GEOMETRICAL OPTIMIZATION OF BEARING HOUSING BASE FOR THROUGH FEED BUFFING MACHINE BY USING HYPERWORK'S OPTISTRUCT SOFTWARE.

A.R. Mohite [1], P.J.Sawant[2]

[1]*Student, P.G. Department of Mechanical Engineering, SGI, Kolhapur, Maharashtra, India*
[2]*Assistant Professor, Dept. of Mechanical Engineering, SGI, Kolhapur, Maharashtra, India*

Abstract - *In this paper, the forces acting on base has been calculated with considering factor of safety. CAD model of Base has been carried out using Solidworks software. The static analysis as well as optimization of fix Base has been carried out in Hyperworks. Finally manufacturing casting validate on machine.*

Keywords: *Hyperworks, Topology optimization, Optistruct, Through feed buffing machine Base.*

1.INTRODUCTION

All modern manufacturing companies are finding new techniques to develop a cost effective product with optimized weight by using CAD/CAM/CAE. CAE is cost effective technique to obtain final product through optimization of design. In the present scenario weight reduction has been the main focus of machine tool manufacturers, because reduction in weight reduces the cost of the product. Reduction of cost by 100 rupees of one unit will results in lot of profit in mass production units.

In automobile and aerospace sector, geometrical optimization has become an integral tool of product design and development. design is enhanced by structural optimization tools like geometrical optimization, shape optimization or size optimization. Geometry (topology) assisted design model gives optimized results which are better and innovative in terms of product design with improved structural performance and stability. In other words, simulation saves time, reduces costs, and strengthens the competitiveness and market position. Also Rapid Prototyping has been used over traditional manufacturing for the pattern making process which helps reducing material wastage and time associated with tooling. Rapid Prototyping allows a complex shape to be produced directly and automatically from its 3D CAD model usually within hours with or without the aid of skilled labor. It is suitable and economical for small parts which are required in small numbers. This optimization delivers a design concept which gives a basic idea about from where to remove material, where to place (stiffening) ribs, etc. After the optimization results have been interpreted and smoothed (in the CAD system) and analysis is carried out to verify the optimized components performance. Based on these results, another optimization iteration may follow, addressing questions such as how to change main geometry in order to reduce stress peaks or how thick do the stiffening ribs really need to be.

By using this Structural optimization method we are going to develop bearing housing base & cap with optimized Structural design so we can reduce product cost. This paper deals with 3D modeling of Base, calculation of forces, Analysis, optimization, getting IGES model from Optistuct (OSsmooth) and Make appropriate change in that model for manufacturing and reevaluate displacement and stresses within model.

1.1 PROBLEM DEFINATION

Through feed buffing machine is an important machine tool in leather processing industry. This industry want to optimize the weight of some parts in Through feed buffing machine without changing material, due to competition in market. Presently they need to optimize the weight of roller housing base & cap. In this roller housing assembly base and cap are two separate parts in which base weighs19 kg and cap weighs 13 Kg made of Grey Cast Iron. Here objective is to reduce the weight of Bearing housing base & cap by optimized geometrical design.

1.2 OBJECTIVES

The specific objectives of the research work are as described below,

A. To optimize the geometrical design of the roller housing assembly of through feed buffing machine using topology optimization method without reduction in load bearing capacity.

B. To manufacture physical casting of optimized roller housing assembly.

C. To reduce the weight of roller housing assembly by reducing material use.

D. To test experimentally the optimized roller housing assembly on UTM for load sustaining capacity without fail.

d. To reduce the weight of roller housing assembly by reducing material use.

2. METHODOLOGY

Prepare CAD model

↓

Meshing

↓

Run Analysis

↓

Analysis result

↓

Optimization (variable, response, constraints, objectives)

↓

Hyperview

↓

OSsmooth

↓

Prepare CAD model ←

↓

Analysis optimized model

↓

disp. & stress lower than initial model → *NO*

↓

Yes

↓

Mfg. Casting

↓

Experimental test

Fig -1. Work Flow Diagram

In above diagram show that way of project work. In order to proceed with this study various forces acting on Base were calculated. CAD model of Base designed in solid works was imported in Hypermesh for geometric cleanup and meshing. Meshed model of Base all are 3D Tetramesh (volumemesh). Tetra elements give enhanced result as compared to other types of elements, therefore the elements used in this analysis are tetra elements.

Ductile iron material was used for Base. Calculated forces and boundary conditions were applied on meshed model in Hypermesh as shown in figure 3.

Static analysis was performed by using Optistuct and results were viewed in Hyperview. All specifications, material property and results are shown in table1.

