International Journal for Ignited Minds (IJIMIINDS)

Stress Analysis and Design Optimization of Impeller Mount Machine Structure

Bhaskar M[1] & Suhas V[2]

[1]P G Scholar, The Oxford College of Engineering, Bengaluru.
bhaskarm827@gmail.com

[2]Asst. Prop., The Oxford College of Engineering, Bengaluru.

***ABSTRACT* — In the present work, a frame structure over which two impellers are mounted is modelled and meshed using hyper mesh. One dimensional beam elements, two dimensional shell elements are used for the mesh. The mesh is checked for all the quality requirements for better results. The load application is done through rigid body elements (RBE3) connecting to the slave or dependent nodes. After application of boundary conditions, the problem is executed for structural conditions. Self-weight analysis is carried out and the developed stresses and deformations are well within the limits. Later analysis is continued with external loads for stress and deformation conditions. Since rigidity is the main design requirement, design optimization is carried out based on limits of deformation. Totally 11 design sets are represented along with two state variables taking weight as the objective function. Here the design variables are nothing but geometrical variable or thickness of the members which need to be reduce for weight optimization. All the design sets are represented along with the best set. Finally, modal analysis is carried out to check the dynamic vibration conditions.**

Keywords— Impeller, Mount, Design, Stress, Optimisation, Ansys.

1. INTRODUCTION

Fixtures are very essential in holding the members either during machining operation or production operations. Fixtures should be rigid enough to hold the members rigidly so that required machining accuracy can be obtained. Fixture design involves structural engineering for functional safety Rigidity is the keyword of any fixture design. Modern industry always aims at perfection to compete in the engineering industry. So efficient design is the coin of success in the present world. Due to the advances in the computational technology, virtual simulation is slowly replacing the conventional practice of design. Fixture design is mainly on locating points and constraints for proper strength to obtain the required accuracy in the design. Impellers can be defined as a forcing device for pumping the fluid under pressure in desired direction. Similarly, another definition of impeller is a rotor or a rotor blade. Structural designing methods investigate stability, strength and rigidity of structures, which usually depends on formulae. The results obtained from these methods are feasible designs may not be necessarily an optimum design. This drawback leads to choose structural component from various available domain of solutions or to choose optimal layout of certain structure. Thus, optimized designs chosen from various solutions produce highly efficient and reliable results. The major objectives of structural optimization are thickness, shape and topology. Specialized structural optimization software emerged utilizing more advanced approximation technologies for enhancing overall efficiency of results. The progress in structural optimization occurs parallel in the fields of structural analysis, mathematical programming, geometric modelling and computer hardware. Real words problems are very complex in nature and classical approaches of optimization are not sufficient to solve them.

2. PROBLEM DEFINITON

Optimum design of impeller mount structure using finite element analysis is the main definition of the problem which includes the dynamic stability of the system for vibrations. The problem objective includes

- Theoretical calculations to check the designed geometry
- Geometrical modelling
- Meshing and analysis for optimization

3. METHODOLOGY

- ✓ Plate Thickness calculations based on mechanics of material concepts.
- ✓ Geometrical Modeling and Finite Element Representation.
- ✓ Structural analyses and factor of safety for the components.
- ✓ Design optimization of Machine structure for the given loads using APDL code.
- ✓ Geometrical Modelling of the Problem.

3.1 Material Details

Material Name: SAILMA450 (Steel Authority of India Manufactured with 450Mpa Yield Stress), Young's Modulus=200Gpa, Density =7800kg/m3. Factor of Safety=5, Allowable stress: 90Mpa, Total Load: 40000N

3.2 Assumptions

- Material is assumed to be isotropic and homogenous.
- Analysis does not consider cracks or voids in the structure.
- Structure is assumed to be of uniform strength for the given material and material properties does not vary through the loading process.
- Simple shell and beam elements are used for analysis.
- RBE3, an interpolation element is used for load transfer.

3.3 Element Type

Shell and beam elements are used for the problem. The description of the elements is as follows.

