International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 – 5161

Available at http://inpressco.com/category/ijcet

Research Article

Structural Analysis of a Guide Bracket for Weight Reduction using Optistruct

Anish Pramod Borkar* and Dr.V.R.Naik

Mechanical Engineering Department, DKTE, Ichalkaranji, Kolhapur, India

Received 12 Feb 2018, Accepted 15 April 2018, Available online 19 April 2018, **Vol.8, No.2 (March/April 2018)**

Abstract

CAE driven design processes boosting all modern manufacturing units by providing cost effective and optimized design. The product design is stimulated by structural optimization tools like geometrical (topology) optimization, shape optimization and size optimization. Geometrical optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets. Using geometrical optimization, engineers can find the best concept design that meets the design requirements. Geometrical optimization is used at the concept level of the design process to arrive at a conceptual design proposal that is then fine tuned for performance and manufacturability. The main focus of the machine tool manufacturers has been shifted to conservation of natural resources and optimum energy utilization through weight reduction. Weight reduction can be achieved by the introduction of better material, better manufacturing processes and design optimization. In automobile and aircraft sector, topological optimization has become an integral tool of product design and development for the optimum weight evaluation. Enhanced structural performance and stability is achievable by geometry (topology) assisted design model which gives optimized results which are better and innovative in terms of product design. In other words, simulation saves time, reduces costs, and strengthens the competitiveness of companies, thus strengthened market position through enhanced structural design. Also Rapid Prototyping has been used over traditional manufacturing and forming for the pattern making process which helps reducing material wastage and time associated with tooling (for Forming) and it allows a complex shape to be produced directly and automatically from its 3D CAD model usually within hours. This paper deals with the structural optimization of guide bracket for horizontal band sawing machine by using topology based optimization technique using Optistruct. It is observed that weight of the guide bracket has been optimized up to 26% without changing its load bearing capacity under current operating load conditions. Also it saved cost of material up to 26% for the optimized model.

Keywords: *Hyperworks, Optistruct, Structural Analysis, Geometrical Optimization, Rapid Prototyping*

1. Introduction

CAE driven design processes boosting all modern manufacturing units by providing cost effective and optimized design. The product design is stimulated by structural optimization tools like geometrical (topology) optimization, shape optimization and size optimization. Geometrical optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets. Using geometrical optimization, engineers can find the best concept design that meets the design requirements. (Waqas Saleem *et al*,2008).

*Corresponding author's ORCID ID: 0000-0002-4328-4424
DOI: https://doi.org/10.14741/ijcet/v.8.2.29

Also Rapid Prototyping has been used over traditional manufacturing and forming for the pattern making process which helps reducing material wastage and time associated with tooling (for Forming). Rapid Prototyping or Additive Manufacturing 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.

To solve problem it is discredited by using the finite element method (FEM) and dividing the design domain into discrete elements (mesh). The resulting problem is then solved using optimization methods to find which elements are dense materials and which are not. This optimization delivers a design concept which answers questions such as where to remove material, where to place (stiffening) ribs, etc. After the optimization results have been interpreted and smoothed (in the CAD system) an analysis is carried out to verify the optimized components performance. Based on these results, another optimization step may follow, addressing questions such as how to change local geometry in order to reduce stress peaks or how thick do the stiffening ribs really need to be Geometrical optimization. (Sandro L. Vatanabe*et al,*2016)

The most general form of structural optimization is Geometrical optimization. As with shape and size optimization, the purpose is to find the optimum distribution of material. In Geometrical optimization the resulting shape or geometry is not known. The purpose is to find the optimum distribution of material and voids. For this we used Altair Hyperworks and Optistruct is used as solver. Final Results are then evaluated in a suitable CAD model.

The final model should not exceed the maximum displacement and stress induced than initial model to fulfill the objective of the work. This paper deals with the preparation of CAD model, calculation of forces acting on Guide Bracket, analysis and optimization of the same, appropriate changes in the geometry considering manufacturing constraints, reanalysis of new model. (Vinayak Kulkarni*et al,*2014)

2. Problem definition

Guide bracket of Horizontal Band Sawing machine is proposed for project work. Guide Bracket is a part of feeding system. It supports the bar being cut by a cutting saw and keeps it in a perpendicular in direction while cutting. There are 2 Guide Brackets used for this purpose. Existing Guide bracket weight is 2.7 Kg of each (total 2*2.7=5.4 kg) and made up of IS Standard Grey Cast Iron (FG220).Here objective is to reduce the weight of Guide bracket by minimum use of material.

