INTERNATIONAL JOURNAL OF PURE AND APPLIED RESEARCH IN ENGINEERING AND TECHNOLOGY

A PATH FOR HORIZING YOUR INNOVATIVE WORK

STRUCTURAL OPTIMIZATION OF HOOPING PLATEN OF 400TON COTTON BALING PRESS FOR COST REDUCTION

SATISH G SONWANE[1], S D KSHIRSAGAR[2], NITIN BAGOKAR[3], CHETAN MAHATME[1]

1. PG student, YCCE Nagpur, Nagpur, India.
2. Mech. Eng. dept., YCCE Nagpur, Nagpur, India.
3. Bajaj steel industries Pvt. Ltd., Nagpur, India.

Accepted Date: 27/02/2014 ; Published Date: 01/05/2014

Abstract: This Structural optimization tools and computer simulations have gained the paramount importance in industrial applications as a result of innovative designs, reduced weight and cost effective products. In this paper we took the ‘hooping platen’ of 400 ton cotton baling press as research object, established its 3D model through CATIA software. Then carried out finite element analysis for it by using FE software ANSYS 13.0 to gain the stress and strain distribution of the platen. The detailed analysis of the results was then carried out. At the end, shape optimized design model is compared with the actual part that is being manufactured for the press. It is inferred that topology optimization results in a better and innovative product design. It also serve as guidance to further improve its performance with less cost; at the same time provide a theoretical basis for the structural design of the same type of platens.

Keywords: FE analysis, Modeling, shape optimization, Simulation, Structural

Corresponding Author: MR. SATISH G SONWANE

Access Online On:

www.ijpret.com

How to Cite This Article:

Satish Sonwane, IJPRET, 2014; Volume 2 (9): 379-389

PAPER-QR CODE

INTRODUCTION

Bale packaging is the final step in processing cotton at the gin. The bale press consists of a frame, one or more hydraulic rams, and a hydraulic power system. The system in consideration is a 400 ton cotton baling press, a product of Baja steel Industries ltd. MIDC Nagpur.

Bale press is used to make Bales of cotton. Weight of one bale ranges from 170-200 Kg. Process occurs in two stages. Compression occurs in large box-like structure called 'Lint box'. The complete structure rests on two steel sills. The presses range from 8 bales per hour (bph) to 60 bph. As the hydraulic ram compresses the cotton the platen bears a tensile load of about 3924 KN. Thus platen undergoes repeated stress cycle of 0 KN to 3924 KN. Officials perceive a chance of overdesign of the platen.

This is in tune with the fact that today, all the modern manufacturing enterprises are striving to develop best optimized reduced weight and cost effective products that meet the intended design functionality and reliability. This result in an innovative design proposal irrespective of dependency of the designer experience and conventional design approaches. The figure shows one of the platen which is placed above bottom sills (i.e. press bed) of the press.

Figure 1: Hooping Plate

LITERATURE REVIEW

The research on machine tool structures was stepped up by the application of the finite element method (FEM) [6-9]. This is a more generalized method in which a continuum is hypothetically divided into a number of elements interconnected at nodal points to calculate the strain, displacement and stress. The FEM is preferred because it permits a much closer topological resemblance between the model and the actual machine. It has been only recently employed for press structures [11]. Due to the diversification of structural optimization problems, most structural optimization problems can be classified as size, shape and topology optimization. The main application of optimal design of steel structures is the size optimization, because this method is possible to minimize the weight of structures [4].

To obtain results of acceptable accuracy and reliability, it is required to have a very fine mesh of finite elements but this enhances the core memory requirement and the cost of computation. Several techniques, such as “model-sub structural analysis” [10] and the “semi-analytical FEM” [5], have been developed and used to reduce the time and cost of computation. In complex structures like hydraulic press welded frames, the concept of a substructure cannot be applied. The semi-inalytical FEM is ideally suited for structures of regular geometric shape.