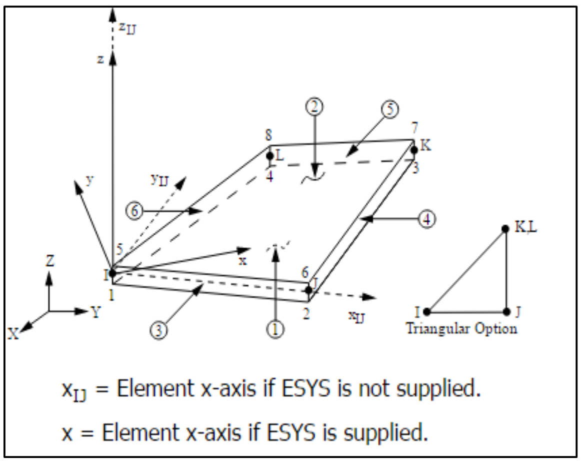

Fig 1: Shell63 Element.

Shell63 element is useful for both bending and torsion. It is a degenerative element which converts from 4 nodes to 3 nodes. So it can be used for axisymmetric problems where element is a ring element which requires three nodes. Stress stiffening and large deformation effects are included in the element formation. The advanced version of element is shell181 which includes both linearity and nonlinearity. Shell63 element can't calculate the residual stresses generated in the problem. The element is defined with six degrees of freedom. So both translational and rotational loads can be applied.

Beam188 element is suitable for one dimensional problems. The element is defined with six degrees of freedom. This element is created to replace the old elements of Beam3, Beam4 and Beam24 elements. It supports both linearity and nonlinearity along with residual stress estimations. It has load key specification for traction load applications. Orientation node can be specified for beam orientation which increases the moment of inertia in the loading directions. Beam188 element is mainly used to represent bolts and connections in the structural assembly

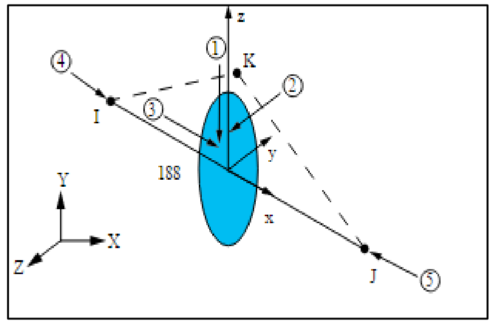

Fig 2: Beam188 Element.

RBE3 Element: RBE stands for rigid body elements which are used to connect one node to multiple nodes for load transfer. Here rigid body stands for no deformation in the element which helps in transferring complete load to the connected nodes. Practically no structure is rigid, but this assumption helps in simplifying the process of calculations along with worst case of loading for the components as there is no loss of load. Rbe3 element transfers loads based on their distance from the master nodes. The moments are calculated based on the normal distance from the point of consideration.

3.4 Geometry of the Problem

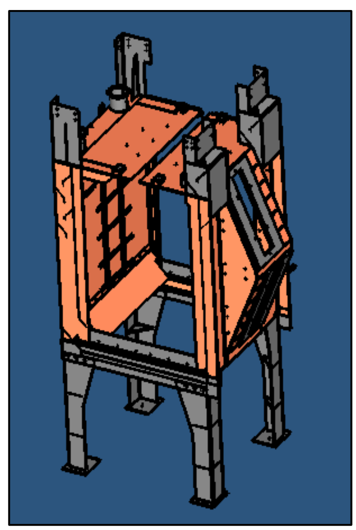

Fig 3: Geometry of the Problem.

The fig 3 shows geometry of the problems. The geometry is built using Solid Edge software. Initially two dimensional views are created using sketcher of Solid Edge and later converted to three dimensional views by Solid Modelling. Finally, all the part models are assembled to form the three dimensional problem. Solid Edge software has better options of drafting to find the dimensions of the problem.

