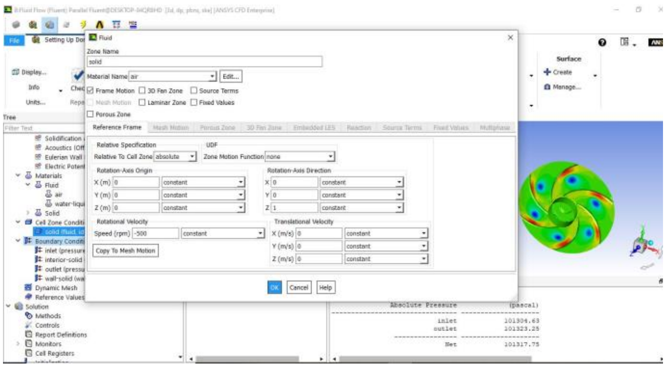


Finite element model created in ANSYS

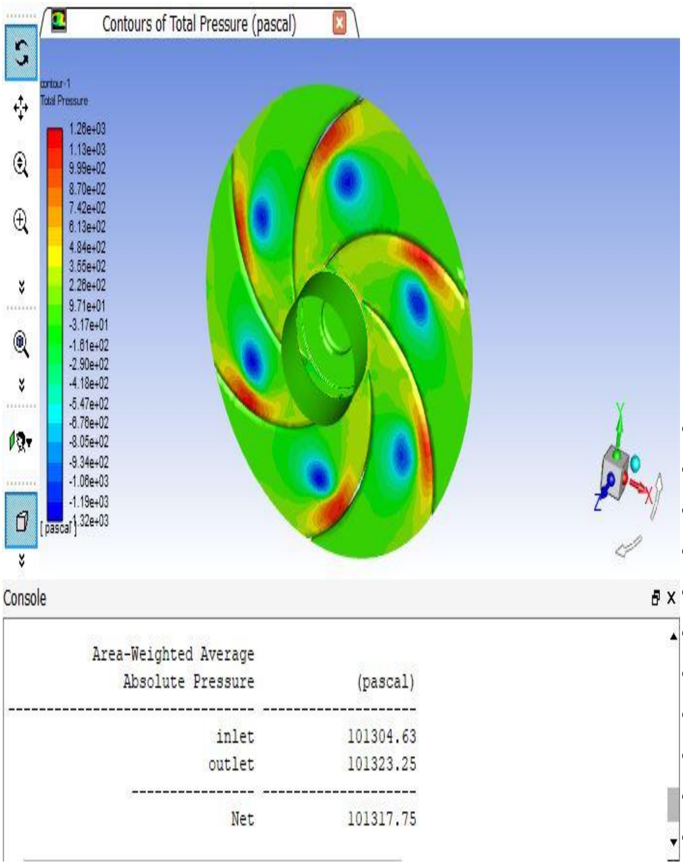
Boundary conditions:



Inlet of fluent domain



Frame motion applied in fluent.



pressure contour on Impeller.

The image above illustrates the pressure distribution on the blade profile of the impeller, with color coding to represent varying pressure levels. The red areas indicate regions of maximum pressure, while the blue areas correspond to the minimum pressure. The pressure within the impeller increases to 1.2e3 Pa, a result of the rotational speed of 500 RPM. This pressure variation is typical of centrifugal pump operation, where fluid experiences increased pressure as it moves radially outward from the center of the impeller. The higher pressure regions, marked in red, are likely located near the leading edges of the blades, where the fluid is accelerated, while the blue areas indicate lower pressure regions, typically found toward the outer edges of the impeller.

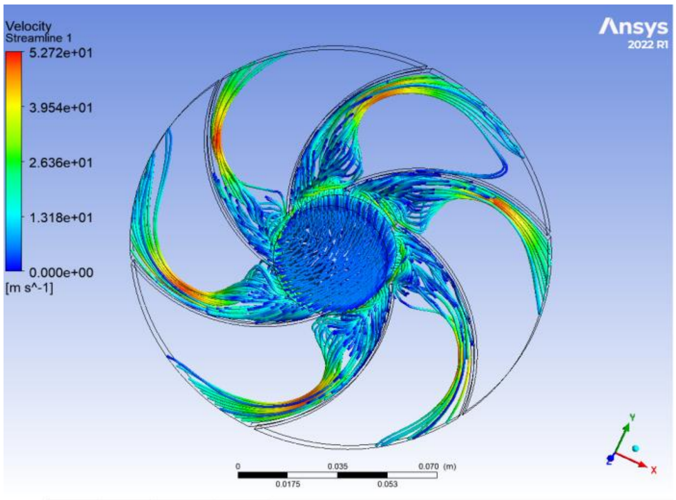


Fig5.7: vector velocity distribution on impeller.

4.FINITE ELEMENT ANALYSIS:

The Finite Element Method is a significant tool for those who are involved in design engineering; it is now regularly used to solve issues in the below areas:

- Acoustics
- Analysis of shock (underwater & in materials)
- Dynamic analyses
- Structural interaction with fluid flows
- Thermal analysis
- Crash simulations
- Fluid flows
- Electrical analyses
- Coupled analyses
- Vibrations
- Mass diffusion
- Buckling problems
- Electromagnetic evaluations

- Metal forming
- Structural Strength design

Ansys software is used in different industries like automobiles, aircrafts, shipyard, spacecraft, manufacturing, education, thermal power stations, fluid problems etc, are used. From ANYSY10.0, it has been dived in to two parts

1. ANYSY APDL(ANSYS programming developing Language)
2. ANYSY workbench

ANSYS APDL is totally depending on Finite Element Methods or Analysis. In this there are different types of elements they are

- Link element
- Beam element
- Shell element
- Solid element
- Pipe element

Through the ANSYS software can analyze the different engineering problems such as

- Structural analysis

1. Static structural
2. Modal analysis
3. Frequency response analysis
4. Buckling analysis
5. Fatigue analysis
6. Creep analysis
7. Contact analysis
8. Impact analysis
9. Transient analysis
10. Response spectrum analysis

