STRUCTURAL WEIGHT OPTIMIZATION OF A BRACKET USING ANSYS

Prashant Kumar Srivastava[1], Rajive Gupta[2], Simant[3]

[1]*Asst Professor, Department of Mechanical Engg., Ambalika Institute of Management &Technology, Lucknow*
[2]*Professor, Department of Mechanical Engg., HBTI, Kanpur*
[3]*Director, Aryavart Institute of Technology and Management, Lucknow*

Abstract

The paper deals with the design optimization of a bracket with Finite Element Analysis using ANSYS software. A procedure has been developed for the optimum design of a bracket in ANSYS software. The function design emphasize that structure will not fail, but some aspects like dimensions of structure can also be improved. In the paper, the volume of a bracket is optimized. A mathematical model is developed for the bracket based on its volume. It consists of objective function, design variable and state variable. ANSYS software is used to find the result in which the safe designing will be done by the help of the optimization algorithm. A series of the analysis were carried out searching for the minimum volume by changing the dimensions each time. A best design set is obtained from many feasible design sets. The best design set gives the best dimensions of the chosen parameter resulting in the optimized volume at the same strength. The optimized volume resulting the saving in structural weight.

Keywords: *Design optimization, FEM, ANSYS, Design set, Optimized volume*

---***---

1. INTRODUCTION

Now days, every manufacturing organization is striving to focus in lowering the production costs and reduction in material weight. Material saving can be achieved only through optimizing the volume of the shape. The engineers are prompted to look for rigorous methods of decision making such as optimization. As a result, engineering optimization was developed to help engineers design systems that focuses lot on efficiency & economy and to develop innovative methods to improve the performance of the existing systems.

Engineering design optimization can be classified as one of the best mathematical approach to identify and select a best set from the set of probable design alternatives. For problems involving complex shapes it is difficult and in many cases intractable to obtain analytical solution that satisfies the governing differential equations given extreme value to the governing functions. Hence for most of the practical problems, numerical method provides appropriate but acceptable solutions. Numerical solutions to even very complicated stress problems for a complex shaped bracket can now be obtained routinely using FEA (Finite Element Analysis), and the method is so important that even introductory treatments of Mechanics of Materials of complex shaped brackets. Here the bracket is optimized by FEM and ANSYS software and the best design set would be found. ANSYS is a general-purpose finite-element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis (both linear and nonlinear), heat transfer, and fluid problems, as well as acoustic and electromagnetic problems .In this paper ,we are focusing on structural analysis and optimization of a bracket against given constraints .

2. OPTIMIZATION

Optimization is a mathematical discipline that concerns the finding of minima and maxima of functions, subject to so-called constraints. Optimization originated in the 1940s, when George Dantzig used mathematical techniques for generating "programs" (training timetables and schedules) for military application. Since then, his "linear programming" techniques and its descendents were applied to a wide variety of problems, from the scheduling of production facilities, to yield management in airlines. Today, optimization comprises a wide variety of techniques from Operations Research, artificial intelligence and computer science, and is used to improve business processes in practically all industries. The primary goal of the optimization is to minimize /maximize the objective function. In this paper, our objective function is volume of the bracket. In the work presented here volume is optimized, results in the saving of material of the bracket. Now days, reduction in material weight is primarily focused to reduce the overall production cost.

3. PROBLEM DESCRIPTION

A bracket is constructed from structural steel, which has a modulus of elasticity of 200 GPa and Poisson’s ratio of 0.3 as shown in figure. The Bracket is loaded by a pin with a total force of 7.5 kN. The Force is approximately distributed along the contact as a tapered pressure varying linearly along the lower half of the circumference as shown in figure. The dimensions of the bracket (shown in the figure below) are to be optimized in order to minimize the volume of the bracket without exceeding a maximum von Misses stress of 105 MPa. The total load 7.5 kN is distributed along the lower half of the circumference. The maximum pressure is calculated will be 58.90MPa