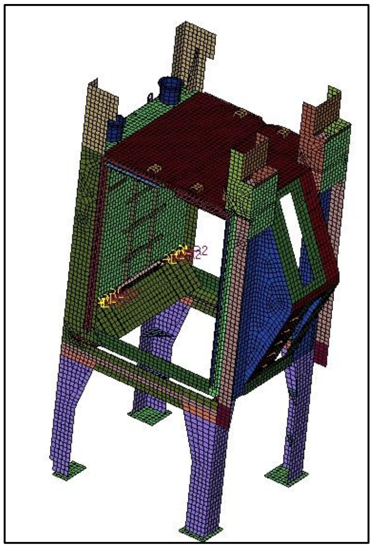

Fig 4: Finite Element Model of the entire system.

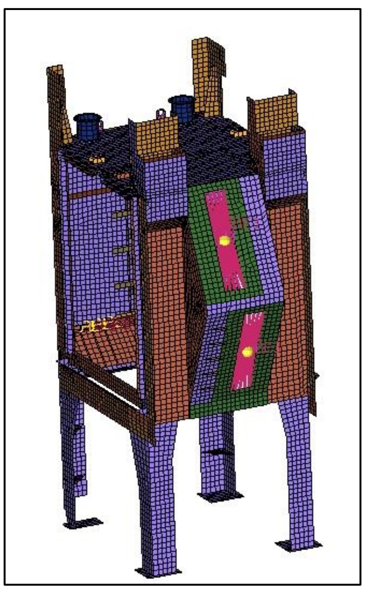

Fig 5: RBE 3 Element for Load Transfer.

4. RESULTS & DISCUSSION

Analysis has been carried out to find the structural safety of the frame structure to mount impellers and to check the possibility of design optimization in the problem. The overall analysis procedure is as follows.

4.1 Self Weight Analysis

Initially self-weight analysis is carried out to check the structural safety. The results are as follows. Self-weight is very important when self-weight of the structure is heavy compared to the external loads. The present problem has a weight of 1120kgs, but the external load is 8 tons. So external load has more value compared to the self-weight of the problem.

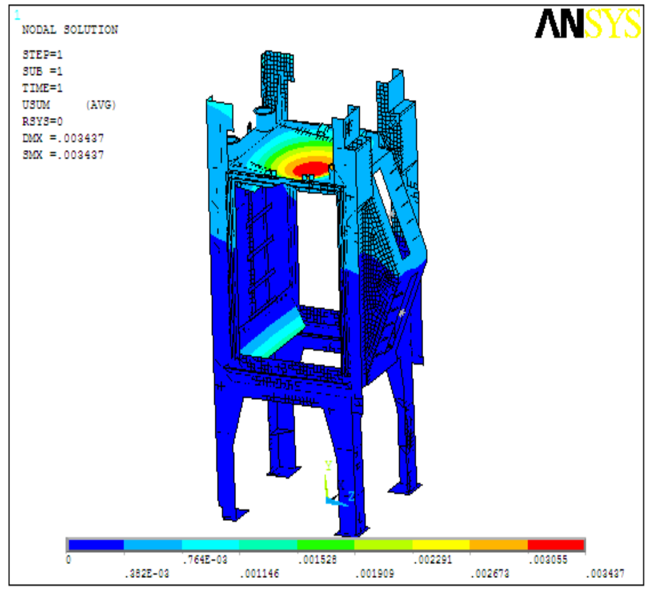

Fig 6: Deformation in the Structure.

The fig 6 shows deformation in the problem equal to 0.003427mm as shown in the fig. Maximum displacement is shown with red color region.