- Thermal analysis

1. Conduction problems
2. Convection problems
3. Radiation problems

- Computational fluid dynamics

1. Two dimensional problems
2. Three dimensional problems

- Couple field analysis

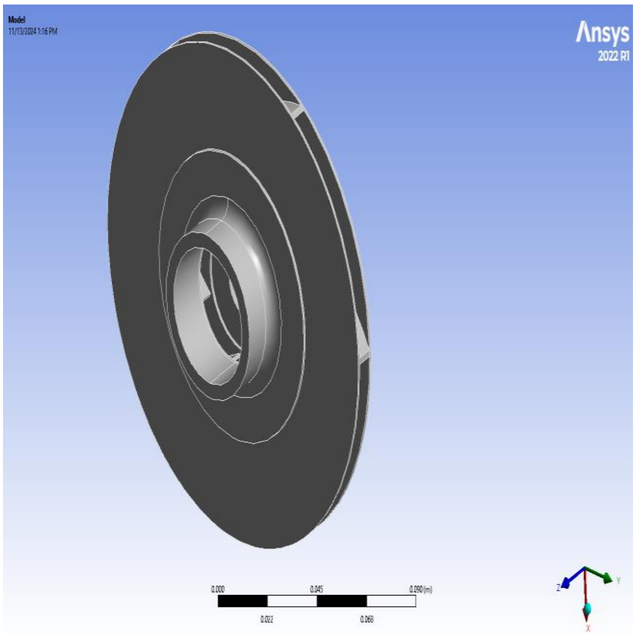
1. Structural and thermal
2. Thermal and fluent
3. Structural thermal and fluent

The above different area of interest can be solved using ANSYS software.

ANSYS workbench is user-friendly software, to use to this software not only skilled person can use but also normal person can use. We can get accurate results like ANSYS APDL. This type of interface is available from 10.0 versions. This is mostly used for industrial projects as well as educational projects. But if we want perfect results, we want to concentrate on boundary conditions, loadings, materials.

The procedure to solve ANSYS is shown below.

- Preference
- Pre-processor
- Postprocessor.
- file that we can import they are IGS file, STEP file and para solid file.



Geometry model of impeller

Material definition

- In ANSYS workbench, important materials are already inbuilt in the software. We can either modify the material parameters or else duplicate materials can be created. Different materials can be assign to different solids can be assigned.

Material properties

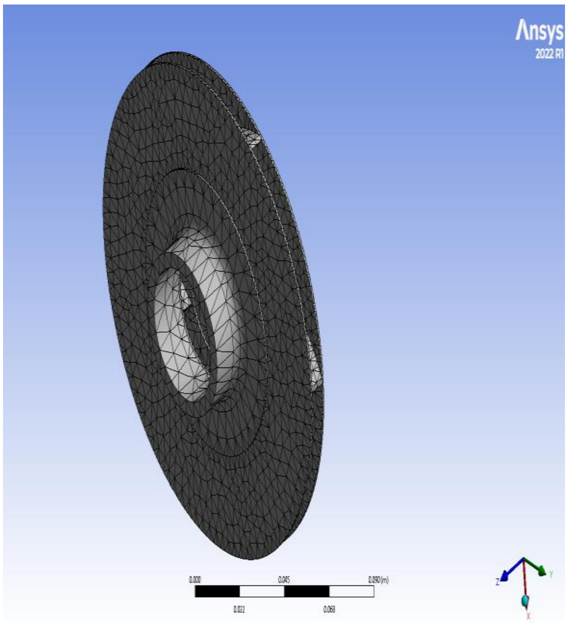
Connections:

	youngs modulus	poission ratio	density
	G.Pa		Kg/m3
AA6061	68	0.33	2700
INCONEL	170	0.29	8200
CAST IRON	110	0.28	7200
STAINLESS STEEL	190	0.265	7850
STRUCTURAL STEEL	200	0.3	7850

- ANSYS workbench has different types of different types of connections. These connections can be applied to edge to edge and face to face. A solid is connected to another solid, few connection is used they are such as bonded connection, frictionless connection, no separation connection, rough connection and joint connections.

Meshing:

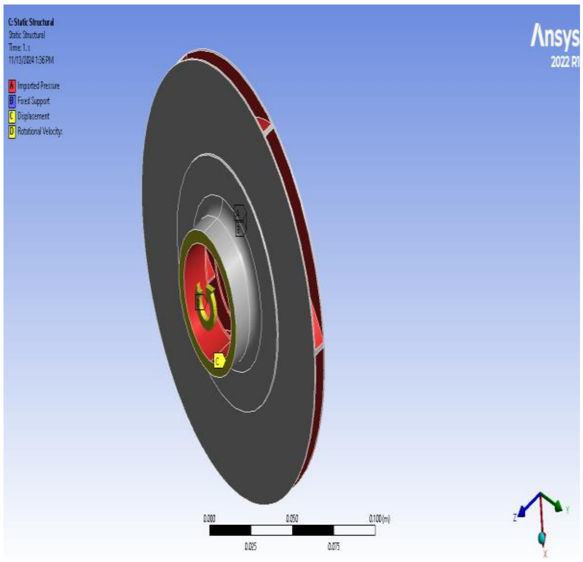
- It is an important technique in Finite element methods. Generally mesh 200 element is available in the ANSYS workbench. These elements are used to solve in the shell and solid element. When there is number of element present we can get the exact results. At the same time, hexagonal meshing gives best results.



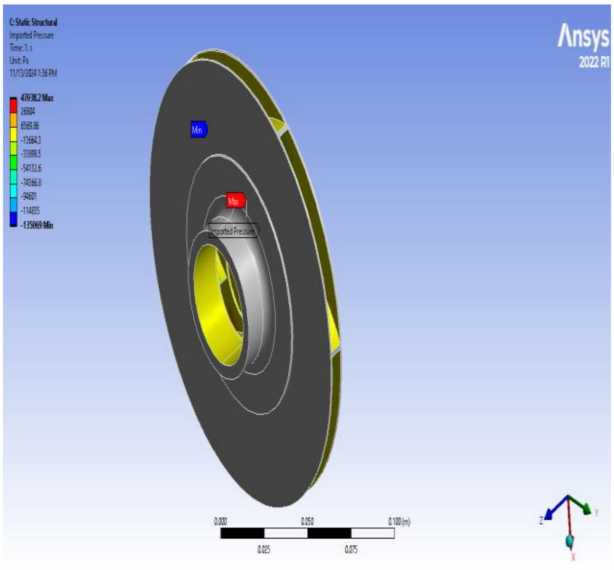
Finite element model of Impeller

- ### Solution
- ### Boundary conditions and loading

- There are different types of boundary conditions that can be applied to edge or faces. In this there is generally fixed support, displacement, remote displacement and frictionless support are used in ANSYS workbench.
- To run an analysis for a structure, loading plays a vital role. The loading given to perfect loading and prefect direction is very important. In ANSYS workbench, loading can be given to edge, face and body. This has point load, pressure load, hydrostatic load, joint load, bolt penetration loading etc, are available. After the load and boundary conditions are applied, we have to click solve radian button to solve the analysis.



- Fig: boundary condition on Impeller



- Fig: Imported pressure from CFD.