In the history of fatigue research, many approaches [1, 2] for the prediction of fatigue life of notched elements have been developed. Most of the approaches can be classified into three types according to their assumptions: nominal stress approach (NSA), local stress–strain approach (LSSA) and stress field intensity approach (SFIA). We followed NSA method as it is the most traditional method, with much published data available. It is easiest to implement for a wide range of applications and represent high cycle applications adequately [12]. Mostly, these methods integrate knowledge based systems with computer-aided design (CAD) and other tools [13][14] to estimate manufacturing cost and facilitate automated process planning. However, structural performance and manufacturing cost were treated as disparate entities. Structural shape optimization reduces material, but it may be accompanied by increase in the geometric complexity due to movement in the design boundary, which ultimately increases manufacturing cost [17]. The main contribution of this paper is that it recognizes the trade-off between reduction in the material cost due to structural optimization and the increase in the manufacturing cost due to increase in the geometric complexity. This paper proposes a design process that incorporates manufacturing cost into structural shape optimization and produces cost-optimum components that satisfy specified structural performance requirements.

RESEARCH METHODOLOGY

The steps of optimization approach using topology optimization can then be stated as,

- Identify the design space for the analyzed body
- Create the topology optimization model
- Formulate the optimization problem based on design requirements
- Determine magnitude of load on the platen at the time of bale compression

- Perform topology optimization
- Determine Von mises stresses and Structural factor of safety
- Determine total deformation
- Create an optimized design based on the optimization results.

Fe analysis

Simplification of the 'Platen' Model

According to the structural features and working conditions, its structure model was simplified as follows:

1. Assume that the platen is a constant linear system, and ignore the influence of damping
2. The materials of the frame are isotropic material with uniform density, and the platen is a fully elastic body
3. The Materials, loads and physical shape of the platen have symmetry

Objective function: Weight Minimization

Constraints: - Equivalent stresses

Parameters: - By changing the thickness of plate (size optimization) Changing the Material (Design optimization) [15] [18]

The element selection is made from three categories of elements: 1. Axi-symmetric solid elements 2. Shell/plate elements 3. 3-D brick elements. Although nearly all problems can be solved using 3-D brick elements, the other two types offer significant reductions in the solution time and effort where they are applicable, Therefore a four node quadratic shell Elements is selected [9] [12].

Creating FE model

CATIA V5R19 is used to create the model because the model is too complex to be created by ANSYS 13.0. In order to save computer resources and calculate time, axi-symmetric model is considered. During structural shape optimization, updating the FEA mesh may cause mesh distortion due to movement of the design boundary (Hwang et al. 1997). In this research, p-version FEA is employed considering the following characteristics [12].

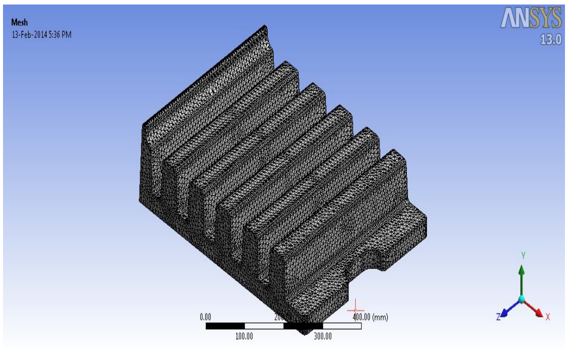

Figure 2: Meshed Model

B. Defining constraints and loads

I) *Define symmetry constraints*: Contact surfaces of platen are defined as Z=X=Y=0

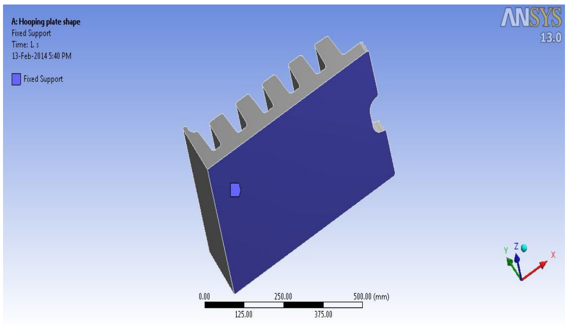

Figure 3: Fixed Support

2) Output:

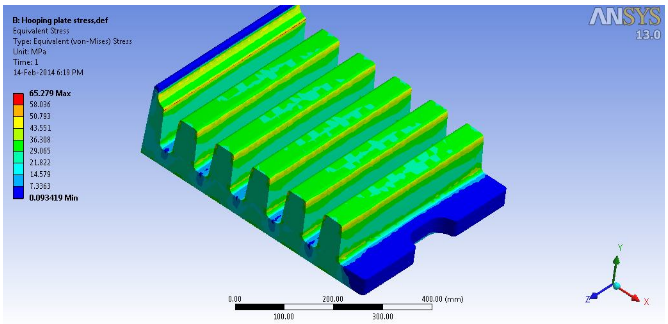

Figure 4: Von mises stress

Figure 4 shows that he max von mises stress of the rod is far less than the materials yield limits in the working condition.

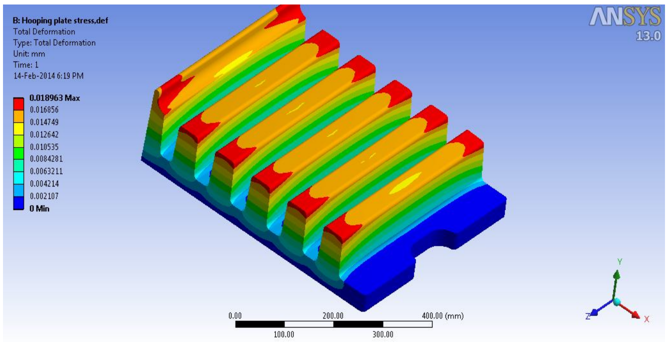

Figure 5: Total deformation

Figure 5 shows that the maximum deformation is 0.02 mm which is negligible.

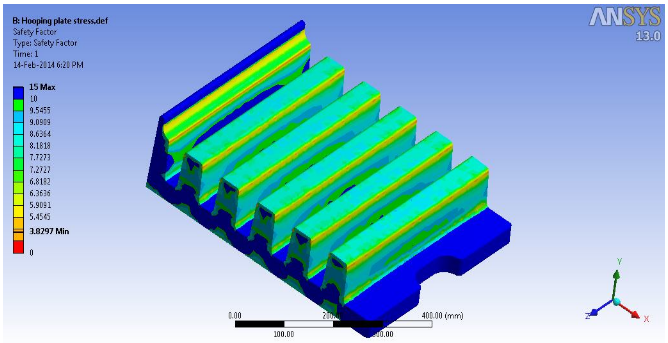

Figure 6: Factor of safety

Figure 6 shows that the minimum factor of safety is 3.8.

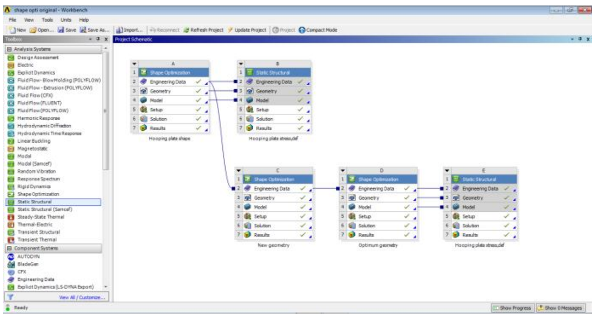

Figure 7: Solver Window

The figure 7 shows the solver window in Ansys workbench 13.0 environment. One can see output of one analysis can be conveniently utilized as an input to next set of analysis.

C. Shape optimization

Structural optimization is defined as an automated generation of mechanical components based on structural properties, or as a method that automatically generates a mechanical component design that exhibits optimal structural performance. Size optimization involves a modification of the cross section or thickness of finite elements. The optimization is carried out by mathematical optimization algorithms with different objective functions e. g. maximum stiffness or minimum weight.

Compared to size optimization, shape optimization is more complex. The coordinates of the surface are regarded as design variables which will be modified during the optimization. The resulting component shape is optimally adjusted to the strains resulting from the specified loads and boundary conditions. Thus the reliability and life of a component can be increased. [19]

The optimization of the platen is carried out using shape optimization module in Ansys 13.0. The optimization focused on the uncritical sections which need to be reduced. From the shape optimization study, the software suggested the unnecessary shape and design of the platen. The results of shape optimization of the platen are shown in Figure 8.

