Structural Optimization of Roller Bracket of Horizontal Band Saw Machine by using Topology Optimization

Mr. Pratik R. Magdum[1] Mr. Vijay A. Kamble[2] Mr. Ajit R. Balwan[3]
[1]M.E. Student [2,3]Assistant Professor
[1,2,3]Department of Mechanical Engineering
[1,2,3]DKTE Society's Textile & Engineering Institute, Ichalkaranji, India

Abstract— Weight reduction of machine tool is an inevitable trend for machine tool manufacturing industry to meet the strategic requirements of material saving, energy saving and environment protection. This project is about Weight reduction of roller bracket of horizontal bandsaw machine by using structural optimization technique. Roller bracket used in stock bar supporting assembly in this assembly total 6 roller bracket are required. In this Roller used to carries weight of stock bar which going to cut by bandsaw machine and this roller supported to roller bracket. This work for structural design and weight reduction of Roller bracket by using topology optimization method. In this work reduce the use material for cost reduction by using Altair Hyper works software and modeled using solid works. First conducted analysis on existing bracket for getting Displacement and stress. Next Topology Optimization method. As per topology result prepared CAD model in solid works and again carried analysis on optimized model. From the validation purpose conducted experimental test on photo elastic model of optimized roller bracket on polariscopic. Here we validated stress value which is within range of software stress value. Finally manufactured casting of optimized roller bracket by 3d printed pattern. With the help of Altair CAE Optimization tools it has possible to decreased weight (min. material) of roller bracket by 27%. Total cost saving per machine 336/-.

Key words: Roller Bracket, Horizontal Bandsaw Machine, Hypermesh, Optistruct, Topology Optiomization, Photoelastic Model, Polariscope

I. Introduction

For conserve natural resources and minimize use of energy, weight reduction has been the main focus of machine tool manufacturers in the present scenario. Weight reduction can be achieved by the introduction of better material, better manufacturing processes and design optimization.

SPM TOOLS, Ichalkaranji a company of FIE GROUP. This company is leading in market for manufacturing band saw machine Main product of company is HORIZONTAL BANDSAW MACHINE. Company is interested to reduce cost of machine through a weight reduction.

The sawing machine is a machine tool designed to cut material to a desired length or contour. It functions by drawing a blade containing cutting teeth through the workpiece. The sawing machine is faster and easier than hand sawing and is used principally to produce accurate pieces of workpiece.

Sawing machine is an important machine tool of mechanical workshop. A sawing machine is a machine tool designed to Cut off bar stock, tubing, pipe, or any metal stock within its Capacity, or to cut sheet stock to desired contours. The sawing Machine functions by bringing a saw blade containing cutting. Teeth in contact with the work piece to be cut, and drawing the cutting teeth through the work piece. The sawing machine is much faster and easier than hand sawing. Sawing machine has two types:

1) Power hack saw
2) Band saw machine.[19]

Where power hack saw machine uses a reciprocating cutting Action, while band saw uses a continuous band blade for this project work to selected rear vice for weight optimization of model SPMFCAF 245 by using topology optimization method.[18]

Presently company needs to optimize the weight of roller bracket. In which roller bracket used in stock bar supporting assembly in this assembly total 6 roller bracket are required. In this Roller used to carries weight of stock bar which going to cut by band saw machine and this roller supported to roller bracket. So roller bracket is proposed for project work.

Topology Optimization is part of structural optimization and is defined as finding out the best possible material distribution in selected design space with considering the given sets of objective and design constraints. For solving any topology optimization problem have to specify parameters that are Design Variables, responses, Design objective and design constraints also consideration of manufacturing constraints.[20]

Hence in order to accomplish the objective of weight reduction over existing design, Finite Element Analysis method is used. Since from last decade a powerful FEA packages have proven good to analysis. Hence we are going to use finite element analysis software for weight optimization. In which Hypermesh is pre-processor.

Optistruct is solver which now days much famous in industry and Hyperview is Post-processor. After will do FEA analysis of optimized model and check stress, displacement of optimized model do not to exceed magnitude of initial model [1]. From the validation purpose conducted experimental test on photoelastic model of optimized roller bracket on polariscope. Here we validated stress value which is within range of software stress value.