Postprocessor:

- Post possessing is output of analysis. After we solved the problems, to check the results postprocessor is used. In this we can find in different direction for deformation, strains and stresses. And this can be converted into an image and we can paste it in the report.
- In this present work, initially we took different geometric parameters are taken form existing optimized article. In this Impeller, using detailing drawings we created in CATIA software and saved as IGS Software in particular location of the computer.
- To run the Analysis, 64 bit operating system, 4GB ram And ANSYS 2022 is used. Without any problem, to run analysis software this configuration is very apt. The previously created IGS file is imported on ANSYS file geometry. In ANSYS, static structure and modal analysis are performed. The table which is shown below, different materials for different components are used Impeller. Solid mesh 193 element are used to divide the geometric body in to small strips (Finite elements)
- In the present work, the entire Impeller component are divided into 16289 tetrahedron elements and 32318 nodes. In real time, the position where we place the hub of impeller in its exact location, likewise in ANSYS software same location we applied the boundary condition. Fixed boundary conditions are applied at the hub of impeller. Bounded connections are provided between surfaces and solid shaft. Dynamic pressure Load applied on front face of blades of impeller. The final post processing result such as deformation and stresses are placed below.

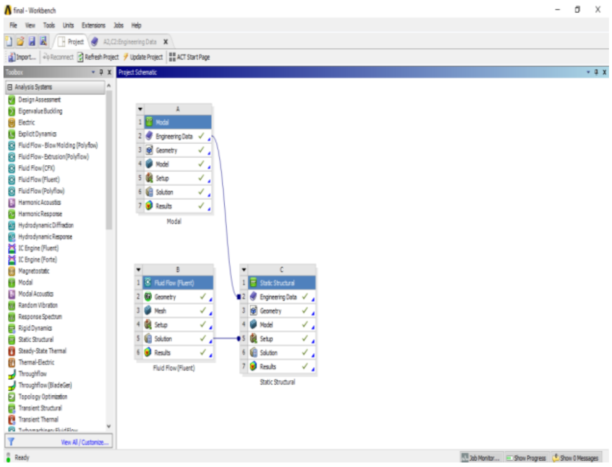


Fig6.2: planning to run the analysis.

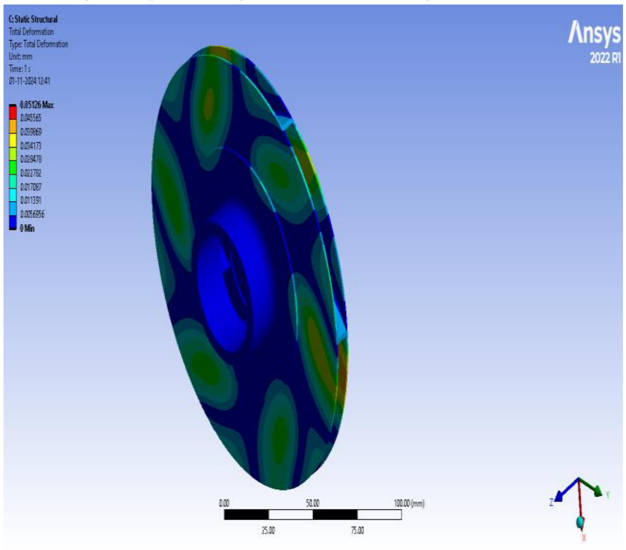
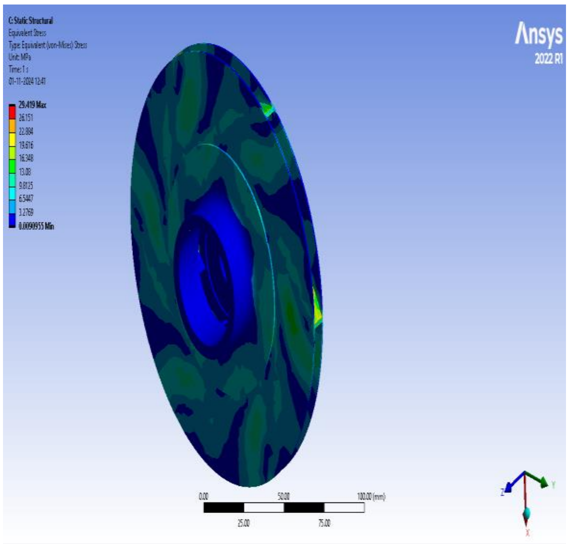
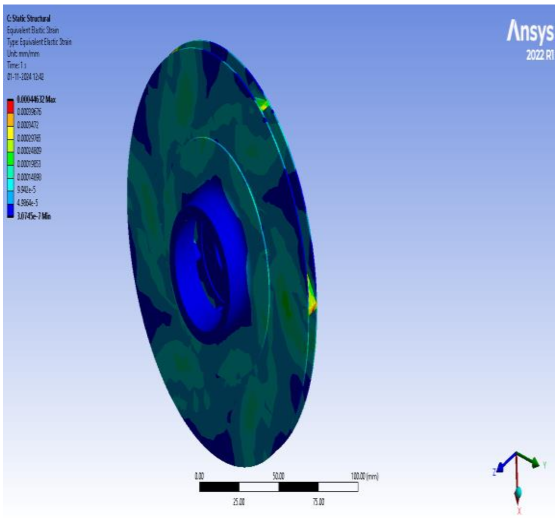


Fig : deformation of Impeller with AA6061 material.

In the above image shows the total deformation of Impeller with AA6061 material. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum deformation and blue indicates minimum deformation all together maximum deformation is 0.0512 mm. This deformation is much of considerable deformation.



- Fig : Stress on Impeller with AA6061 material.
- In the above image shows the Stress on Impeller with AA6061 material. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum Stress and blue indicates minimum Stress all together maximum deformation is 29.4 M.Pa. This deformation is much of considerable Stress.



- Fig: Strain on Impeller with AA6061 material.
- In the above image shows the Strain on Impeller with AA6061 material. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum Strain and blue indicates minimum Strain all together maximum deformation is 0.00044.

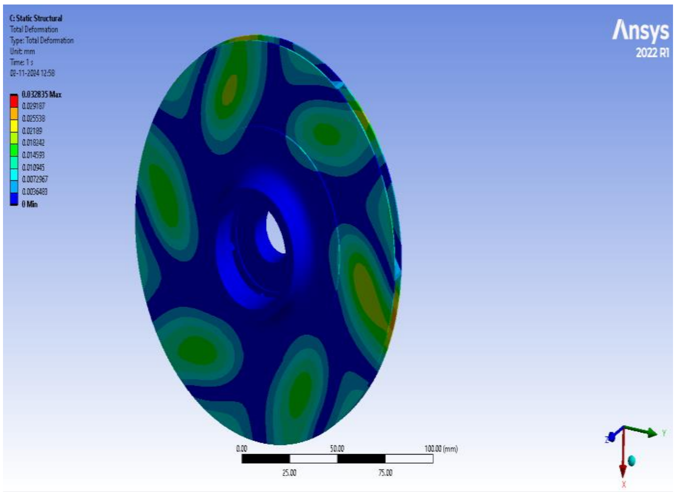


Fig : deformation of Impeller with Cat Iron material.