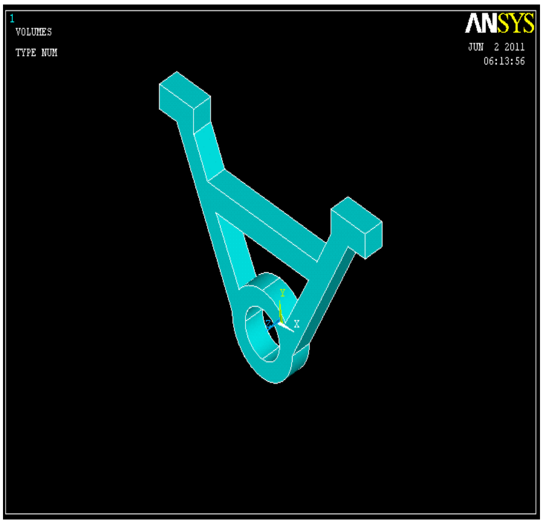
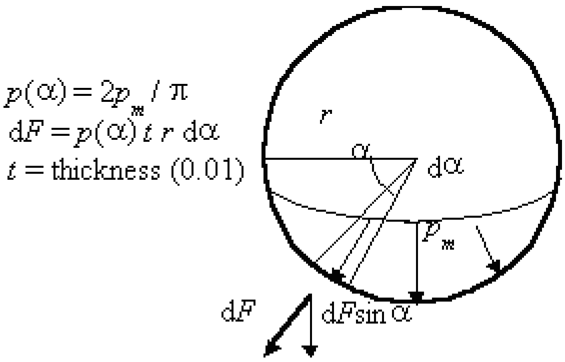


Fig 1 Modeling by ANSYS

$$F_y = 2\int_0^{\pi/d} \frac{2p_m \alpha}{\pi} tr \sin \alpha d\alpha$$

$$\Rightarrow p_m = \frac{\pi F_y}{4tr} = 58.90\ Mpa$$

Where
Fy =7.5 kN
t=10 mm
r = 10 mm



3.1 Design Variables:

The Bracket will have the following scalar parameters (or DVs) with the minimum and maximum values as in Table 1.

Table 1

Scalar Parameter	Min Value (mm)	Max Value (mm)
Width (W)	8	12
External radius (R2)	15	20
Height of the rib (A)	33	37
Rib Thickness (T)	8	12

State Variable:
The maximum von Mises equivalent stress (Smax) will be limited to 105 MPa

Design Objective:
Minimize the volume of the bracket

4. MATHEMATICAL FORMULATION

The design optimization problem for the bracket is:

Minimum of Volume ,V=f(W,R2, A,T,)

Objective Function
[Volume=1065.21W+400W+Π(R2^2-10^2) W+2(A+T/2+13.846) (13/24) W T, mm^3]
[±100]

Subjected to
Von-mises stress,

Vvm ≤ 100 MPa *State variable*

8 ≤ W ≤ 12	*Design variable*
15 ≤R2 ≤ 20	*Design variable*
33≤ A ≤ 37	*Design variable*
8≤ T≤ 12	*Design variable*

5. RESULT AND DISCUSSION

Fig. 2 shows the algorithm to optimize the volume of the bracket;

