Finite Element Analysis and Optimization of Mixer Blade of Sand Preparation Machine

Alok J Tandel[1]
[1]P.G. Student (CAD/CAM)
[1]A. D. Patel Institute of Technology, New Vallabh Vidhyanagar - 388120, Gujarat (India)

Abstract— Finite element analysis has been widely used for analyzing the component and optimizing component size and shape. In this research paper we will analyses the mixer blade under static structural analysis by using ansys and will study the stress and deformation induced in the component. After, analysis of stress and deformation we will optimize the weight of the blade using different thickness of the plate and check the stress and deformation induced.

Key words: Finite Element Analysis (FEA), Weight Reduction, Mixer Blade, Ansys 14.0, Static structural analysis

I. INTRODUCTION

The sand preparation machine is basically used in the foundry industries. They are used for preparation of moulding sand which is used for mould making for casting. In these machines the moulding sand, betonies, water, saw dust is mixed through the use of mixer blade and rotor. The process of mixing the sand mixture is carried out in a controlled manner in the pan by the mixer blade and rotor which are rotated in the pan and continuous mixing is carried out. This paper will deal with finite element analysis of mixer blade and optimization of the weight of the mixer blade. Finite element analysis of the blade is carried out in Ansys 14.0 after the modelling of blood in core 2.0, then after analysis, optimization of the weight of the blade will be done modelling the component with different thickness of the blade and analyzing the same component in Ansys 14.0. The material for component considered is structural steel.

Sr. No.	Parameters	Descriptions
1	Material	Structural steel
2	Density (ρ)	7850Kg/m^3
3	Yield stress	250N/mm^2
4	Ultimate stress	460N/mm^2
5	Young Modulus(E)	2.1x10^{5}N/mm^2

Table 1: Material Properties

II. OBJECTIVES

The main objective is to carry out finite element analysis of the blade by 3d modelling of the component and then analyzing the von-mises stress and total deformation induced by static structural analysis and then optimization of weight of the same.

III. 3D MODELLING OF THE BLADE

3d modelling of the blade was done on the 3d modelling software. 2D AutoCAD drawing was available and was used to model the blade as per the dimensions. First the part drawing of the component of the blade was modelled then the part was assembled to form the complete 3-d modelled blade. The 3d modelled blade is shown in the fig. 1

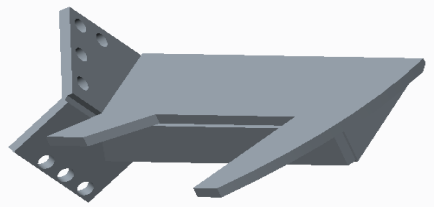

Fig. 1: 3d modelling of blade

IV. ANALYTICAL CALCULATIONS

- Batch Capacity = 1500kgs.
- No. of Blades = 3nos.
- Blade speed = 33RPM.
- Rated Torque = 48500NM.

The forces acting on the blade are:
Ft= Force due to the torque applied to the blade = 23303 N
Fv= Total vertical force acting on blade = 6978 N
Fi= Inertia force = 821.68 N

A. Permissible Deformation Calculation:

Considering the blade as a Cantilever beam with uniformly distributed load
Maximum Deflection = wl^4/8EI
= 1.52mm.

V. STATIC STRUCTURAL ANALYSIS:

A. Meshing Of the Blade:

The selection of meshing element is very important as the element represents the actual properties of material. The meshing of the blade was done by a hex dominant method with default mesh size. The total no of nodes and element are 32880 and 8303. The meshing of the blade is shown in fig. 2.

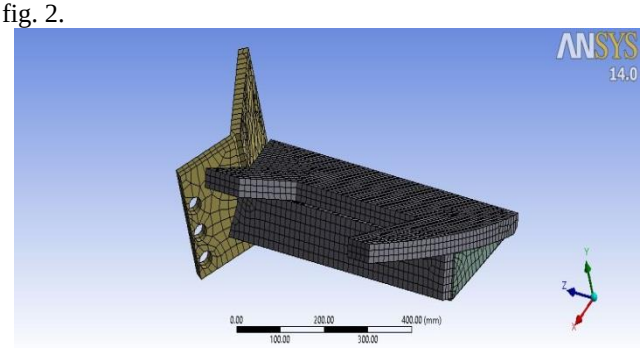

Fig. 1: Meshing of Blade

1) Assumption:

All three blades is assumed symmetrical. All the face connection of bodies is considered bonded. All the forces are considered to be acting on face of blade. All the material for the blade is assumed to be structural steel. The material is considered isotropic.