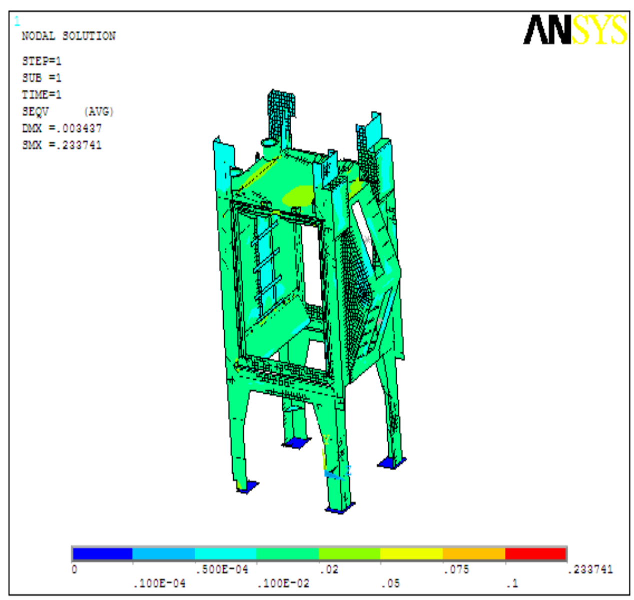

Fig 7: von-Mises Stress Plot.

The fig 7 shows developed von-Mises stress in the structure due to self-weight. Maximum stress is around 0.233Mpa as shown in the status bar. Even constraint regions are subjected to stress represented by different color coding.

4.2 External Load Calculation

Further external load of 40000N is applied normal to the impellor mount position and results are obtained again for structural safety.

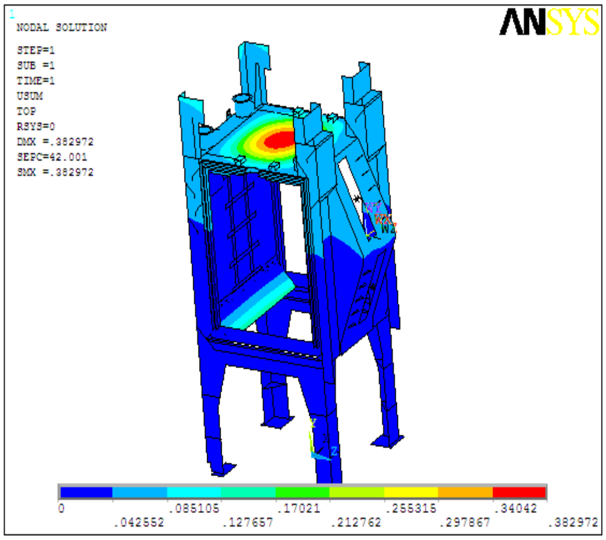

Fig 8: Deformation due to External Load.

The fig 8 shows developed displacement of 0.382mm for the given loading conditions. Maximum displacement is less than the allowable deflection of the problem.

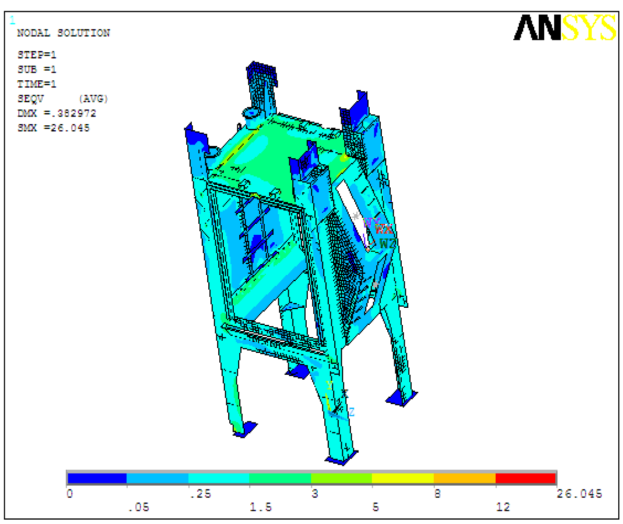

Fig 9: von-Mises Stress Plot.

The fig 9 shows developed von-Mises stress in the problem. Observation of stress and deformation in different components helps in finding the optimum designs. Since the objective is to design and optimize the structure, uniform thickness members are grouped in separate components for results presentation. These thickness groups are optimized for better results. The results of individual components are represented as follows. The table 1 shows components and its thickness in the assembly for Optimization. For design optimizer, the members to be optimized need to be submitted in scalar form, which later will be optimized in the iterations based on the method of optimization.