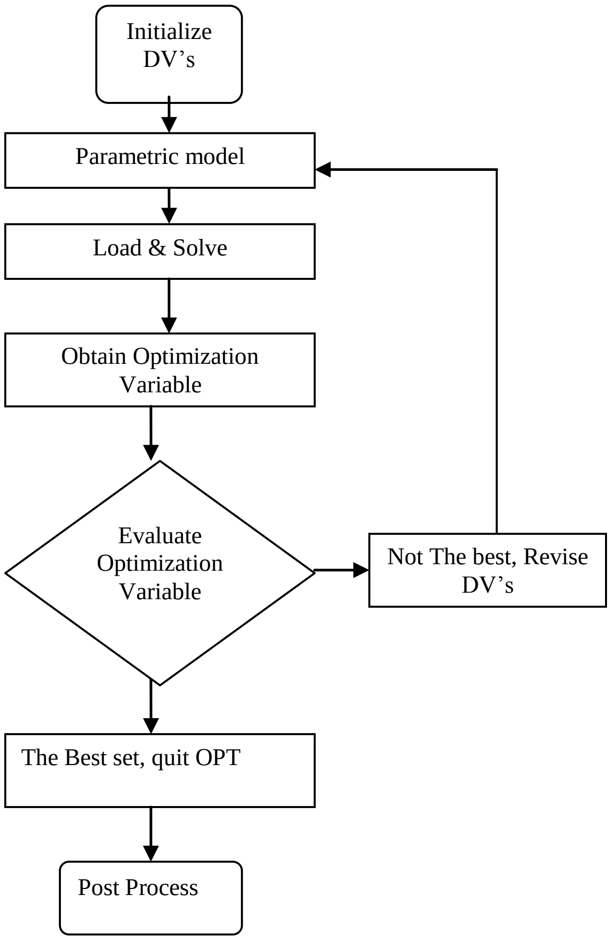


Fig 2 Algorithm for optimization of bracket using ANSYS

The Sub Problem Approximation method is employed to solve this problem. This is an advanced zero order method, which requires only the values of the dependent variables (state variables and the objective function), and not their derivatives. It is a general method, which can be applied efficiently to a wide range of engineering problems. There are two concepts that play a key role in the sub problem approximation method: the use of 'approximations' for the objective function and the 'conversion' of the constrained optimization problem to an unconstrained problem. State variables and limits on design variables are used to constrain the design and make the optimization problem a constrained one. The ANSYS program converts this problem to an unconstrained optimization problem because minimization techniques for the latter are more efficient. The conversion is done by adding penalties to the objective function approximation to account for the imposed constraints.

A list of the optimization result sets obtained after 15 iterations is shown where SET 15 represents the best solution. The histories of the objective function; the state and design variables are depicted in Fig. 4, 5 and 6 where a convergence of the solution can be observed. Table 2 shows the initial and final values of the variables after the complete optimization iterations:

Table 2

Result	**Initial value**	**Final Value**
S_{Max}	104.79	104.79
R2	17.5	17.845
T	10	8.3340
W	10	8.0081
A	35	34.425
VOLUME	26886 mm^3	20830 mm^3

