RESEARCH PAPER | ENGINEERING | Volume : 4 | Issue : 7 | July 2014 | ISSN - 2249-555X

Design Optimization of Impeller Supporting Frames Using Finite Element Analysis

KEYWORDS: Impellers, design optimization, Hypermesh, modal analysis, finite element analysis.

Yashavantha Kumar M.p.	Dr. MOHAMED HANEEF
M.Tech student ,MECHANICAL ENGINEERING	Principal &Professor, Mechanical Engineering Dept. , Ghousia College Of Engineering, Ramanagara.

ABSTRACT *Impellers plays important role in the mixing equipment for industrial applications.Impellers are generally rotating structures.So a platform is required to install these members for its functioning.In the present work, a supporting frame of impeller need to be optimized for its weight satisfying the functional requirements.Initially the load calculations are done for the given design specifications.The geometry is built using Catia software and imported to Hypermesh for better quality mesh.The meshed file is imported to Ansys for further design optimisaition.According to the results, considerable stress of 15Mpa is observed for static weight of 1100kgs,which is less than the allowable stress of the material.Later the modal analysis is carried out to find the dynamic stability.Ansys subproblem approximation technique is used for design optimisation.Design sets are obtained and the optimizer has given the best set.*

I. INTRODUCTION

An impeller is a rotor inside a tube or conduit to increase the pressure and flow of a fluid. Also defined as, impeller is a rotating component of a pump, usually made of iron, steel, aluminum or plastic, which transfers energy from the motor that drives the pump to the fluid being pumped by forcing the fluid outwards from the center of rotation. Impellers are usually short cylinders with protrusions forming paddles to push the fluid and a splined center to accept a drive shaft A design process is the set of technical activities within a product development process that work to meet the marketing and business case vision[2]. This includes refinement of the product vision into technical specifications, new concept development, and embodiment engineering of the new product. Design tasks may be classified in several different ways. To indicate the extent of the effort required, one approach is to classify a development of project as:

1.Original design or inventing: It involves elaborating original solutions for a given task. The results of original design are an invention. Few successful original designs occur over time, and when they do, they can disrupt the market. Original inventions are often high-risk opportunities for changing marketplace and then dominating it.

2.Adaptive design: It involves adapting a known system to a changed task or evolving a significant subsystem of a current product. Adaptive design can be very novel, but they do not require a massive restructuring of the system within which the product operates because customers generally want new products that fit in their current life style.

3.Variant design: Redesign implies that a product already exists that is perceived to fall short in some criteria, and a new solution is needed.

It is commonly necessary to feed back information from the later stage in the design process in order either to modify the original need or to improve the modeling of the physical situation. i.e. design is an iterative process[7]. The design process for engineering products can use a block diagram form as shown in figure 1.

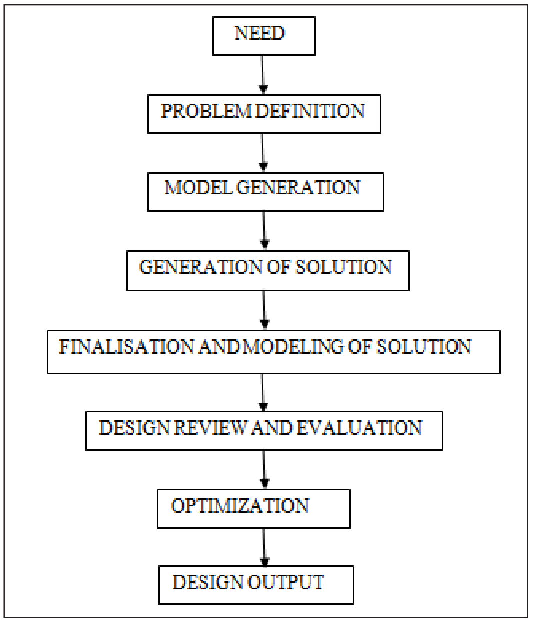

Figure 1: Block diagram of the engineering design process

II METHODS AND METHODOLOGY

STRUCTURAL ANALYSIS: Structural analysis is probably the most common application of the finite element method[5]. The term structural (or structure) implies naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools. A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

Static analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed;

Volume : 4 | Issue : 7 | July 2014 | ISSN - 2249-555X

that is, the loads and the structure's response are assumed to vary slowly with respect to time. The types of loading that can be applied in a static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements
- Temperatures (for thermal strain)
- Fluences (for nuclear swelling)

The procedure for a static analysis consists of these tasks: Build the Model, Set Solution Controls, Set Additional Solution Options, Apply the Loads, Solve the Analysis, Review the Results.