In the above image shows the total deformation of Impeller with Cat Iron material. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum deformation and blue indicates minimum deformation all together maximum deformation is 0.032 mm. This deformation is much of considerable deformation.

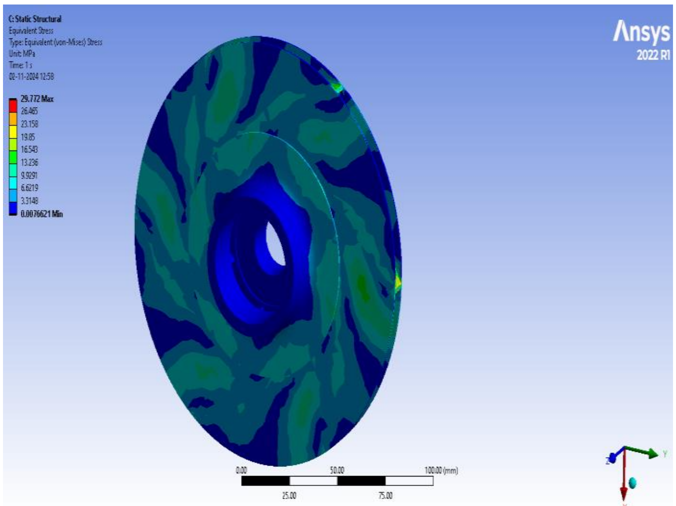


Fig : Stress on Impeller with Cat Iron material.

In the above image shows the Stress on Impeller with Cat Iron material. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum Stress and blue indicates minimum Stress all together maximum deformation is 29.74 M.Pa. This deformation is much of considerable Stress

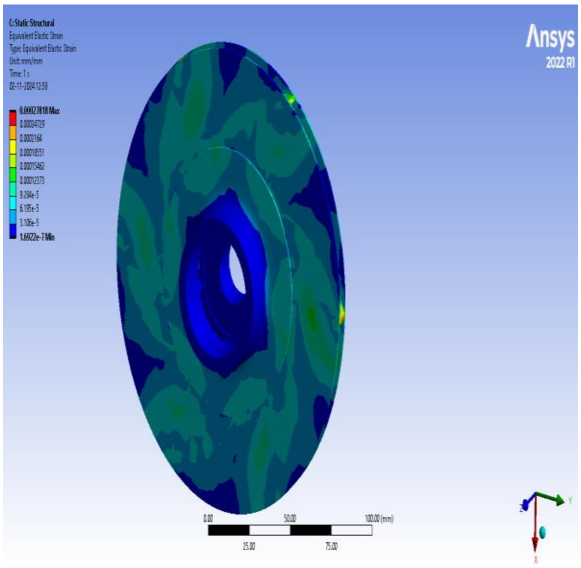


Fig: Strain on Impeller with Cat Iron material.

In the above image shows the Strain on Impeller with Cat Iron material. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum Strain and blue indicates minimum Strain all together maximum deformation is 0.00027.

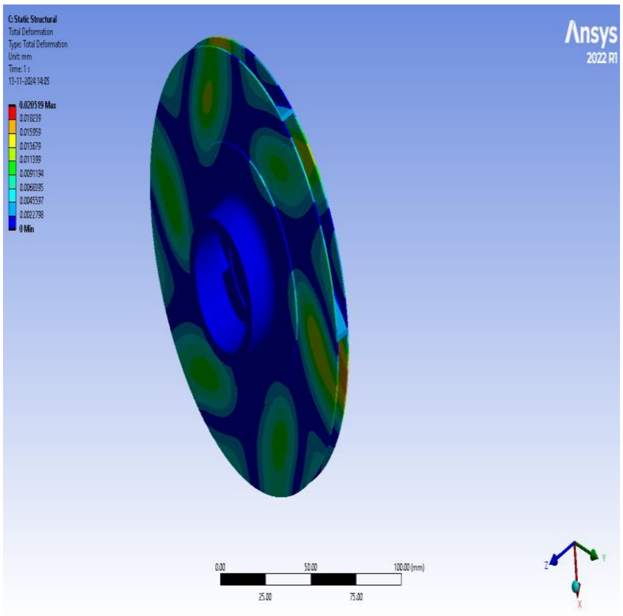
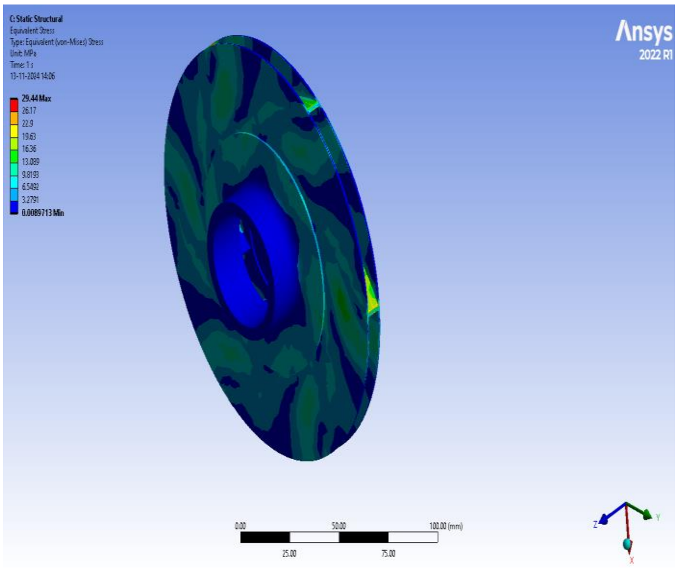


Fig: Deformation of impeller with Inconel.

In the above image shows the total deformation of Impeller with Inconel. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum deformation and blue indicates minimum deformation all together maximum deformation is 0.0204 mm. This deformation is much of considerable deformation.



- Fig: Stress on impeller with Inconel.
- In the above image shows the Stress on Impeller with Inconel. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum Stress and blue indicates minimum Stress all together maximum deformation is 29.4 M.Pa. This deformation is much of considerable Stress.