This paper deals with 3D modeling of Roller bracket. Calculation of force, Analysis, optimization, getting IGES model from optistuct (OSsmooth) and Make appropriate change in that model for manufacturing point of view and cross-check value of stress within range.finally manufacturing casting by using 3D printered pattern.

II. Theory

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.[2] It is seen that topology optimization

All rights reserved by *www.ijsrd.com*

results gives better and innovative product design with enhanced structural performance and stability. In this scenario, structural optimization tools like topology optimization with manufacturing constraints are becoming attractive in product design processes. These tools also aid in reducing product development times. In last few years, topology optimization has emerged as the valuable tool to develop new design proposals in machine tool manufacturing industries. Topology optimization calculates the optimal loads compatible design, under specified loading and boundary conditions. [7]

The structural design process may be divided into four stages

a) Formulation of functional requirements
b) Conceptual design stages
c) Optimization
d) Detailing [21]

Today the availability of high speed digital computer has played a central role in the development of structural design and optimization so use of computer for minimize design and optimization process time and possible to solve complicated and large calculation problem. [1]

Presently Computer based Structural optimization software have been developed and interfaced with other CAE software's. Topology optimization using Altair's OptiStruct optimization software found to be very useful for generating new concept designs in less time. [11]

A. *Structural Optimization*

A structure is as "any assemblage of materials which is intended to sustain loads." Optimization means making things the best. Thus, structural optimization is the subject of making an assemblage of materials sustains loads in the best way. Structural optimization separated in three different areas: sizing optimization, shape optimization and topology optimization.

1) *Sizing Optimization*

The shape of the structure is known and the objective is to optimize the structure by adjusting sizes (Thickness) of the components.

2) *Shape Optimization*

In shape optimization the Design variables can be thickness distribution along structural members, curvature, and diameter of holes, radii of fillets or any other measure.

3) *Topology Optimization*

The most general form of structural optimization is topology optimization. Topology optimization results to find the optimum material distribution and voids. [17]

III. PROBLEM DEFINITION

During discussion with SPM TOOLS, It was observed that they want to optimize the weight of some parts in their band saw machine. But without affecting strength, material and functionality of component. Presently company needs to optimize the weight of roller bracket. In which roller bracket used in stock bar supporting assembly in this assembly total 6 roller bracket are required. In this Roller used to carries weight of stock bar which going to cut by band saw machine and this roller supported to roller bracket. So roller bracket is proposed for project work. Existing roller bracket weight is 3 Kg of each (total 6*3=18 kg) and made of grey cast iron. Here objective is to reduce the weight

Fig. 1: Stock Bar Supporting Assembly

A. *Objective*

1) The objective is to optimize the structural design of the roller bracket and thereby reduce the weight (material use) of roller bracket without reduction in load bearing capacity.
2) Reduce the cost by minimizing the weight of the material thereby increase in the sale by decreasing machine price..
3) Minimize load on resources by using minimum material.

IV. METHODOLOGY

1) Phase 1: Collect data from operational instructional manual and design and development department of SPM TOOLS, Ichalkaranji.
2) Phase 2: Static analysis carried out on current roller bracket to find out max. Displacement and stress and Setting topology optimization parameters such as design variable, responses, constraints, objective and manufacturing constraints and submit for topology optimization and make appropriate change structural change.
3) Phase 3: For design validation it is required to conduct experimental test on optimized roller bracket. For this made photo elastic model of 5mm thickness optimized model. Develop fixture on 3d printer and conduct stress test on Polariscope. If this Photoelastic model of optimized roller bracket shows the stress region & range near about to numerical test (Hyperworks analysis) then this design validate.
4) Phase 4: In last stage, prepare part drawing for regular production. It will be used for next machine