FINITE ELEMENT METHOD: Most often the mathematical models results in algebraic, differential or integral equations or combinations thereof. Finite Element Method is one of the numerical methods of solving differential equations. The FEM originated in the area of structural mechanics, and has been extended to other areas of solid mechanics and later to other fields such as heat transfer, fluid dynamics and electromagnetic devices[6]. In fact FEM has been recognized as a powerful tool for solving partial differential equations and integral-differential equations. One of the reasons for FEM's popularity is that the method results in computer programs versatile in nature that can be used to solve many practical problems with least amount of training. Obviously there is a danger in using computer programs without proper understanding of the theory behind them, and that is one of the reasons to have a thorough understanding of the theory behind the finite element method.

HYPERMESH: Altair® Hyper mesh is a high-performance finite element pre- and post-processor for major finite element solvers, allowing engineers to analyze design conditions in a highly interactive and visual environment. hyper mesh's graphical user interfaces easy to learn and supports the direct use of cad geometry and existing finite element models, reducing redundancy. advanced post-processing tools that ensure complex simulations are readily visualized and easily understood. hyper mesh offers unparalleled speed, flexibility and customization.

ANSYS: Ansys is a general purpose finite element program that can be applied to varied range of engineering problems varying from linear to nonlinear to transient. Ansys directly interfaces with the pre-processing module of 3-d interactive color graphics macro (prep7) with extensive modeling capabilities for finite element model generation and problem definition.

PROBLEM DEFINITION: Static analysis and design improvement of Impeller Supporting Frame for Harmonic Response and damper level estimation is the main definition of the problem. The objectives include

- Modeling of the impeller support frame structure
- Meshing and analysis
- Checking for the dynamic stability
- Checking for damping levels required in the problem

Methodology:
- Literature on support structures and frames
- Three dimensional modeling
- Meshing with Hypermesh for good quality elements to obtain better results
- Analysis for static loads
- Harmonic response analysis
- Improvement analysis

Material : Structural Material :St42 , Yield stress: 420 N/mm^2 , Factor of Safety : 6 (Variable load) , Allowable stress: 70N/mm^2 , Poison's ratio=0.3 , Density =7800kg/m^3.

III RESULTS
The impeller frame has been modeled and analyzed for static and harmonic conditions to find the structural safety of the member. The analysis results are as follows.

Static Analysis is carried out in two stages. In the first stage analysis is carried out for self weight and later carried out for impeller load. Few times self weight also plays important role in failure of the components due to its overweight. So it is always better to start with any analysis with self weight. The results are as follows.

Case 1 : Self Weight Analysis

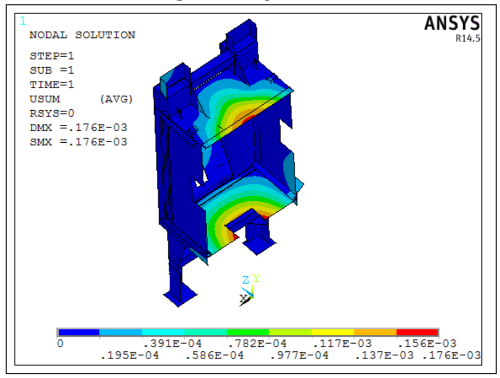

Figure 2:Deflection due to self weight

The figure2 shows deflection due to self weight as 0.176mm or 0.000176m. Maximum deflection location is represented at the bottom and top of the supporting frame.

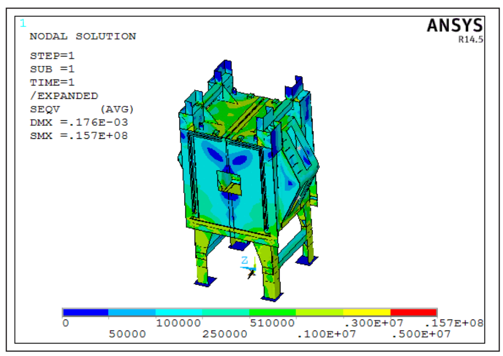

Figure 3: Vonmises stress distribution due to self weight

The results shows maximum stress of 15.7 Mpa due to self weight. This stress is much smaller then the allowable stress of the material.