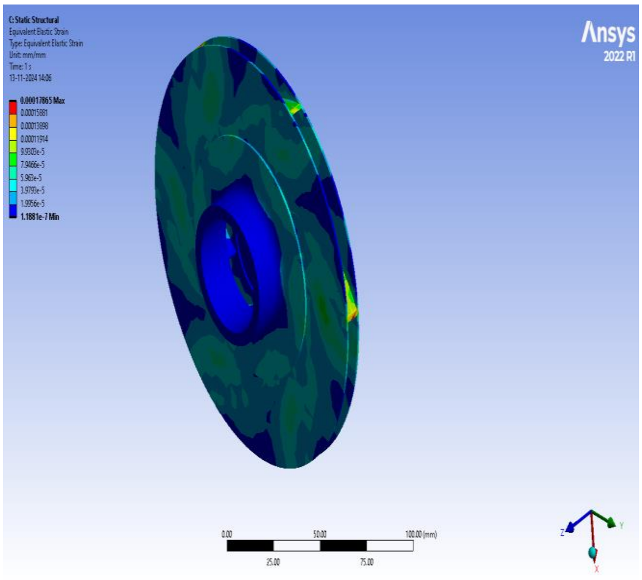


Fig: Strain on impeller with Inconel.

In the above image shows the Strain on Impeller with Inconel. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum Strain and blue indicates minimum Strain all together maximum deformation is 0.00017.

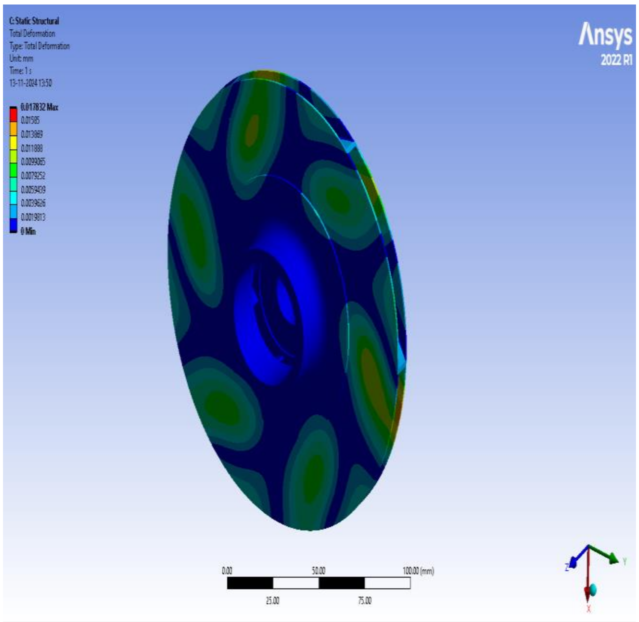


Fig:Deformation of structural steel impeller.

In the above image shows the total deformation of Impeller with structural steel. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum deformation and blue indicates minimum deformation all together maximum deformation is 0.017 mm. This deformation is much of considerable deformation.

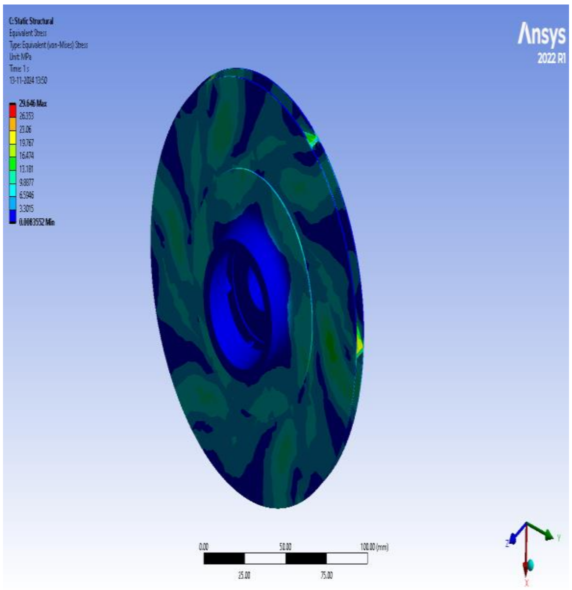


Fig: Stress on impeller with structural steel.

In the above image shows the Stress on Impeller with structural steel. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum Stress and blue indicates minimum Stress all together maximum deformation is 29.61 M.Pa. This deformation is much of considerable Stress.

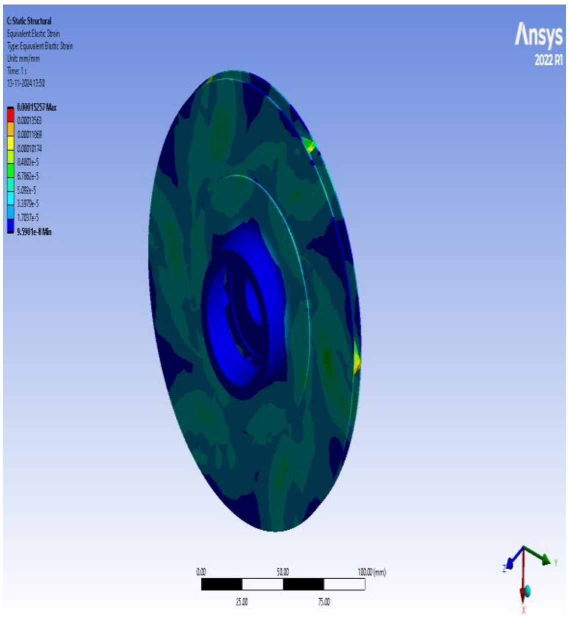


Fig: Strain on impeller with structural steel.

In the above image shows the Strain on Impeller with structural steel. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum Strain and blue indicates minimum Strain all together maximum deformation is 0.00015.

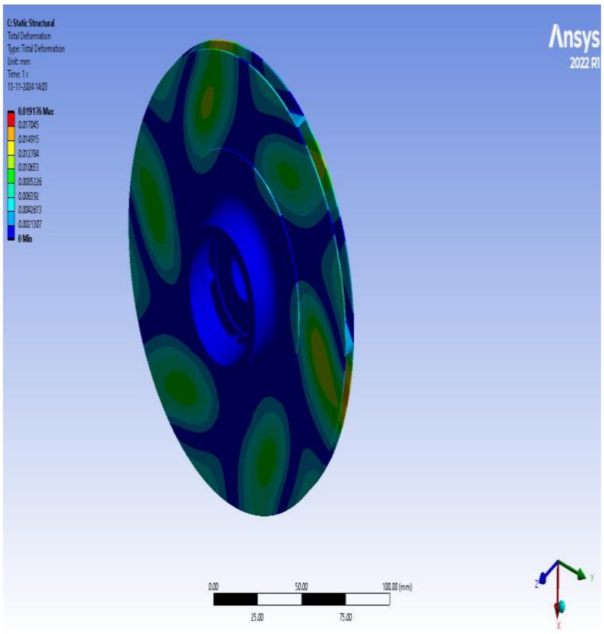


Fig: Deformation of impeller with Stainless steel.