B. Loading Condition:

All the three loads are applied on the face of the blade. The vertical force is applied in a downward direction, the torque force is applied in the direction of rotation of the blade and The Inertia force will be acting in the outside direction. The loading condition is shown in the fig 3.

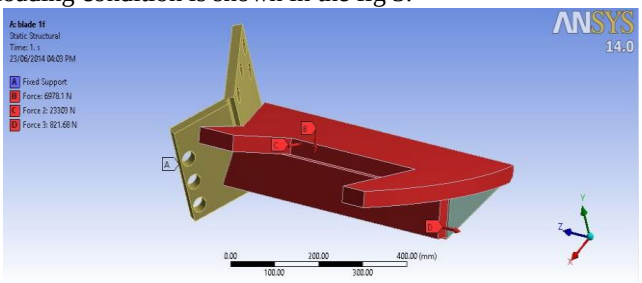

Fig. 3: Loading Conditions

C. Static Structural Analysis Results:

A static structural analysis with given loading conditions and assumptions is carried out in Ansys solver. The two static analysis result is considered for current problem they are Equivalent (Von-mises) Stress and Total Deformation. The Result are shown below in fig. 4. & 5

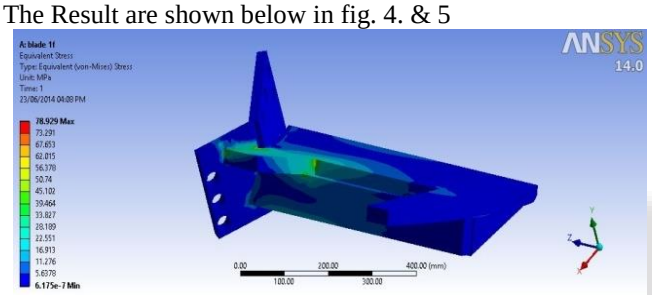

Fig. 2: Equivalent (Von-mises) stress with original blade

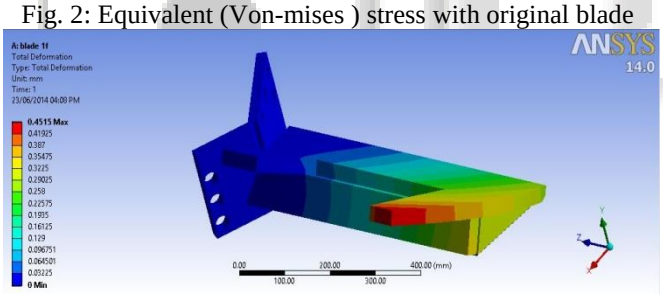

Fig. 3: Total deformation with original blade

1) Result interpretation:

Maximum equivalent (von-mises) stress induced in blade is 78.929 MPa in a very small region and Maximum total deformation observed is 0.4515 mm which are well within the permissible limit of stress and deformation so from the result we can conclude that optimization of the blade through the use of smaller thickness plate is possible as the stress and deformation are well within permissible limits.

VI. Optimization of Weight of Blade

As we can conclude from the original blade static structural analysis that the weight of the blade can be optimized we will carry out the optimization using different thickness of plate available in the market. Some steps in the optimization process are discussed.

A. Design Modification (25mm Plate).

Now the blade is modified by using 25mm plate and the static structural analysis is carried out with the original conditions in ansys. The result is shown in the fig 6. & 7