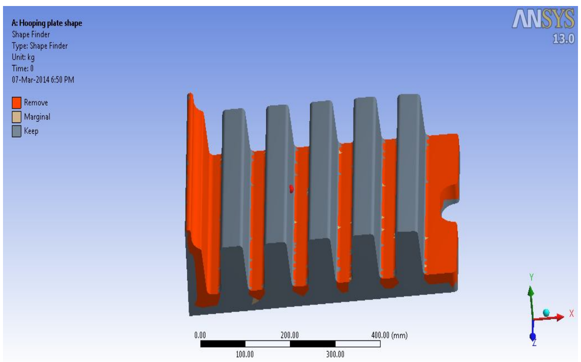

Figure 8: Shape optimization result

The main objective is to minimize the weight of the platen as well as the total production cost. It can be seen that the optimized model is reduce the weight from initial design until the value converges. The implementation of these optimizations is to find out the best design and shape of the platen to improve the performance and the strength especially at the critical location.

The possible modification section of the optimized platen is indicated in the figure 8 the section with lower value than initial value considered as the suggestion to be optimized in the new design.

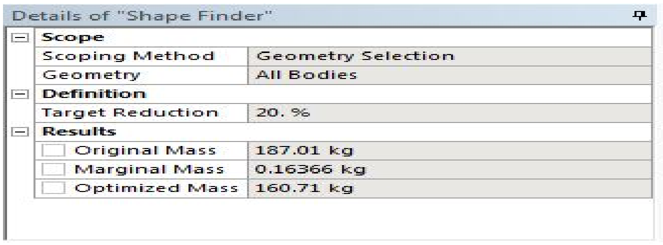

Figure 9: Details of shape finder

Table 1 and 2 shows the comparison between initial and optimize designs on max principles stress and mass of the platen. The optimized platen was chosen as the better design due to the lower value of stress and mass and hence the cost.

TABLE I: Weight comparison

Sr. no.	Percentage reduction	Mass (Kg)
1	Original	187.01
2	20	160.71

TABLE II: Maximum Von mises stress comparison

Sr. no.	Percentage reduction	Max stress (MPa)	Deformation (mm)
1	Original	65.978	0.01424
2	20	64.082	0.0156

As the figure 9 shows if the new model is again tested for Shape optimization we find that there is again a scope for material removal. However considering manufacturing process of the platen, further reduction is not advisable as it would result in a thin section resulting in increased casting complexity.

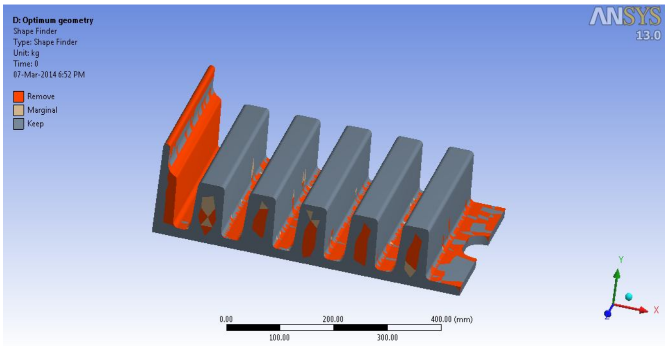

Figure 10: Further material reduction possible

D. RESULTS

1. Maximum stress of 64.082 MPa occurs at the projected portion of the platen.
2. Hence the Factor of safety comes out as equal to 3.8. This value is much higher than what is essentially required in such application.
3. 20% mass reduction was possible which ultimately results in proportionate cost saving.
4. Maximum Von mises stress levels do not vary appreciably hence the new design can be considered safe.
5. Same applies if we consider maximum deformation of the platen.