In the above image shows the total deformation of Impeller with Stainless steel. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum deformation and blue indicates minimum deformation all together maximum deformation is 0.019 mm. This deformation is much of considerable deformation.

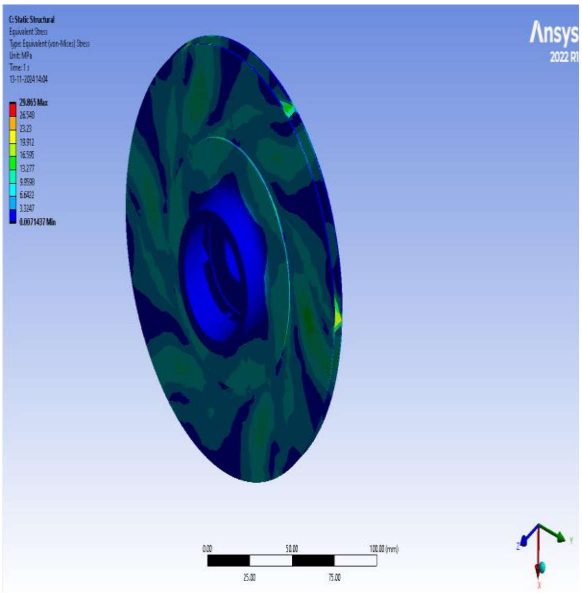


Fig: Stress on impeller with Stainless steel.

In the above image shows the Stress on Impeller with Stainless steel. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum Stress and blue indicates minimum Stress all together maximum deformation is 29.83 M.Pa. This deformation is much of considerable Stress.

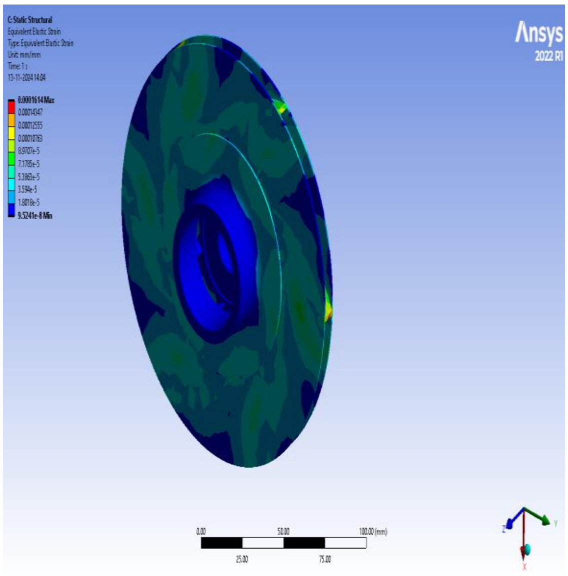


Fig: Strain on impeller with Stainless steel.

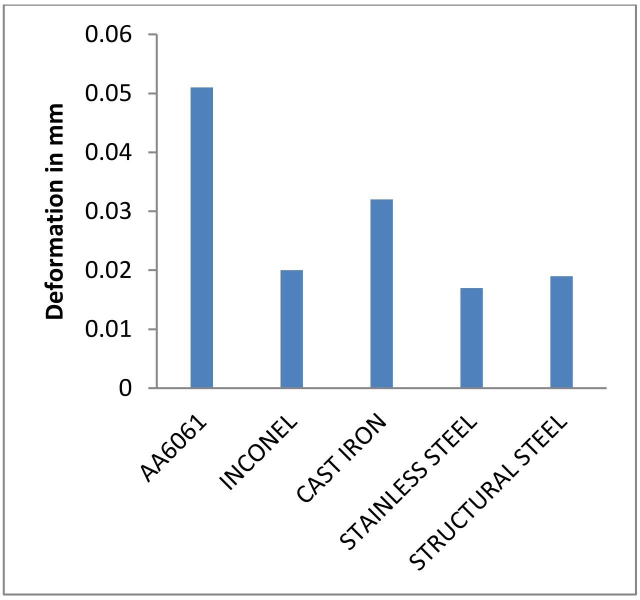
In the above image shows the Strain on Impeller with Stainless steel. After dynamic load 0.04 MPa load applied on face of impeller walls and the red indicate the maximum Strain and blue indicates minimum Strain all together maximum deformation is 0.00016.

5.RESULTS AND DISCUSSION:

Fluid structure interface analysis is performed on impeller with five types of materials such as AA6061, incoel, cast iron, stainless steel and structural steel. Here the deformation, stress and strain reading on noted in below table and graphical representation.

Material	Deformation in mm
AA6061	0.051
INCONEL	0.02
CAST IRON	0.032
STAINLESS STEEL	0.017
STRUCTURAL STEEL	0.019

Table: deformation of impeller with different materials.

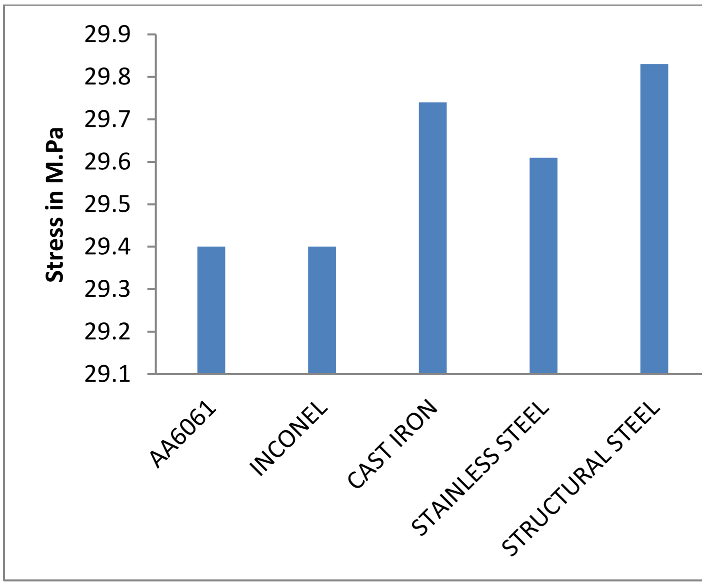


Graph: deformation of impeller with different materials.