4.3 Design Optimization

The component and its thickness is mentioned below: -

Component	Thickness (mm)
T1	3.125
T2	4
T3	5
T4	6
T5	7
T6	8
T7	9
T8	10
T9	12
T10	18
T11	33

Table 1: Component Number and Thickness.

The individual component results are presented in the following table;

		SET 17 (FEASIBLE)	*SET 18* (FEASIBLE)
MAXS	(SV)	25.319	25.317
MAXD	(SV)	0.51300E-01	0.49878E-01
T1	(DV)	3.1095	3.1092
T2	(DV)	1.0100	1.0071
T3	(DV)	1.0133	1.0097
T4	(DV)	2.0163	2.0100
T5	(DV)	6.0321	5.2203
T6	(DV)	3.0107	3.0104
T7	(DV)	5.9540	3.5008
T8	(DV)	6.9027	6.5532
T9	(DV)	4.0447	4.0220
T10	(DV)	17.660	17.657
T11	(DV)	12.045	12.043
WT	(OBJ)	397.71	393.40

Table 2: Optimized Design Sets.

		SET 18 (FEASIBLE)
MAXS	(SV)	25.317
MAXD	(SV)	0.49878E-01
T1	(DV)	3.1092
T2	(DV)	1.0071
T3	(DV)	1.0097
T4	(DV)	2.0100
T5	(DV)	5.2203
T6	(DV)	3.0104
T7	(DV)	3.5008
T8	(DV)	6.5532
T9	(DV)	4.0220
T10	(DV)	17.657
T11	(DV)	12.043
WT	(OBJ)	393.40

Table 3: Best Design Set.

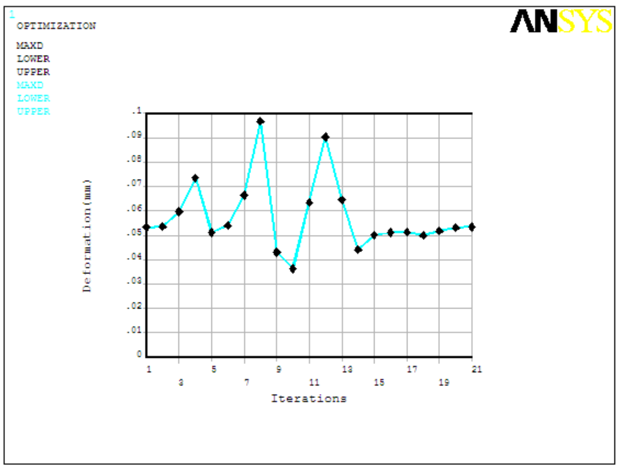

Fig 10: Iterations vs. Deformation.

The graph shows final convergence for the deformation. During 7 to 13 iterations, more change in the deformation can be observed.

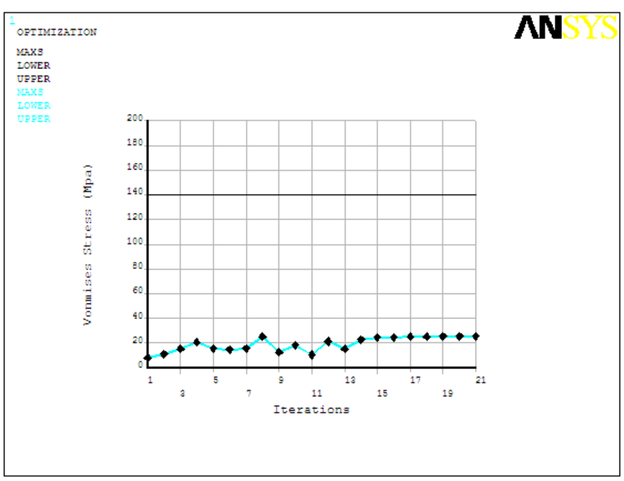

Fig 11: Iterations vs. Stress (Mpa).