All rights reserved by *www.ijsrd.com*

A. Outline of the Project Work

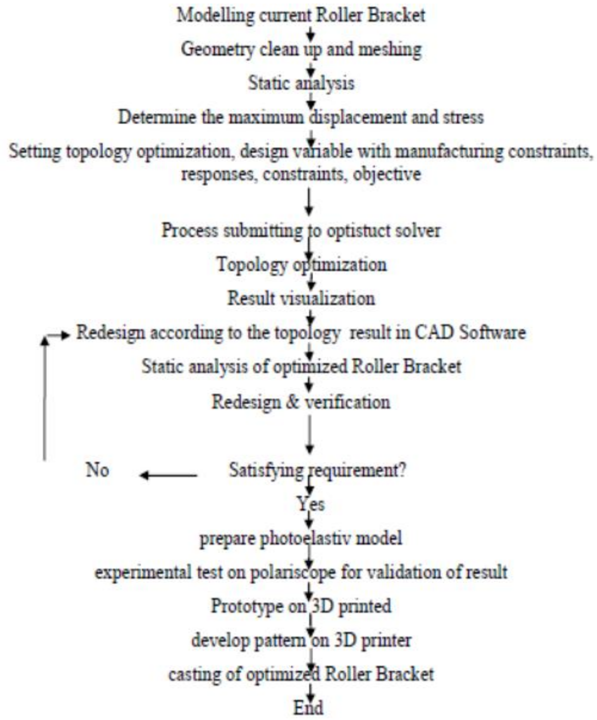

Fig. 2: Flow Chart of Work Carried Out

V. NUMERICAL ANALYSIS IN HYPER WORKS

CAD model of Roller bracket designed in solid works was imported in Hypermesh for geometric cleanup and meshing. For here we used tetra elements.

Sr no.	parameter	Description	value
1	E	Young's Modulus (Mpa)	66000
2	NU	Poisson"s Ratio	0.27
3	RHO	Density (kg/m3)	7200
4	F	Force(N) factor of safety 1.5	2028.9
5	M	Wt. before optimization	3

Table 1 Material Property of Grey Cast Iron, Specification & Loading Conditions

A. Forces Acting on Roller Bracket

Loading condition on roller bracket:

For project we have taken FCAF 250

Band saw machine and their max. Bar holding capacity is **<u>250mm</u>**

Volume of bar/mm = 3.14*radius2 * Length = 3.14*125*125 * 1 mm = 49093.75 mm^3

Weight of bar/ mm = volume * density (steel) = 49093.75* 0.0000078 (kg/mm3)=0.383 kg

<u>Roller 1</u>

Weight on roller 1 = Wight of bar per mm * total length = 0.383 * 720mm = 275.76 kg

Force on roller 1 = weight * 9.81 = 2705.2 N

<u>Roller 2</u>

Weight on roller 2 = Wight of bar per mm * total length = 0.383 * 240mm = 91.92 kg

Force on roller 2 = weight * 9.81 = 901.7 N

<u>Roller 3</u>

Weight on roller 3 = Wight of bar per mm * total length = 0.383 * 480mm = 183.8 kg

Force on roller 3 = weight * 9.81 =1803.5 N

Total no of roller used in machine are 6 in which max load on roller 1 ,

<u>factor of safety 1.5</u>

So total force for design roller bracket is = 2705.2*1.5 = **<u>4057.8 N</u>**

Total force on each bracket is 4057.8/2 = 2028.9N Calculated forces and boundary conditions were applied on meshed model in Hypermesh as shown in figure 3. Static analysis was performed by using optistuct. Viewed Result in Hyperview.