In the analysis of material deformation under operational conditions, the minimum deformation is considered the best as it indicates greater material strength and resistance to stress. The deformation values for different materials show that Inconel exhibits the least deformation at 0.02 mm, making it the most suitable material in terms of structural integrity for applications requiring minimal distortion under stress. Stainless Steel follows closely with a deformation of 0.017 mm, also indicating strong resistance to deformation. Structural Steel and Cast Iron show slightly higher deformations at 0.019 mm and 0.032 mm, respectively, but still provide satisfactory performance in many applications. AA6061, an aluminum alloy, shows the highest deformation at 0.051 mm, which suggests it may not perform as well as the others under high-stress conditions. Choosing materials with lower deformation is crucial for ensuring the longevity and reliability of components, especially in high-performance systems such as impellers.

Material	Stress in M.Pa
AA6061	29.4
INCONEL	29.4
CAST IRON	29.74
STAINLESS STEEL	29.61
STRUCTURAL STEEL	29.83

Table: Stress on impeller with different materials.

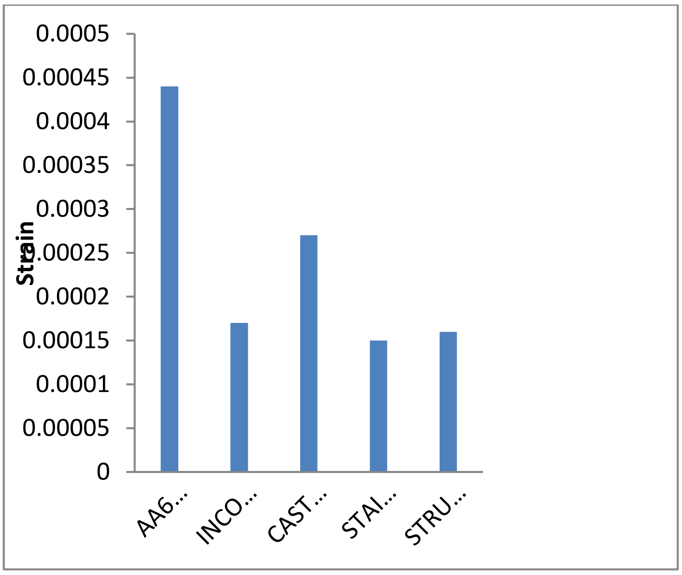


Graph: Stress on impeller with different materials.

The material stress values indicate how much stress each material can withstand before failure. AA6061 and Inconel both exhibit a stress of 29.4 MPa, showing that they can handle similar amounts of stress under load. Cast Iron shows a slightly higher stress value of 29.74 MPa, while Stainless Steel and Structural Steel have stress values of 29.61 MPa and 29.83 MPa, respectively. These differences in stress values are minimal, suggesting that the materials are fairly similar in terms of their strength under typical operational conditions. However, these values do not account for other factors such as material fatigue or environmental conditions, which could influence material performance. In applications where high strength is required, materials like Inconel and Structural Steel may provide additional benefits due to their higher resistance to wear and higher-temperature performance, even though their stress values are very close.

Material	Strain
AA6061	0.00044
INCONEL	0.00017
CAST IRON	0.00027
STAINLESS STEEL	0.00015
STRUCTURAL STEEL	0.00016

Table: Strain on impeller with different materials.

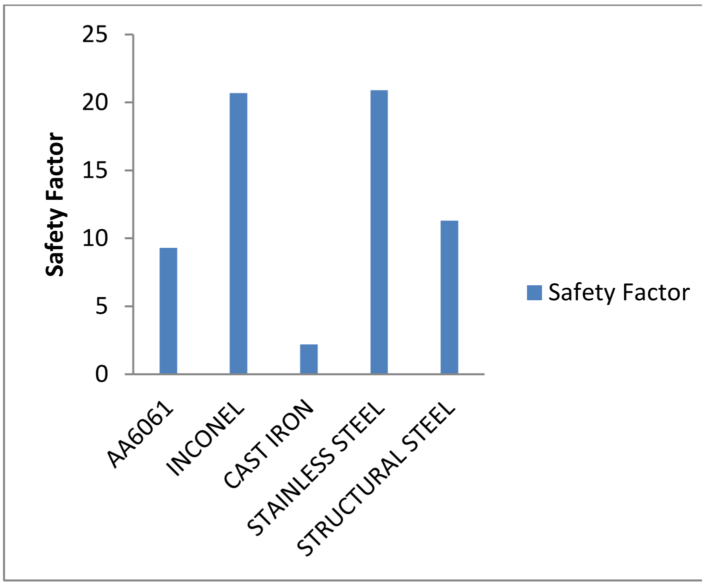


Graph: Strain on impeller with different materials.

The strain values represent the material's ability to deform elastically under stress, with lower strain indicating higher stiffness and resistance to deformation. Inconel exhibits the lowest strain of 0.00017, meaning it deforms the least under stress and is highly resistant to elastic deformation, making it an excellent choice for high-stress, high-temperature applications. Stainless Steel follows closely with a strain of 0.00015, offering similar stiffness. Structural Steel has a strain value of 0.00016, slightly higher than stainless steel but still indicating strong resistance to deformation. Cast Iron shows a strain of 0.00027, and AA6061 has the highest strain value of 0.00044, suggesting that it is more prone to deformation under stress compared to the other materials. Materials with lower strain, such as Inconel and Stainless Steel, are generally preferred in high-performance applications where minimal deformation is critical to maintaining structural integrity and functionality.

Material	Safety Factor
AA6061	9.3
INCONEL	20.7
CAST IRON	2.2
STAINLESS STEEL	20.9
STRUCTURAL STEEL	11.3

Table: Safety factor of impeller with different materials.

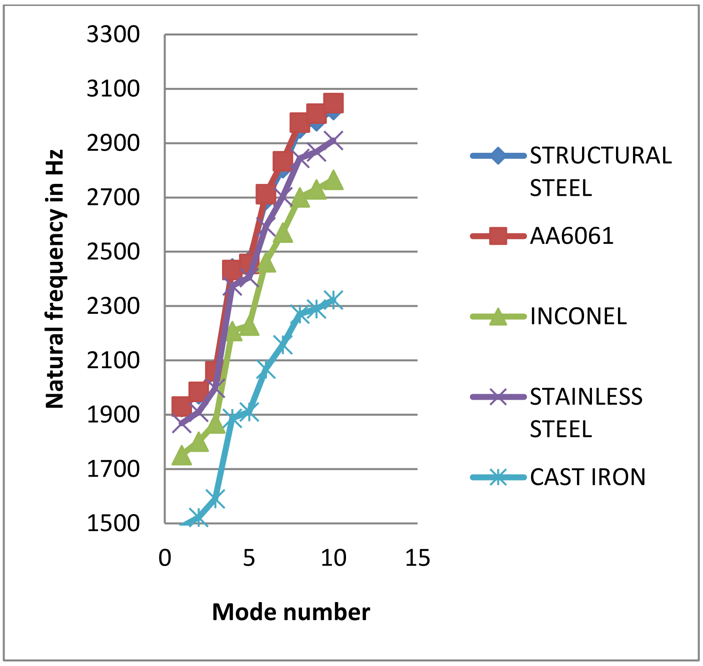


Graph: Safety factor of impeller with different materials.