Case 2 : Static analysis for the given loads

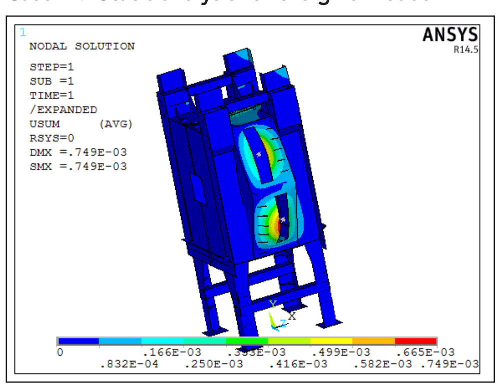

Figure 4: Deflection under external load

Volume : 4 | Issue : 7 | July 2014 | ISSN - 2249-555X

The figure 4 shows maximum deflection of 0.749mm or 0.000749m. This deflection is mainly observed near the loading region. The maximum deflection is shown with red color region.

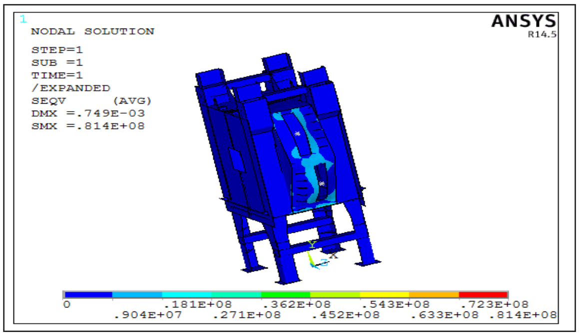

Figure 5: Vonmises Stress generation due to external static loading

Maximum stress is observed as 81.4Mpa and is concentrated near the weld regions. Vonmises stress is the most important stress corresponding to stored energy in the system due to loading.

Case 3 : Modal Analysis

Modal analysis is the critical analysis of the components mounted with rotating members. Generally resonance is the most critical design aspect for maintaining stability of the members. Resonance will take place in the systems, if its natural frequency match with resonant frequencies. Generally a system is said to be more stable, if its resonant frequency is much higher than the operational frequencies. In most of the mechanical structures, if first natural frequency is higher than the 30Hz, it is taken as rigid system free of dynamic vibration problems.

Table 1: Modal Frequencies

Set No	Frequency (Hz)
1	16.091
2	30.507
3	32.291
4	47.741
5	50.178

The table shows modal frequencies of the supporting frame. The fundamental mode frequency is higher than the normal operation frequency of 6Hz. So system is dynamically stable and will not resonate as the natural frequency is much higher then the operational frequency. Even considering damping also, the system natural frequency will not come down to 13Hz(For 20% variation due to damping).

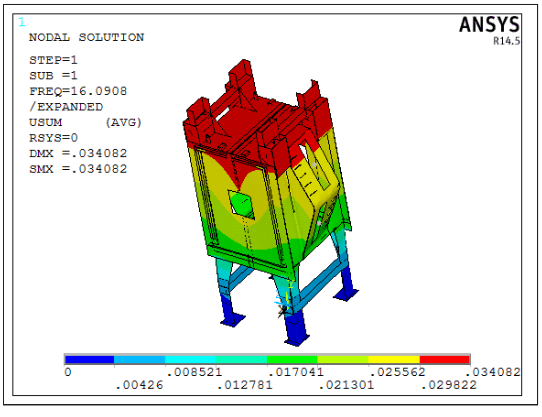

Figure 6: Mode Shape corresponding to first natural frequency (16Hz)

The figure6 shows mode shape of the supporting frame for the first natural frequency. Maximum deformation is taking place on the top structure compared to the bottom constrained region. In reality, the figure shows proportional deformations, this helps in finding the weaker side of the problem for proper constraint.

In the present design optimization, total of 12 geometrical variables are considered to reduce the weight of the structure. In ansys upto 120 design variables can be specified. Similarly 2 state variables are considered(Stress and deflection).