Table -1: Material Properties

Parameter	Description	fix Base
E	Young's Modulus (Mpa)	170000
NU	Poisson's Ratio	0.275
RHO	Density (kg/m3)	7100
F	Force(N) factor of safety 1.5	16326
M	Wt. of Base before optimization	19 kg

Table -2: Here we calculate total force applying on Base

FORCES ACTING ON BASE				
Sr. No.	Load Description	Load In Kg	Load In N	Considering force factor of safety 1.5
1	Roller wt.	274	274*9.81=2687.94N	2687.94N *1.5=4032N 4032/2(2 base) =2016N each base
2	Wt. of cap on base	13	13*9.81 =127.53 N	127.53*1.5= 191.25N
3	Radial load of bearing	50* 2 (bearing in each base)	50*9.81 =490.5N	490.5N*1.5= 735.75N 735.75*2=1471.5
4	Feed roller force on cylinder roller which rested on base & cap	50	50*9.81 =490.5N	490.5N*1.5=735.75N 735.75/2=367.75N

2.1 RESULT AND DISCUSSION

STATIC ANALYSIS

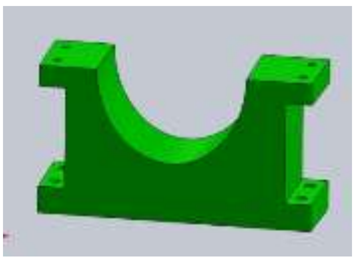


Fig -2: Static Analysis

Static analysis was performed by using Optistuct solver. It is observed that the maximum displacement Developed is 0.001mm. For Base Stress developed is 3.35N/mm^{2}. which is lower than the yield strength. Hence, design is safe i.e. The values of maximum stresses are acceptable when compared with yield strength, so design constraints for optimized Base is to maintain displacement value lower than 0.001 mm.

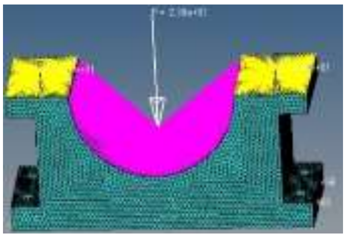


Fig -3. Base with meshing, loading and boundary Condition

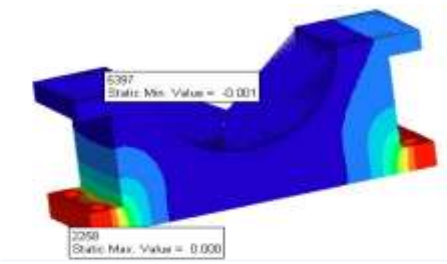


Fig -4. Displacement Results

Table 3: Analysis results of original geometry

Description	Fix Base
Max displacement in mm	0.001
Von -Mises stress (N/mm^{2})	3.35

2.2 TOPOLOGY OPTIMIZATION

Topology Optimization technique gives an optimum material distribution in given design space. This design space is defined using solid elements. The topology optimization is carried out by defining design variables and mfg. constraints. Design variable is selected as solid and setting up two manufacturing constraints

1. Minimum member size control (8 mm)
2. Draw direction type (single).

Design responses were volume fraction and weighted component displacement. Volume fraction for that upper bound was 0.3. Displacement constraint was upper bound 0.001 mm for Base.

Finally design objective was minimum weight compliance. Run optimization by using Optistuct solver. Finally viewed results in Hyperview.

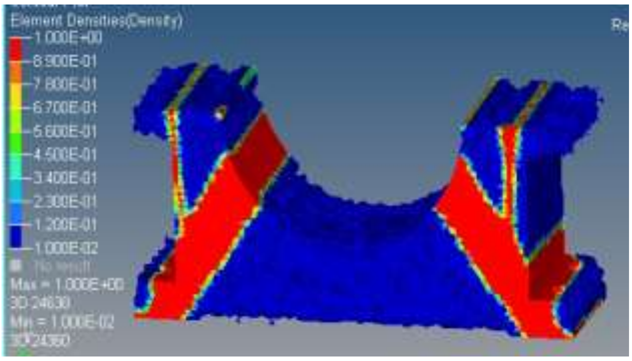


Fig 5. Hyperview element densities of Base

Figure 5 shows element density for Base. Optistruct calculate and show material distribution pattern throughout Base and remove material from that region in successive iterations based upon set of objectives and constraints. This material removal is given by varying density of each element from 0 to 1. After number of iterations, when solution converges the density pattern of component a region with lower densities indicate that it can be removed without compromising safety of component. So by removing the material from these design space (low density region) of component fulfills the objective of reducing weight of component with all design constraints.

A conceptual design was imported in a CAD system using an iso-surface generated with OSSmooth, which is part of OptiStruct. This IGES model was imported in Solidworks and changes were made as per manufacturing aspect. Figure6. Shows CAD model of modified base.