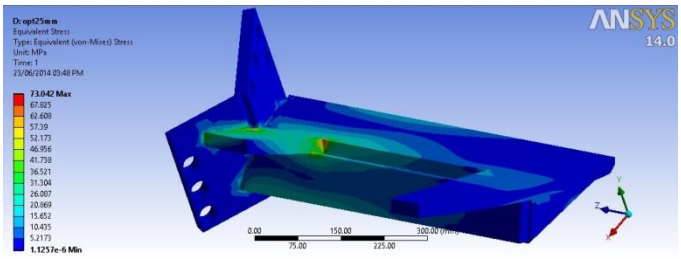

Fig. 4: Equivalent (Von-mises) Stress with 25mm plate

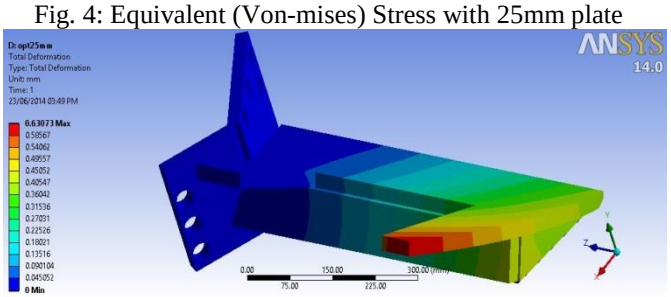

Fig. 5: Total deformation with 25mm plate

1) Result Interpretation:

The Maximum Equivalent (von-mises) stress induced is 73.042 Mpa and total deformation is 0.6307 mm. Which are well within permissible limit of stress and deformation so further optimization with smaller thickness is to be done.

B. Design modification (20mm Plate).

Now the blade is modified by using 20mm plate and the static structural analysis is carried out with the original conditions in ansys. The result are shown in the fig 8. & 9

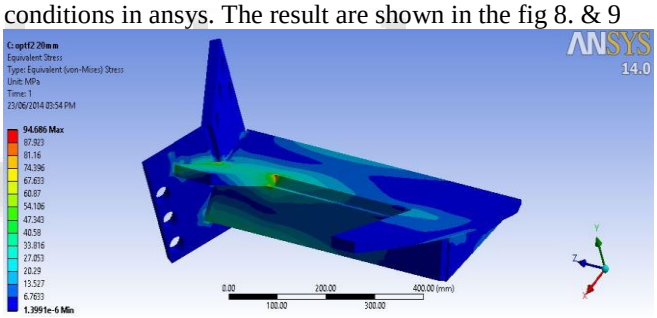

Fig. 6: Equivalent (Von-mises) Stress with 20mm plate

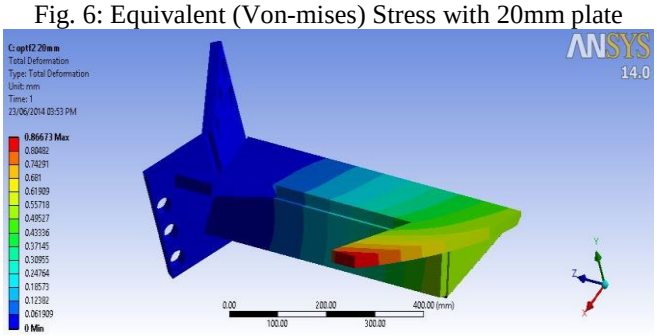

Fig. 7**:** Total deformation with 20mm plate

1) Result interpretation:

The maximum equivalent (von-mises) stress induced is 94.680 Mpa and total deformation is 0.8667 mm. Which are also well within permissible limit of stress and deformation so further optimization with smaller thickness (18mm) of the blade to be done.

C. Design Modification (18mm Plate).

Now the blade is further modified by using 18mm plate and the static structural analysis is carried out with the original conditions in ansys. The result is shown in the fig 10. & 11

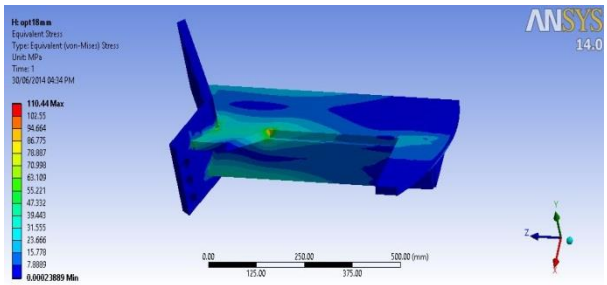

Fig. 10: Equivalent (von-mises) stress with 18mm plate

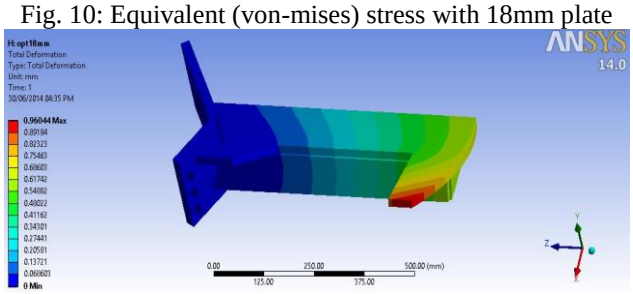

Fig. 11: Total deformation with 18mm plate

1) Result interpretation:

The maximum equivalent (von-mises) stress induced is 110.44 Mpa and total deformation is 0.9604 mm. Which are also well within permissible limit of stress and deformation so further optimization with a smaller thickness of the blade (15mm) to be done.

D. Design Modification (15mm Plate).

Now the blade is further modified by using 15mm plate and the static structural analysis is carried out with the original conditions in ansys. The result are shown in the fig 12. & 13

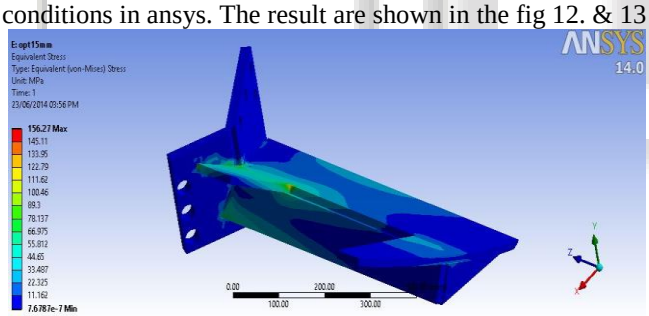

Fig. 12: Equivalent (Von-mises) stress with 15mm plate

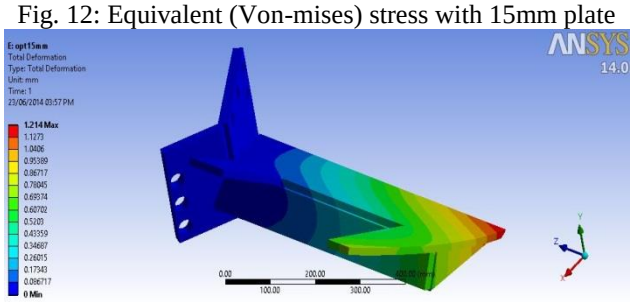

Fig. 13: Total deformation with 15mm plate

1) Result interpretation:

The maximum equivalent (von-mises) stress induced is 156.27 Mpa and total deformation is 1.214 mm. Which are also well within permissible limit of stress and deformation so further optimization with a smaller thickness of the blade (12mm) to be done.

E. Design Modification (12mm Plate).

Now the blade is modified by using 12mm plate and the static structural analysis is carried out with the original conditions in ansys. The result is shown in the fig 14. & 15

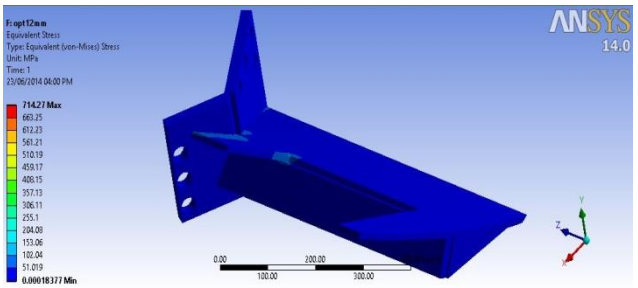

Fig. 14: Equivalent (von-mises) stress with 12mm plate)

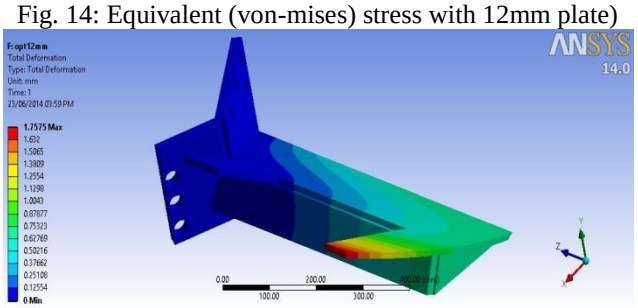

Fig. 8: Total deformation with 12mm plate

1) Result interpretation:

The maximum von-mises stress induced is 714.27 Mpa in a very small region and total deformation is 1.7575 mm. Which is not within permissible limit of Stress(250 Mpa) and Deformation(1.52 mm) so further optimization cannot be done and the optimization process is stopped and the previous step of design modification using 15mm plate is considered final as the stress and deformation value of the blade using 15mm plate is well within permissible limits.