		SET 31 (FEASIBLE)	*SET 38* (FEASIBLE)
MAXD	(SV)	0.20089E-02	0.20102E-02
MAXS	(SV)	0.40714E+08	0.40714E+08
T1	(DV)	0.50411E-02	0.50386E-02
T2	(DV)	0.50996E-02	0.51260E-02
T3	(DV)	0.51533E-02	0.51142E-02
T4	(DV)	0.37321E-02	0.37123E-02
T5	(DV)	0.54053E-02	0.53462E-02
T6	(DV)	0.51686E-02	0.54981E-02
T7	(DV)	0.50154E-02	0.50151E-02
T8	(DV)	0.36222E-02	0.36288E-02
T9	(DV)	0.30243E-02	0.30212E-02
T10	(DV)	0.41868E-02	0.41825E-02
T11	(DV)	0.50216E-02	0.50199E-02
T12	(DV)	0.10170E-01	0.10170E-01
WT	(OBJ)	472.50	471.58

Table 2: Optimised Design Sets

		SET 38 (FEASIBLE)
MAXD	(SV)	0.20102E-02
MAXS	(SV)	0.40714E+08
T1	(DV)	0.50386E-02
T2	(DV)	0.51260E-02
T3	(DV)	0.51142E-02
T4	(DV)	0.37123E-02
T5	(DV)	0.53462E-02
T6	(DV)	0.54981E-02
T7	(DV)	0.50151E-02
T8	(DV)	0.36288E-02
T9	(DV)	0.30212E-02
T10	(DV)	0.41825E-02
T11	(DV)	0.50199E-02
T12	(DV)	0.10170E-01
WT	(OBJ)	471.58

Table 3 : Best Set

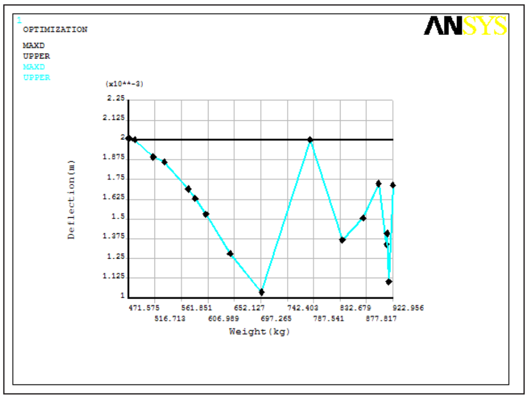

Figure 7 : Deflection to Weight

Volume : 4 | Issue : 7 | July 2014 | ISSN - 2249-555X

The figure 7 shows deflection variation with reference to weight. The results shows varying displacement with weight variation. Generally the deflection and weight are inversely proportional as the weight reduces the deflection will increase.

IV CONCLUSIONS

The structural optimization is carried out for impeller supporting frame for the given loads and overall summary is as follows.

- The analysis for displacements and stresses results shows 0.1mm displacement and 15Mpa stress in the problem; these stresses are less compared to the allowable stress limits of the problem. The Modal analysis results shows obtained initial natural frequency of 16Hz is much higher than the operational frequency of 6Hz, hence the frame is dynamically rigid for resonant conditions. The design optimizer in the Ansys is used with 12 design variables, 2 state variables with one objective function. The design optimizer with subproblem approximation technique has given the best set after iterating for 50 sets. Only feasible sets are presented in the problem. The best set has given an 471 kgs against initial weight 1160kgs. Also it has represented the thickness required to lower the weight satisfying the deflection and stress limits.

FUTURE SCOPE:

- Composite material usage can be checked as the composites gives better strength to weight ratio[1]. Topology optimization can be carried out along with size optimization. Spectrum analysis can be carried out as the machine is built in the shop floor. Possible thermal effects can be considered for finding the residual stress in the frame which will affect the strength of the overall machine structure.

REFERENCE

[1].Dai Gil Lee, Hak Sung Kim, Jong Woon Kim, Jin Kook Kim "Design and manufacture of an automotive hybrid aluminum/composite drive shaft" Composite Structures, Volume 63, Issue 1, January 2004, Pages 87-99 | [2]. Robert L. Norton, " Machine design an integrated approach" second edition PEARSON Education publication, page No 588-591. | [3]. Joseph E Shigley and Charles R Mischke "Mechanical Engineering Design," sixth edition, TATA Mc GRAW HILL Publication, page No 1102. | [4]. P.L Srinivasa Murthy and J.B.K Das, "Design of Machine Elements I and II", fourth revised edition April 2009. | [5].Vijayarangan, N. Ganesan"Static stress analysis of a composite bevel gear using a three-dimensional finite element method"Computers & Structures, Volume 51, Issue 6, 1994, Pages 771-783. | [6].Farouk O. Hamdoon "Application of a Finite Element Package for Modeling Rotating Machinery Vibrations"Engg & Tech Journal, Vol.27, NO.12, 2009. | [7]. K.Lingaiah, "Machine Design Data Hand Book," Volume I and II. |