The fig 11 shows almost minimum changes in the stress conditions. Mainly fixtures are designed for deformation rather the stress conditions.

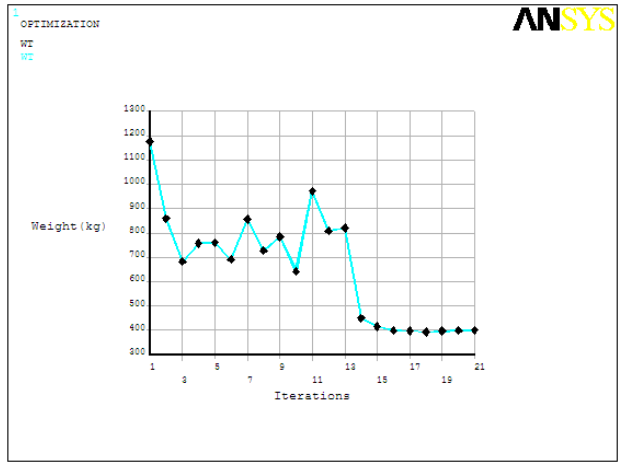

Fig 12: Iterations vs. Weight.

The fig 12 shows weight convergence after 13th iteration. From 13 to 21, no variation of weight can be observed in the graph.

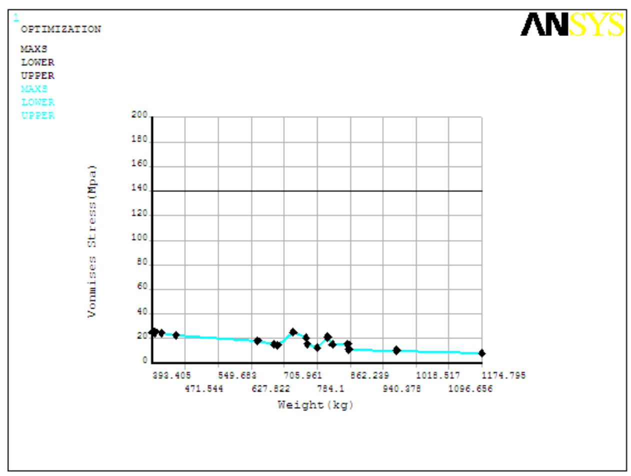

Fig 13: Weight vs. Von-Mises Stress.

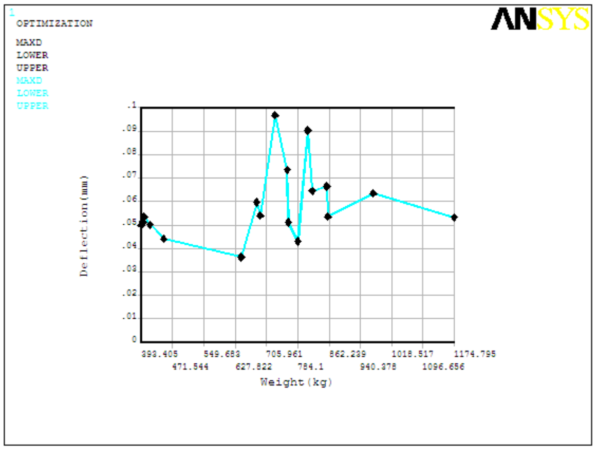

Fig 14: Weight vs. Deflection (mm).

The fig 14 shows influence of weight on deformation. More changes can be observed in the middle iterations.