VII. COMPARISON FOR DIFFERENT THICKNESS OF PLATE.

A. Total Deformation Vs Thickness Of Plate

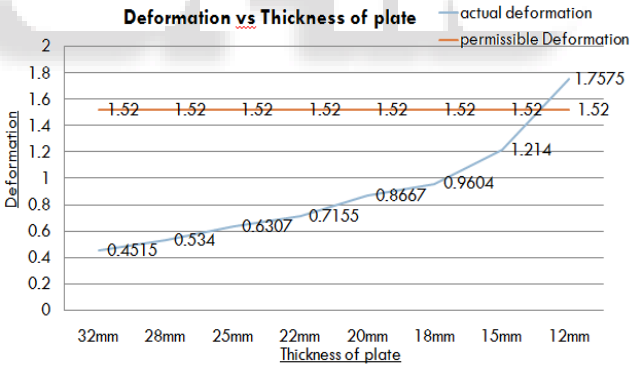

Fig. 16: Deformation Vs Thickness of plate

B. Equivalent Stress Vs Thickness of Plate

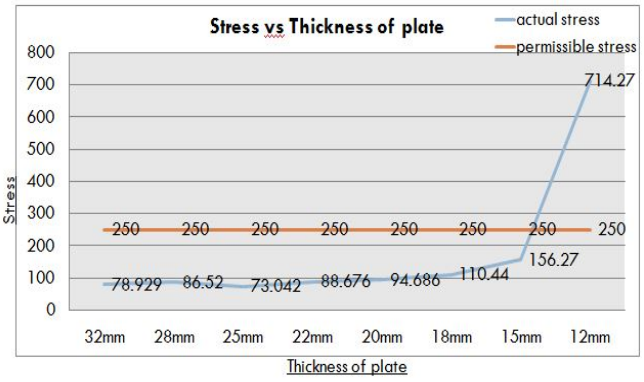

Fig. 17: Equivalent stress vs. Thickness of plate)

1) Interpretation:

As seen from the graph the stress value and deformation values goes beyond the permissible value when 12mm plate is used and analyzed. So further analysis was stopped and 15mm plate was considered as the optimal size of plate for the blade.

C. Comparison of Original Blade And Modified Blade

	Original Blade(32mm)	Modified Blade(15mm)
Overall weight	84.172 Kg	51.328 Kg
Equivalent (von-mises) Stress	78.929 Mpa	156.27 Mpa

Table 2 Comparison of Original & Modified Blade

VIII. CONCLUSIONS

- The original blade static structural analysis shows that the maximum equivalent (von-mises) stress induced well within permissible limit and also the total deformation induced well within permissible limits. Thus, optimization of the blade can be carried out.
- The optimized weight of the new modified blade is 51.33 Kg (which is a 39 % weight reduction of original blade).
- The maximum equivalent (von-miss) stress induced in the modified blade is 156.27 Mpa and total deformation induced is 1.21 mm which is well within permissible limits.

REFERENCES

[1] Kunkel Wagner (India) Pvt. Ltd. "Manual for WM-30 sand preparation machine".
[2] Kunkel Wagner (India) Pvt. Ltd. "Part List of WM-30 sand preparation machine" Part list no-849-00.
[3] Kunkel Wagner (India) Pvt. Ltd. "Drawing of WM-30 sand preparation machine" Drawing no. 849_00-91-01.
[4] Steven M Lepi, Practical guide to finite element: a solid mechanic approach, Publisher: New York: Marcel Dekker, ©1998. Cook, R. D.
[5] Moaveni, S. Finite Element Analysis: Theory and Application with ANSYS, Prentice-Hall, Upper Saddle River, NJ, 1999, 2003.