3. Objective

1) To optimize the structural design of the Guide bracket and thereby reduce the weight (material use) of Guide bracket without reduction in the load bearing capacity.
2) To reduce the cost of the material associated with the Guide Bracket.
3) To reduce load on the natural resources.

4. Experimentation

CAD model of Guide Bracket designed in Solidworks was imported in Hyperworks. Then geometrical cleanup and meshing has been done. Meshed model of Guide Bracket consist of 33664 nodes and 151185 elements. There are 3 rigid body elements. All elements are 3D Tetramesh (volume mesh).Tetra elements give enhanced result as compared to other types of elements, therefore the elements used in this analysis is tetra elements.

IS Grade Gray Cast Iron has been used for Guide Bracket. Material is created for Guide Bracket. Calculate forces acting on Guide Bracket. Apply boundary conditions to meshed model in Hypermesh as shown in Fig. 3. Static analysis was performed by using Optistuct. Results are viewed in Hyperview. All specifications such as material property and results are shown in Table 1.

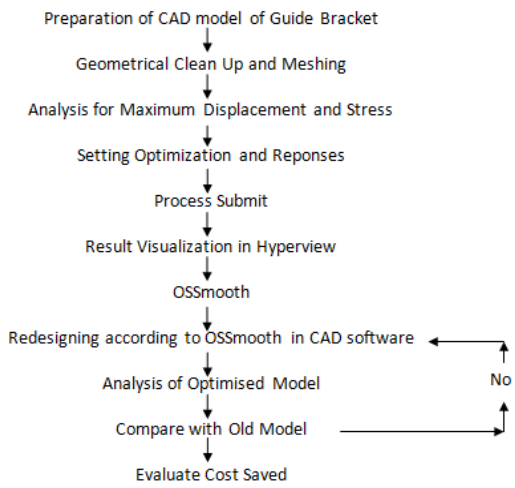

Fig. 1: Way of Work(PurushottamDumbre*et al,*2014;Jikai Liu *et al,*2016)

Above Fig.1 shows the way of work. In order to precede with this study the various forces acting on Guide Bracket has been calculated first.

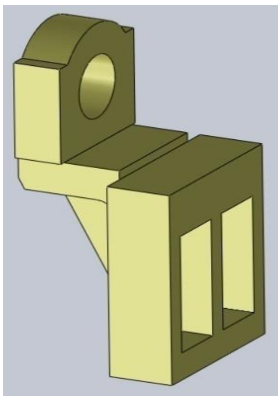

Fig. 2: CAD Model of Existing Component

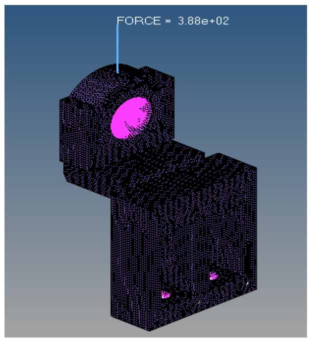

Fig. 3: Meshing and Load application

5. Design parameters

1. Weight due to Parts mounted on Guide Bracket :
Weight of roller = 3.5 kg =3.5*9.81=34.33 N
Weight of left clamper assembly = 2.6 + 2 kg =4.6 kg = 45.12N
Weight of Right clamper assembly = 2.6 + 2 kg =4.6 kg = 45.12N
2. Total Force on Guide Bracket :
Maximum 20 kg compression force on each clamper (left & right) =20+20 = 40 kg = 40*9.81 = 392.4N
Total load on both guide bracket= 34.33+45.12+45.12+392.4 = 516.97 / 2 = 258.5N
Factor of Safety =1.5
So total force for single roller guide bracket is = 258.5*1.5 = 387.75 N

Table 1 Properties of Material

Parameters	Quantity
Elastic Modulus (E)	6.6178e+010N/m²
Poisson's Ration (Nu)	0.27
Density(RHO)	7200 Kg/m³
Force on Guide Bracket	387.75 N
Weight before optimization	2.7 Kg.

6. Analysis before optimization and Optimisation

1. Static Analysis

Static analysis is carried out by using Optistruct Solver. Inputs were given in terms of single point constraints, load cell and rigid body elements along with magnitudes. It shows that the maximum static displacement is at node 65 of 0.016 mm under compression. Maximum Stress induced is 9.506 N/mm2.