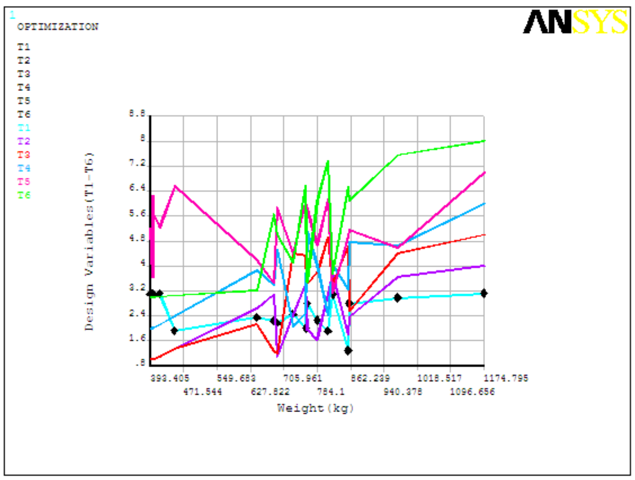

Fig 15: Weight vs. Design Variables (T1-T6).

The fig 15 shows final increase of all design parameters for obtaining the required structural stability.

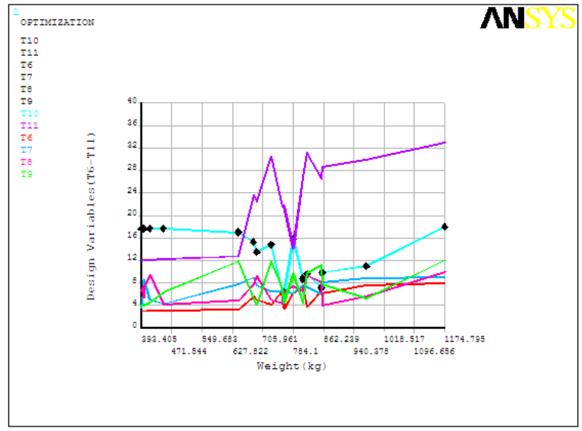

Fig 16: Weight vs. Design Variables (T7-T11).

The fig 16 shows higher influence of design variable 11 on the weight convergence.

Component Number	**Initial Stress (Mpa)**	**Final Stress (Mpa)**
1	7.412	4.184
2	26.045	67.627
3	9.005	41.906
4	14.072	47.026
5	8.872	13.538
6	22.96	77.448
7	4.032	11.367
8	1.801	0.624
9	2.038	4.853
10	0.2	0.031
11	21.8	52.3292
Complete Assembly	26.045	67.627

Table 4: Comparative Results for Components.

Component Number	**Initial Deformation (mm)**	**Initial Weight (kg)**
Initial Configuration	0.382	1174
Final Configuration	2.891	393

Table 5: Comparison for Deformation and Weight.

The variations of stresses and deformations can be attributed to change of self-weight in the structure even though external load is constant throughout the process.

4.4 Modal Analysis

Modal analysis is carried out to find the dynamic stability of the impeller mount structure. The modal frequency helps in finding the resonant condition and along with mode shapes. These mode shapes helps in finding the weaker direction of the problem.

Set No	Frequency(Hz)
1	5.2849
2	5.2937
3	6.4561
4	11.53
5	11.81
10	15.216
20	24.395
30	29.871
40	36.035
50	39.946
60	43.98
70	50.285

Table 6: Modal Frequencies.

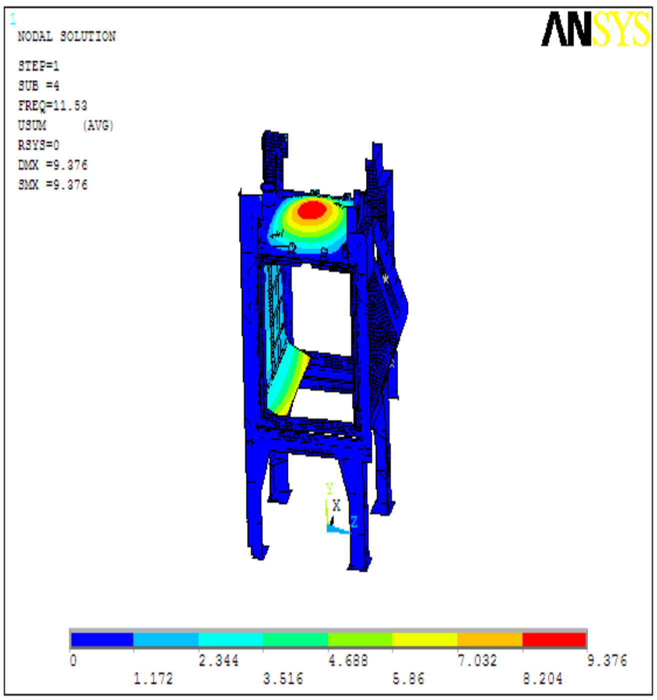

Fig 17: Mode Shape Corresponding to Natural Frequency of 11.53 Hz.