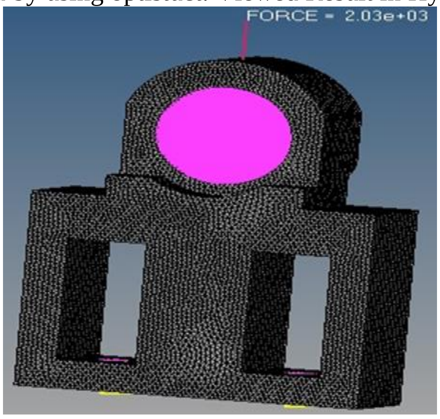

Fig. 3: Roller Bracket with Loading & Boundary Condition

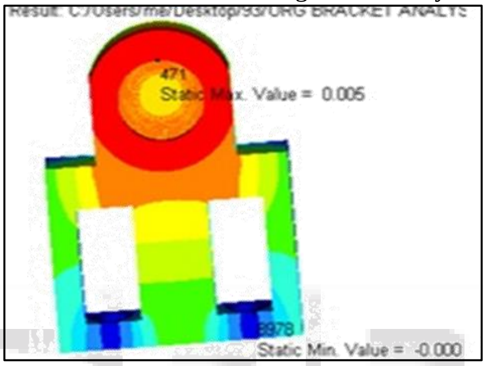

Fig. 4: Max Displacement of Roller Bracket

Static analysis was performed by using optistuct solver. it is observed that the maximum displacement Developed is 0.005mm & Stress developed is 10.2 N/mm2.

Description	value	node
Max disp. lacement (mm)	0.005	471
Von Mises stress (N/mm2)	10.214	52203

Table 2: Result of Existing Roller Bracket

B. Topology Optimization

Topology Optimization technique gives an optimum material distribution within given design space. [7] The design space defined using solid elements. The topology optimization set up in which first is design variable selected as solid, Design responses such as displacement, volume fraction, objective, design constaints- displacement as 0.005 mm upper limit & volume fraction, finally obejective is weighted complance.

Submitted all this optistruct solver to identifies material distribution pattern throughout design space of roller bracket 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, so by removing the material from these design space of component objective of reducing weight of component will be fulfilled with all design constraints.

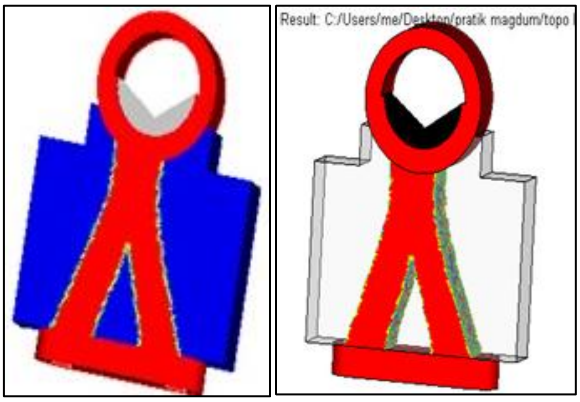

Fig. 5: Topology Result - Element Densities of Roller Bracket

Fig 5 shows the conceptual design which imported in a CAD system using an iso-surface generated with OSSmooth, which is tool available in OptiStruct. This IGES model imported in solid works makes changes as per manufacturing aspect. Figure 6 Shows CAD model of optimized roller bracket.

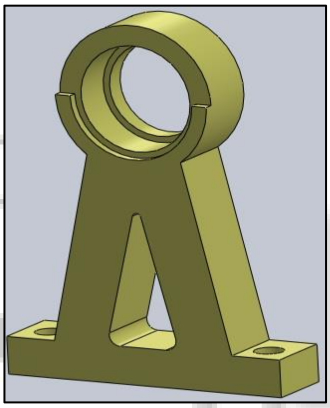

Fig 6: CAD Model of Optimized Roller Bracket

Again conduct analysis on newly optimized Roller bracket 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 Roller bracket. Displacement of optimized Roller bracketis 0.005mm (equal 0.005mm of initial model) & stress 8.12 (N/mm2).