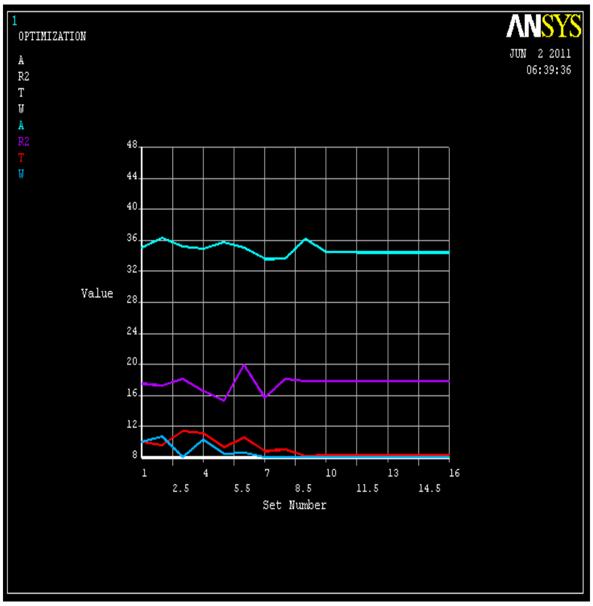


Fig 4: History of the design variables

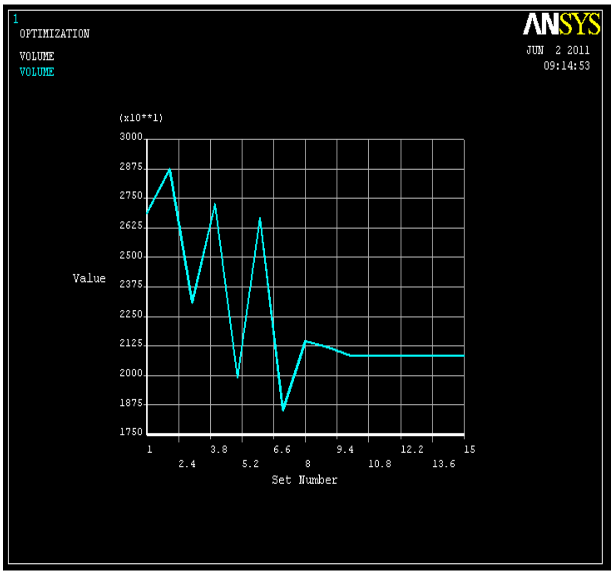


Fig 5: History of the objective function

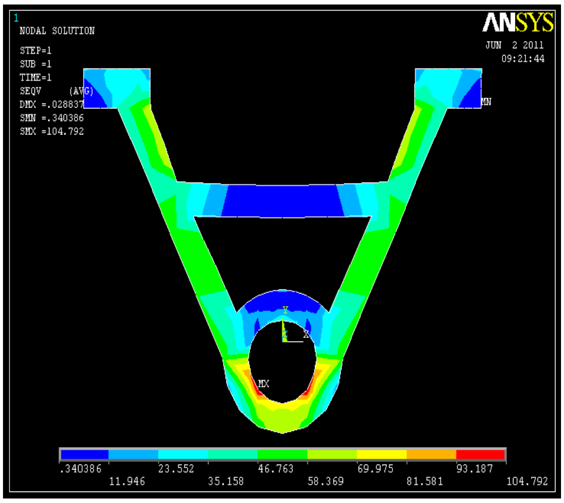


Fig 6: Von Mises stress distribution in the final geometry

6. CONCLUSIONS

The design optimization helps in reducing 22.5% of the structure weight (the initial and final volumes of the bracket are 26,886and 20,830mm^3, respectively), which in turn reduces the cost of the bracket with safe design.

REFERENCES

[1] Ding Y., Shape optimization of structures: a literature survey; Department of Aeronautical, Structures and Materials, computers & Structures Vol. 24. No. 6. pp. 985-1004. (1986)

[2] X-duan, Sheppard T, Shape optimization using FEA software-A v-shaped anvil as an example, Journals of Material Processing technology vol no 120 pp 426-431(2002).

[3] Hongwei Z., Xiaokai C., Lin Yi, Mingjun Z., Topology Optimization for Air Suspension Bracket Integrated Shape Optimization; IEEE Vehicle Power and Propulsion Conference (VPPC), Harbin, September 3-5, China (2008),

[4] Yong-hai W., Xiao-mei . & Feng W , Topology Optimization Design of the Bracket of a Special Vehicle Based on FEA Method; Second International Conference on Computational Intelligence (2010)

[5] Jian M., Yong-Qi L., Rui-xiang L. and Bin Z., 3-D Finite Element Analysis on 480 Diesel Crankshaft; School of Traffic and Vehicle Engineering Shandong University of Technology, China IEEE (2010)

[6] Shuguang G., “ANSYS engineering application and its analysis” Beijing China Machine Press,.4 pp. 35–85.(2003)

[7] Xin L. H., Qi Ping &, Hua Yu, The finite element analysis of cantilever support in foundation pit; International Conference on Information Management, Innovation Management and Industrial Engineering, China (2008).

[8] Huali, H.L. and Xieqing H. L., Structure-strength of Hook with Ultimate Load by Finite Element Method, Proceedings of the International Multi Conference of Engineers and Computer Scientists Vol II IMECS 2009, March 18 - 20, 2009, Hong Kong(2009)

[9] Rao S. S., Optimization Theory & Applications, Wiley Eastern limited, INDIA

[10] Deb K., Optimization for Engineering Design (Algorithms and Examples), Prentice-Hall of India Private Limited, New Delhi, INDIA

[11] Dukkipati, Rao & Baht, Computer Aided analysis and design of Machine Elements, New Age Int Publication, India