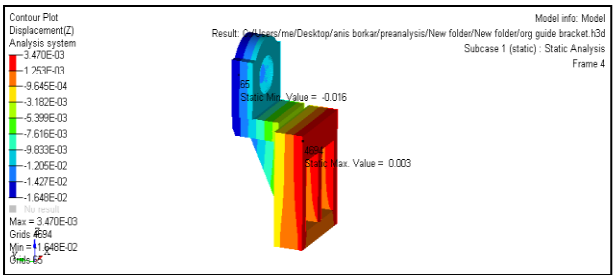

Fig. 4: Initial analysis in Hyperworks

Table 2. Results after initial analysis

Parameter	Quantity
Maximum Displacement	0.016 mm at Node 65
Maximum Stress Induced	9.506 N/mm²

2. Geometrical optimization by using Optistruct

Geometrical Optimization is defined as finding out the best possible material distribution in selected design space with considering the given sets of objective and design constraints and responses. Reponses like volume fraction, weighted component and displacement were established. Also assigning design and non design area is necessary. Material is removed from design area. Design objective was Minimum weight compliance. Optimization algorithm works to find the optimal solution(s) to a given computational problem that maximizes or minimizes a particular function. Total 25 iterations took place for optimization. After running optimization results are viewed in Hyperview.

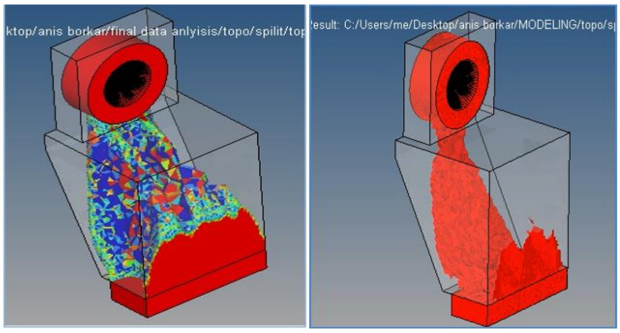

Fig. 5: Element with high densities and high stress.

7. Redesigning Optimized model in CAD software and Re-analysis in Optistruct

Using CAD software Solidworks redraw the component as per inputs from OSSmooth. Unnecessary material is removed and component is redeveloped with the help of design engineer in company. Again its static analysis has been done in Hyperworks. It is necessary that new designed component should stay within its initial limits. Again setup all Meshing conditions, apply initial boundary conditions and evaluate the result. Cross check that displacement and stress of optimized model do not exceed values of initial model. Fig. 9 shows displacement result of optimized Guide Bracket. Displacement of optimized Guide Bracket is 0.013mm (<0.016mm of initial model).

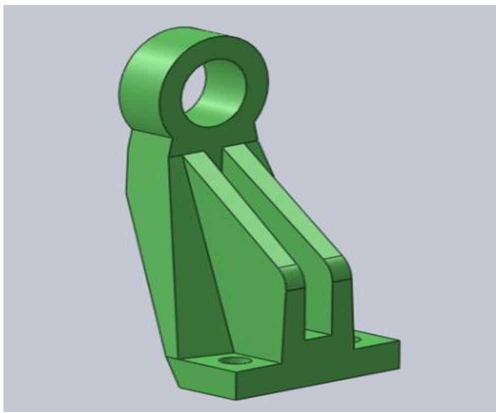

Fig. 6: Redesigned CAD model

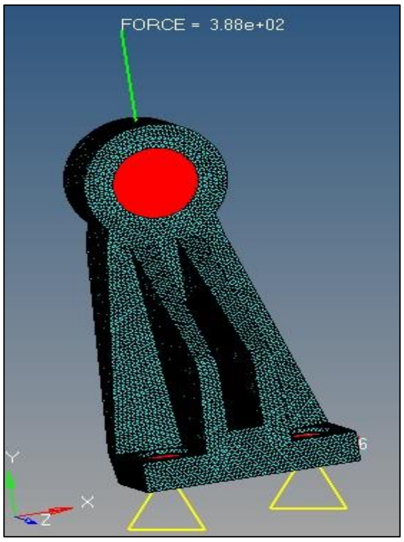

Fig. 7: Meshing and Load Application on new model in Hyperworks

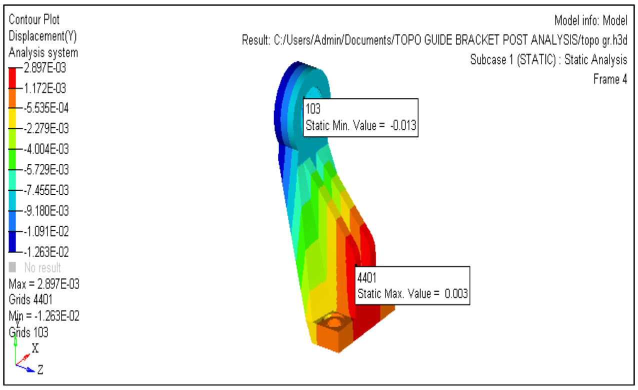

Fig. 8 : Analysis of optimised Model

Table 3 Results after final analysis

Parameters	Before Optimization	After Optimization
Max. Displacement (mm)	0.016	0.013
Max. Stress (N/mm^2)	9.506	11.618