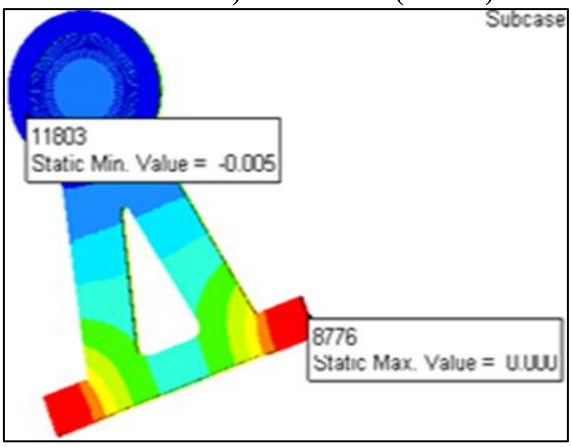

Fig. 7: Displacement of Optimized Roller Bracket

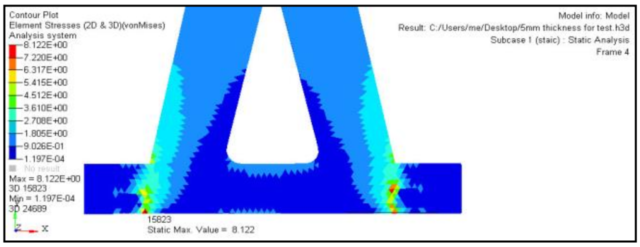

Fig. 8: Stress of Optimized Roller bracket

VI. EXPERIMENTAL VALIDATION

Or experimental validation purpose we conducted test on polariscope here we validated stress region & value. For this we first prepared photoelastic model from epoxy resin material, here we prepared 2D imensional drawing of optimized roller bracket and give to Raste Enterprises, Vishrambag, Sangli - 416415, Maharashtra, India after manufactured of Photoelastic model we prepared fixture for actual loading & boundry conditions for this some fixture part such as bearing block(green color part), loading fixture, pin (white color part) manufactured on 3D printer.

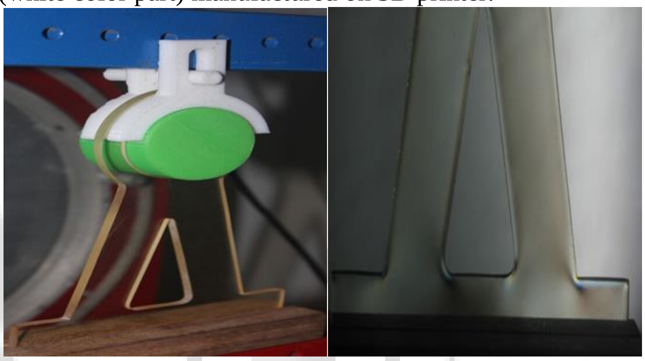

Fig. 8: Experimental Test on Photoelastic Model of Optimized Roller Bracket on Polariscope.

Stress in numeriacal anlysis (hyperworks software)	Stress in experimental (polariscope test)
8.122 N/mm2	7.89 N/mm2

Table 3: Result Comparison

Values of Stress in numeriacal analysis (hyperworks software) & Stress in experimental (polariscope test) are varies within 3 % which acceptable. So we successfully said that our optimized design is validated.

VII. CASTING MANUFACTURING

Here we prepared pattern on 3D printer & manufactured casting in Topgear foundry, Kolhapur. Following some images.

Fig. 9: Pattern & Casting

VIII. RESULT & DISCUSSION

Parameter	before optimised	after optimised
Max.Disp.(mm)	0.005	0.005
Weight(kg)	3	2.2
Total a cost saving/unit	3kg*70=210/-	2.2kg*70=154/- Total saving use of material = 3-2.2 =0.8 kg Total roller bracket used in machine = 6 =6*0.8= 4.8 kg Total a cost saving/unit =4.8*70= 336/-

Table 4: Analyzed Results for Roller Bracket

IX. CONCLUSION

1) As reducing weight and increasing or maintain strength of products are high research demands in the world,
2) Existing roller bracket replaced by a optimized roller bracket & its saves 0.8 kg material use per kg & saves 336/-machine.
3) Reducing cost which beneficial to company to sustain in market with competitor.
4) It can be applied to other components and therefore has a potential to contribute towards environment sustainability better conserving world's metal resources

REFERENCES

[1] Mr.Sagar Awate, Prof.Arun Bhosale, "Bi-directional Evolutionary Structural Optimization of Brake Chamber Mounting Bracket" published in 'International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056 Volume: 03 Issue: 05 | May-2016.

[2] Jikai Liu, Yongsheng Ma, "A survey of manufacturing oriented topology optimization methods" published in Advances in Engineering Software journal, 2016 page no. 161–175.