CONCLUSIONS

1. The results can provide guidance to further improve platen's performance, and at the same time provide a theoretical basis for the structural design of the same type of automatic hydraulic press' platens.
2. The maximum deformations of the platen is far less than the allowable value, so the stiffness of the entire platen can meet the design requirements
3. The max von-mises stress of the platen are far less than the materials yield limits at major locations in working condition, and there is a lot of optimization design space, can consider reducing materials and reduce costs.
4. Change in material keeping major portion of the design same can also be a viable option. This would result in cost saving.

5. As per the results, it can be concluded that the weight of optimized design is up to 20% lighter and maximum stress also predicted lower than the initial design of platen.

The results clearly indicate that the new design much lighter and has more strength than initial design of platen. Material optimization approach will be considered for future research.

ACKNOWLEDGMENT

The authors gratefully acknowledge the support of my guide Prof. Mr. SD Kshirsagar and the faculties of Mechanical Engineering Department of our college. Also the technical detailing and product know-how was greatly facilitated by Mr. Nitin Bagokar, the industry personnel from Bajaj Steel Pvt. Ltd.

REFERENCES

1. Zheng LC, Yao WX. A review of approaches for predicting fatigue crack life. Mechanics and Practice (1996)

2. Dugan TV, Byrne J. Fatigue as a design criterion. Macmillan Press Ltd, (1977).

3. Shigley, Mischke ‘Mechanical engineering design’ Eighth edition

4. B.Tadic, P.M.Todorovic, B.M. Jeremic "Failure analysis and effect of redesign of a polypropylene yarn twisting machine" Department for Production Engineering, 34000 Kragujevac, Serbia.(2011)

5. L. S. Gupta and P. U. Ramanamurti, Stress analysis of 400-tonne hydraulic press cross-head. Proc. AZMTDR Conf. p. 579-585 (1982).

6. Cowley and M. Fawcett, Analysing machine tool structures by computing techniques. Proc. 8th Inf. MTDR Conf. p. 119 (1967).

7. Cowley andand S. Hinduja, The finite element method applied to the deformation analysis of a thin walled column. Proc. I2th Int. MTDR Conf, p. 455 (1971).

8. A Cowley and S. Hinduja, The finite element method for machine tool structural analysis. Ann. CIRP 9, 171 (1971).

9. A. Zienkiewicz, The Finite Element Method. T. M. H. Publishing, New Delhi (1979).

10. K. Yip, M.Sc. Thesis, UMIST, Manchester, Lanes. (1976).

11. Hwang HY, Choi KK, Chang KH (1997) Shape design sensitivity and optimization using p-version design modeling and finite element analysis. Mechan Struct Mach 25(1):103–137

12. Szabó B, Babuˇska I (1991) Finite element analysis. Wiley, New York

13. Shehab EM, Abdalla HS (2002b) A design to cost system for innovative product development. Proc Inst Mech Eng B J Eng Manuf 216(7):999–1019

14. Khoshnevis B, Park J, Sormaz D (1994) A cost-based system for concurrent part and process design. Eng Econ 40(1):101–124

15. LeRoy Fitzwater , Richard Khalil , Ethan Hunter "Topology Optimization Risk Reduc-tion" Presented at the American Helicopter Society 64th Annual Forum, Montreal, Cana-da, April 29 – May 1, 2008.

16. H.T. S´anchez, M. Estrems, F. Faura "Analysis and com-pensation of positional and deformation errors using in-tegrated fixturing analysis in flexible machining parts" Department of Mechanical and Automation Engineering, Da-Yeh University, Changhua 51591, Taiwan (Received 16 September 2008; Revised 5 March 2009)

17. A. Alkin, C.E. Imrak. H Kocabos "Solid Modeling and FEA of on Overhead Crane Bridge" Acta Polytechnica Vol.45 No.3/2005.

18. Osama Bedair "Dynamic analysis of box girders with tee-stiffening us-ing unconstrained optimization techniques" Received: 3 August 2009 / Revised: 20 De-cember 2009 Struct Multidisc Optim (2010) 42:547–55.

19. 'Structural optimization in cad software' nenad marjanovic biserka isailovic mirko blagojevic , Machine design ISSN 1821-1259.