8. Development of pattern/prototype on 3D Printer

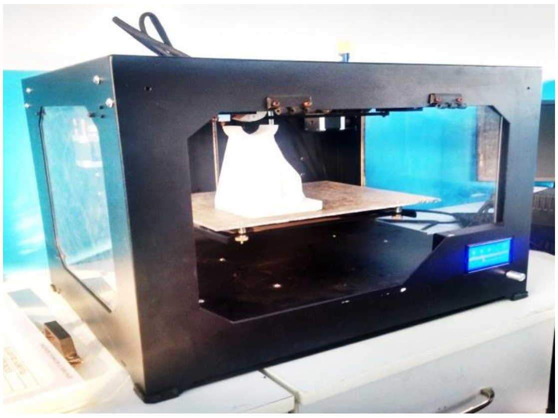

Fig. 9: Rapid Prototyping

Fig. 10: Pattern made by using 3D printer

A prototype has been made on 3D Printer to make pattern for casting. Then the prototype has been tested under double hydraulic pressure to validate the design.

9. Result and Validation

Validation of the optimised Guide Bracket is done on Universal Testing Machine under IS 1828:2005(ISO 7500-1:2004).

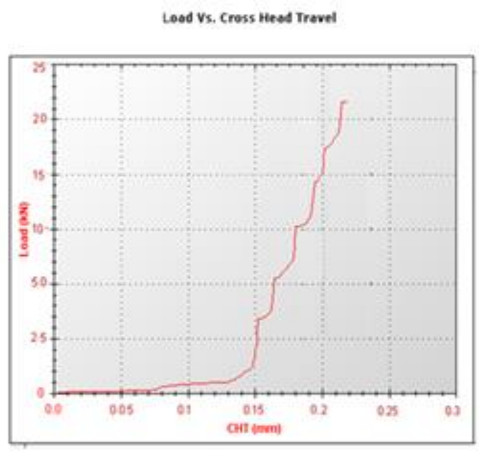

Fig.10 : Graph generated after UTM testing

From Graph it is observed that optimised Guide Bracket sustain present operating load conditions.

Table 4: Final Result

Parameters	**Before Optimization**	**After Optimization**
Max. Displacement (mm)	0.016	0.013
Max. Stress (N/mm^2)	9.506	11.618
Weight (kg)	2.7	2
Reduction in weight	2.7-2=0.7 kg (26%)	
Raw material cost saved per guide bracket	0.7 * 70 = Rs.49/-	
Raw material cost saved for a single machine	49*2= Rs.98/-	
Total cost saved per year (Say 50 Machines)	98*50= Rs.4900/-	

Conclusion

1) The values obtained for the maximum Von Mises stress and maximum displacement are lower than safe limit. Initial Von Mises Stress was 9.506 KN/mm^2 and the Final Von Mises Stress is 11.618 KN/mm^2. Initial Displacement was 0.016 mm and Final Displacement is 0.013 mm.
2) From the graph generated by UTM testing, it is observed that the optimum Guide Bracket withstands the actual working loading conditions (388N). From the graph it is clear that the optimized guide bracket qualify the actual working conditions of current guide bracket after doing performance validation as per the company requirement and standard.
3) The Geometrical optimization generates an optimized material distribution (for weight optimization) against a set of loads and constraints within a given design space. Initial weight was 2.7 kg and final weight is 2 kg (26% Reduction). Total cost saved per guide bracket is Rs.49/-

Future Scope

Every component of Band Saw Machine can be optimized for the purpose of weight reduction. This is will reduce total manufacturing cost for every machine.