[3] Sanket C. Vinchurkar, P. M. Khanwalkar, "FEA and Topology Optimization of an Engine Mounting Bracket", published in 'International Journal of Current Engineering and Technology E-ISSN 2277 – 4106, 2016

[4] Jasmin Bhimani, Sanket Pandya, Tejas Bhatt, Ankur Vachhani, "Topology Optimization Of Fixture Used For Quill Mounting", published in 'IJREAT International Journal of Research in Engineering & Advanced Technology, Volume 3, Issue 1, Feb-Mar, 2015

[5] Purushottam Dumbre, Prof A.K.Mishra, V.S.Aher, Swapnil S. Kulkarni, ,"Structural Analysis of steering knuckle for weight reduction", published in ' International Journal of Advanced Engineering Research and Studies' E-ISSN2249–8974 Int. J. Adv. Engg. Res. Studies/III/III/April-June, 2014.

[6] ErsanGönül, MuhsinKılıc, and FatihKarpat, "A Study on Design and Manufacturing for the Side Wall of Large CNC Portal Milling Machine", presented in '2nd International Conference on Research in Science, Engineering and Technology (ICRSET'2014)', March 21-22, Dubai (UAE) 2014.

[7] R. Rezaie, M. Badrossamay*, A. Ghaie, H. Moosavi, "Topology optimization for fused deposition modeling process" presented in 'The Seventeenth CIRP Conference on Electro Physical and Chemical Machining (ISEM)' Procedia CIRP 6 2013 page no. 521 – 526.

[8] Vivek Zolekar, Dr. L.N. Wankhade, "Finite Element Analysis and Optimization of I.C. Engine Piston Using RADIOSS and OptiStruct", published in „altair technology conference india. 2013

[9] He Bin, "Multi-Constraint Topology Optimization of Forging Machine Bed for Light Weight Design", published in „International Journal of Advancements in Computing Technology (IJACT)", Volume5, No.9, May 2013.

[10] H.B.Ramani, Neeraj Kumar, "Using Shape Optimization Tool in Ansys Software for Weight Reducation of Steel Connecting Rod", Engineering Research & Technology (IJERT)", Vol. 2 Issue 2, February- 2013, ISSN: 2278-0181

[11] Malleswara Swami and K.SunilRatna Kumar,"Design and structural analysis of CNC vertical milling machines", Published in „International Journal of Advanced Engineering Technology", IJAET/Vol.III/ Issue IV/Dec. 2012/97-100.

[12] Rinkle Garg and Sunil Baghla, "Finite Element Analysis and Optimization of Crankshaft Design", Published in „International Journal of Engineering and Management Research", Vol.-2, Issue-6, December 2012, ISSN No.: 2250-0758, Pages: 26-31.

[13] D. M. Chauhan, Prof. S. B. Soni and Prof. A. M. Gohil, "Parametric Optimization of Hydraulic Modular Trailer Frame using ANSYS (APDL)", Published in, Institute of technology", Nirma University, AHMADABAD – 382 481, December 2011.

[14] Muni Prabaharan and V.Amarnath, "Structural Optimization of 5Ton Hydraulic Press And Scrap Baling Press for Cost Reduction by Topology", Published in „International Journal of Modeling and Optimization", Vol. 1, August 2011.

[15] G.I.N. Rozvany, "Aims, scope, methods, history and unified terminology of computer-aided topology optimization in structural mechanics", published as reviewed paper in „Structural Multidisc Optimization", 90–108, Springer-Verlag 2011.

[16] Peter W. Christensen and Anders Klarbring, a book on "An Introduction to Structural Optimization", by Springer Science in 2009.

[17] Bendsøe MP, Sigmund a book on 'Topology optimization: theory, methods and applications'. Berlin: Springer-Verlag; 2003.

[18] SPM tools- Operating instruction manuals.

[19] PDF of Sawing Machine, uv.cheme.cmu.edu/procedures/machining/ch6.pdf

[20] Hyper works student guide, practical aspects of finite element analysis.

[21] Altair hyperworks.11 user Manual.

All rights reserved by *www.ijsrd.com*