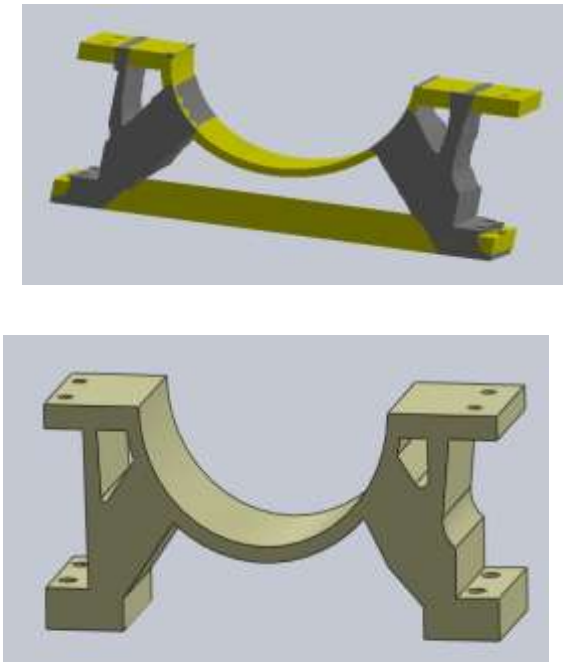
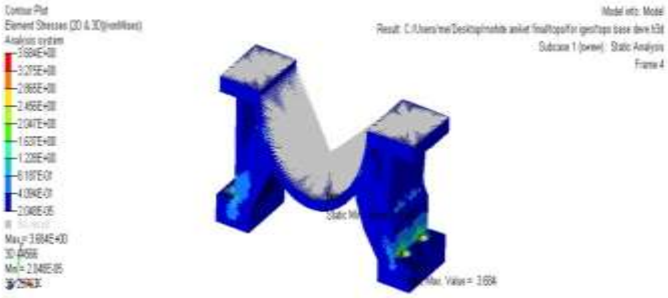


Fig-6. CAD model of optimized fix Base

Again conduct analysis on newly optimized Base model. Setup all meshing, boundary and loading condition. Cross Check that displacement and stress of optimized model, do not exceed value Initial model. Figure 7. Shows displacement result of optimized base Displacement of optimized base is 0.001mm (=0.001mm of initial model).



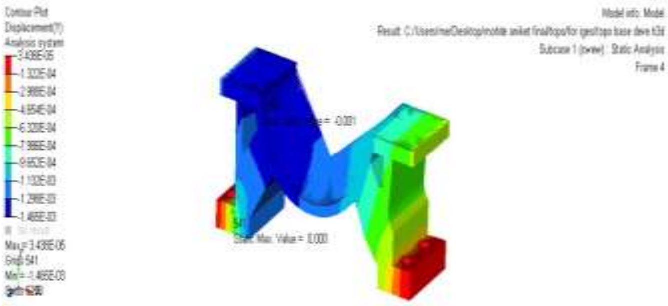


Fig-7. Displacement of optimized Base (0.001mm)

Table 4: Analysis results comparison

Parameters	Base before optimization	Base after Optimization
Max .Displacement (mm)	0.001	0.001
Max. Stress (N/mm²)	3.352	3.68
Wt. optimization (kg)	19	10.30

Displacement and stress values of optimized model are lower than in initial model.

Table 4: Total weight and cost saving

Design parameter	Base after optimized
Weight reduced (kg)	(19-10.3)=8.7kg
Total cost saving/unit	8.7*70=609/-
Total cost saved/year	609*100 = 69,000/-

3. CONCLUSIONS

In this project, the forces acting on base has been calculated with considering factor of safety. CAD model of Base has been carried out using Solid works software. The static analysis as well as optimization of fix Base has been carried out in Hyperworks. Finally manufacturing casting validate on machine. From the analyzed results, it is concluded that.

- The values obtained of the maximum displacement and Von-mises stress in optimized model are lower than original model.
- Topology optimization creates an optimized material distribution for a set of loads and constraints within a given design space. This Optimization reduces weight, manufacturing cost of component following all design constraints.
- Weight optimization of rear vice resulted to 45% of weight reduction than existing model. So that company saves 69,000/- per year.
- Though this work we can minimize load on resources and help to increase productivity and increase sale machine.

3.1 FUTURE SCOPE

We can apply this weight optimization method to all casting parts of through feed buffing machine which are over design so we can minimize total manufacturing cost of machine and it will be better to increase sale in market.

REFERENCES

1. Mohd. Viqaruddina, D.Ramana Reddy, "Structural optimization of control arm for weight reduction and improved performance." Materials Today Proceedings 4 (2017): pp. 9230–9236.
2. Wang Lu, Chen Xiao-kai, Zhao Qing-hai " Muti-objective topology optimization of an electric Vehicles traction battery enclosure." Energy Procedia 88 (2016): pp. 874- 880
3. Sandro L. Vatanabe, Tiago N. Lippi, CiceroR.de Lima, Glaucio H. Paulino, Emilio C.N. Silva, "Topology optimization with manufacturing constraints: A unified projection-based approach." Advances in Engineering Software journal. Vo.100 (2016): pp. 97–112.
4. Tushar M. Patel, Dr. M. G. Bhatt and Harshad K. Patel, "Parametric Optimization of Eicher 11.10 Chassis Frame for Weight Reduction Using FEA-DOE Hybrid Modeling." International Journal of IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) Vol.6 (2013): pp.92-100.
5. Waqas Saleem, Fan Yuqing, Wang Yunqiao, "Application of Topology Optimization and Manufacturing Simulations -A new trend in design of Aircraft components." International Multi-Conference of Engineers and Computer Scientists. vol.2 (2008): pp.19-21.