References

WaqasSaleem, Fan Yuqing, Wang Yunqiao,(2018)Application of Topology Optimization and Manufacturing Simulations – A new trend in design of Aircraft components, Proceedings of *the International Multi-Conference of Engineers and Computer Scientists*, Volume II, pp 19-21

Sandro L. Vatanabe, Tiago N. Lippi, CíceroR.de Lima, GlaucioH. Paulino, Emílio C.N. Silva(2016) Topology optimization with manufacturing constraints: A unified projection-based approach published in *Advances in Engineering Software journal*,pp. 97–112

VinayakKulkarni,AnilJadhav,P. Basker (2014), Finite Element Analysis and Topology Otimization of Lower Arm of Double Wishbone Suspension using RADIOSS and Optistruct, published in '*International Journal of Science and Research* (IJSR), Volume 3 Issue 5

PurushottamDumbre, Prof A.K.Mishra, V.S.Aher, Swapnil S. Kulkarni (2014),Structural Analysis of steering knuckle for weight reduction, published in '*International Journal of Advanced Engineering Research and Studies*' E-ISSN2249–8974Int. J. Adv. Engg. Res. Studies/III/IIIpp 245-265

Jikai Liu, Yongsheng Ma,(2016) A survey of manufacturing oriented topology optimization methods published in *Advances in Engineering Software journal*, pp. 161–175.

MahendraShelar(2016), Optimization of Steering Knuckle using DOE, published in '*International Journal on Mechanical Engineering and Robotics* (IJMER)' ISSN (Print) : 2321-5747, Volume-4, Issue-1,

National Accreditation Board For Testing And Calibration Laboratories(2014) specific criteria for calibration laboratories in mechanical discipline: UTM, Tension Creep and Torsion testing machine issue no. : 05 issue date:

Chuan Jiang, Hongyi Hu, ZhizhouXu, Ruzhen Liu, (2015)The U-type Platform Multi-objective Topology Optimization Research presented paper in ' 4th*International Conference on Computer, Mechatronics, Control and Electronic Engineering (ICCMCEE 2015)*

MohdNizamSudin, MusthafahMohdTahir, FaizRedzaRamli, ShamsulAnuarShamsuddin.(2014)Topology Optimization in Automotive Brake Pedal Redesign, published in *International Journal of Engineering and Technology (IJET)* ISSN : 0975-4024 Vol 6 No 1 Feb-Mar pp 398

R. Rezaie, M. Badrossamay*, A. Ghaie, H. Moosavi,(2013) Topology optimization for fused deposition modeling process presented in '*The Seventeenth CIRP Conference on Electro Physical and Chemical Machining (ISEM)*'Procedia CIRP 6 pp 521 – 526.

Tushar M. Patel, Dr. M. G. Bhatt and Harshad K. Patel(2013), Parametric Optimization of Eicher 11.10 Chassis Frame for Weight Reduction Using FEA-DOE Hybrid Modeling, published in „*International Journal of IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*", e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 6, Issue 2,PP 92-100

K. Mallieswaran, SamadArbaazAnjum and A. Pradeep(2012) Case Study on manufacturing of Light Weight Component by Metal Forming, published in ‚*Journal of Advanced Technology in engineering*", Vol. 1,pp: 63-70.

D. M. Chauhan, Prof. S. B. Soni and Prof. A. M. Gohil(2011)Parametric Optimization of Hydraulic Modular Trailer Frame using ANSYS (APDL), Published in, *Institute of technology*", Nirma University, Ahmadabad – pp 382-481

S. Sánchez-Caballero, M.A. Selles, R. Pla-Ferrando, E. Pérez-Bernabeu(2011)Weight reduction in structures using finite elements and multi objective genetic algorithms, Published in ANNALS of the Oradea University. *Fascicle of Management and Technological Engineering*, Volume X (XX), PDF of Sawing Machine, uv.cheme.cmu.edu/ procedures/machining/ch6.pdf Pp: 1-18

Emmanuil F. Kushnir; Mahendra R. Patel(2001) Terrence M. Sheehan,Material Considerations in Optimization of Machine Tool Structure Published in Proceedings of *2001 ASME International Mechanical Engineering Congress and Exposition* pp.11–16,

ErsanGönül, MuhsinKılıc, and FatihKarpat,(2014)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)*', pp 21-22

SPM tools- Operating instruction manual of Model HBM 250.Pp2-3

Horizontal static forces exerted by men standing in common working positions on surfaces of various t r actions-including coefficients of friction between various floor and shoe materials(1971) by Danny E. Robbinson K. E.

Kroemer,*Aerospace medical research laboratory wright-patterson* air force base, Ohio pp.26-28

Altair Hyperworks.11 user Manual

Hyper works student – Guide, practical aspects of finite element analysis. Pp: 1-112

John Wiley S.S.Rao(2009)Engineering Optimization Theory and Practice, New Jersey, Fourth Edition, p.p693-703