The safety factor represents the ratio of the material's ultimate strength to the actual applied load, indicating the material's ability to withstand unexpected stresses or failure. A higher safety factor means the material is more capable of handling extreme conditions without failure. Inconel and Stainless Steel have very high safety factors of 20.7 and 20.9, respectively, indicating that these materials can tolerate much higher stresses than they are expected to experience in typical operating conditions, making them ideal for demanding applications such as aerospace and high-temperature environments. Structural Steel also has a relatively high safety factor of 11.3, providing a good margin of safety for most engineering applications. AA6061 has a safety factor of 9.3, which is still quite high, but lower than that of the more robust materials like Inconel and Stainless Steel. Cast Iron has the lowest safety factor of 2.2, suggesting that it is less capable of withstanding extreme stresses, and is typically used in applications where loads are more predictable and controlled. The materials with higher safety factors (Inconel, Stainless Steel, and Structural Steel) are better suited for high-stress environments, while AA6061 and Cast Iron might be more suitable for less demanding applications.

	STRUCTURAL STEEL	AA6061	INCONEL	STAINLESS STEEL
1	1928.7	1931.3	1752.2	1868.5
2	1976	1985.2	1801.1	1909.5
3	2059.2	2059.8	1868.9	1998
4	2439.7	2433.8	2208.2	2374.3
5	2466.9	2456.1	2228.4	2405.3
6	2690.9	2712.7	2461.2	2592.5
7	2808.3	2833.7	2570.9	2703.1
8	2953.6	2976.6	2700.6	2844.6
9	2980.7	3009	2730	2868
10	3022	3048.4	2765.8	2910

Table: Natural frequency of impeller with different materials.



Graph: Natural frequencies for different materials.

The table shows the natural frequencies of an impeller made from various materials, with data presented for each material at different frequency points (1 to 10). Structural Steel exhibits the highest natural frequencies, starting at 1928.7 Hz and reaching up to 3022 Hz by the 10th frequency point. AA6061, a lightweight aluminum alloy, follows closely behind Structural Steel, with its natural frequencies ranging from 1931.3 Hz to 3048.4 Hz. Inconel, known for its high-temperature and corrosion resistance properties, starts at a lower frequency of 1752.2 Hz and rises to 2765.8 Hz, indicating that it has lower natural frequencies compared to the other materials. Stainless Steel shows a similar trend to Inconel, with frequencies ranging from 1868.5 Hz to 2910 Hz. Cast Iron, with its higher mass and lower stiffness, has the lowest natural frequencies, beginning at 1488 Hz and rising to 2322.8 Hz. These variations in natural frequencies are crucial for determining the material's response to vibrational forces, which is important in preventing resonance and ensuring the longevity and efficiency of the impeller in operation. Materials with higher natural frequencies, like Structural Steel and AA6061, are generally better at avoiding resonance in high-speed applications.

6.CONCLUSION

In this study, the geometry of the impeller was initially created in CATIA software and imported into ANSYS for simulation. A FLUENT analysis was conducted on the impeller, rotating at 500 RPM, revealing a significant pressure difference of approximately 120 times from the inlet to the outlet. During the impeller's rotation, hydrodynamic forces acted on the impeller blades, with a maximum pressure of 0.04 MPa observed on the impeller wall structure. To evaluate the structural integrity, dynamic pressure loading was applied, and structural analysis was performed, revealing that maximum deformation and stresses occurred at the blade and blade-supporting plate. The stresses were observed to be around 30 MPa, with deformation being less than 1 mm. The analysis was repeated for various materials, and while the stress values remained relatively consistent across materials, the safety factor varied, with stainless steel exhibiting the highest safety factor among all materials.

The vibration analysis showed that frequencies above 1500 Hz were observed for all materials, indicating that they are well away from the resonance frequency, which is crucial in preventing failure due to vibrational resonance. Stainless steel emerged as the best material for the impeller, exhibiting the highest natural frequencies and the highest safety factor, making it the most suitable material to withstand hydrodynamic loading, centrifugal forces, and vibrational stresses. The results suggest that stainless steel impellers offer the best combination of strength, durability, and resistance to deformation, vibration, and fatigue, making it the optimal choice for centrifugal pump applications. Thus, based on these comprehensive analyses, stainless steel is concluded to be the most effective material for impeller design, ensuring reliable performance and longevity under operational conditions.

7.REFERENCES:

[1] Tsui Y.Y. and Lu S.L., (2008), “Evaluation of the performance of valvelessmicropump by CFD and lumped system analyses”, Sensors and Actuators, Vol.148, Issue 1, pp.138-148.

[2] Kumar S. and Bergada J.M., (2013), “The effect of piston grooves performances in an axial piston pump via CFD analysis”, International Journal of Mechanical Sciences, Vol. 66, pp. 168-179.

[3] Fan J., Eves J., Toropov V.V., Kapur, Copley D. and Mincher A., (2011), “Computational fluid dynamic analysis and design optimization of jet pump”, Computers and Fluids, Vol. 46, Issue 1, pp. 212-217.

[4] Alves M. C., Barbosa J.R. and Prata A. T., (2013), “Analytical and CFD modeling of the fluid flow in an eccentric-tube centrifugal oil pump for hermetic compressors”. International Journal For Refrigeration, Vol. 1, Issue 7, pp. 1905-1915.

[5] Lucmann A.J., Alves M.V.C. and Barbosa J., (2009), “Analysis of oil pumping in reciprocating compressor”, Applied Thermal Engineering , Vol. 29, Issue 14-15, pp. 3118-3123.

[6] Kumar S., Bergada J.M. and Watton J., (2009), “Axial piston pump grooved slipper analysis by CFD simulation of three dimensional NVS equation in Cylinder Coordinates”, Computers and Fluid, Vol. 38, Issue 3, pp. 468-663.

[7] Pei J., Hai C., Lv M., Huang X., Shen K. and Bi K., (2013), “Collision Contact Characteristics for reciprocating pump using FEA and experiments”, Advances in Mechanical Engineering, Volume 2013.

[8] Kumar A., Grover K. and Budania B., (2012), “ Optimization of Connecting Rod Parameters using CAE Tools”, International Journal of Latest Treads in Engineering and Technology, Vol. 1, Issue 3, pp. 98-104.