The mode shapes show direction of weakness for the structure. Two mode shapes are obtained whose frequencies are near the operational frequency of the impeller system.

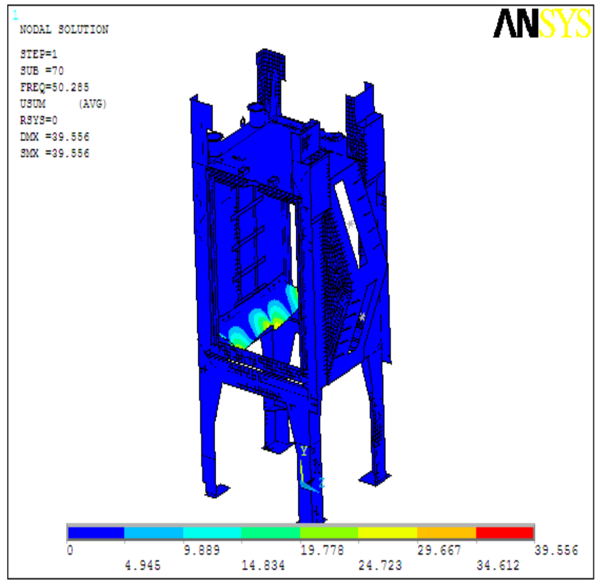

Fig 18: Mode Shape 70 Corresponding to natural Frequency of 50.285 Hz.

The fig 18 shows almost rigidity in the mount as almost blue color can be observed in the system excepting one attached region. Due to overhung, this region is showing deformation. This region is not import and its purpose is to collect only chip formation. So the structure is rigid enough to take the load and system is dynamically stable.

5. CONCLUSIONS

The frame structure used for mounting the impellers is analyzed and optimized using finite element analysis. The analysis summary is as follows. Initially geometry is built based on the two dimensional AutoCAD representation using Solid Edge software. Certain critical geometries are checked for sufficiency of thickness for the given loads. The assembly is shell meshed for design optimization. Shell63 element and RBE3 elements are created for analysis. Analysis is carried out for self-weight and results shows safety of the structure for the given loads. Total of 21 design sets are obtained and the influence of various parameters on the geometrical parameters are represented.

6. FURTHER SCOPE

Analysis can be extended composite material usage in the assembly. Analysis can be extended to vibrational studies Transient response of the structure can be analyzed for possible impact loads. Three dimensional analysis can be done to find the actual structural behavior. Weld strength analysis can be carried out for the problem

REFERENCES

1. R.J.Duffin, E.L.Peterson and C.Zener, Geometric programming: theory and applications, Wiley, New York.
2. G.B.Dantzig Linear Programming and Extensions, Princeton University Press.
3. R. A. Gettatly and R. H. Gallagher, "*A procedure for automated minimum weight structural design Part*".
4. I - Theoretical basis, Part II - Applications," Aero. Quart. Part I, Vol. 17, pp. 216-230 and pp.332-342.
5. M.M. Denn, Optimisation by variational methods, McGraw-Hill, New York.
6. G.S.G. Beveridge and R.S. Schechter, Optimization: theory and practice, McGraw-Hill, New York.
7. J.L. Kuester and J.H. Mize, Optimisation techniques with FORTRAN, McGraw-Hill, New York.
8. M.J. Panik, Classical Optimisation: foundations and extensions, North-Holland Publishing Co., Amsterdam.
9. D. Koo, "Elements of Optimisation" Springer-Verlag, New York