- Solid modeling of the fix jaw of rear vice.
- Determination of displacement and max. Disp. Node numbers of current fix jaw.
- Determination of Von mises stress of current fix jaw.
- Topology optimization setup with minimum member Size (8 mm) and single type draw direction..

4. Methodology

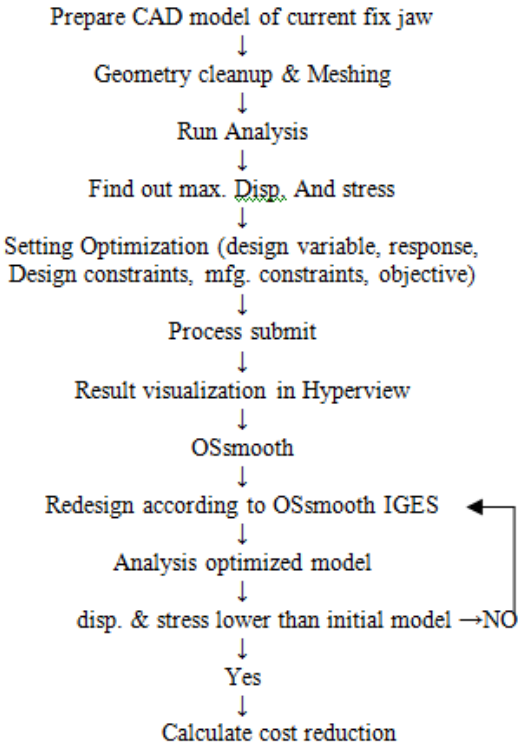

Figure 1: Work flow diagram

In above diagram show that way of work. In order to proceed with this study various forces acting on fix Jaw. CAD model of fix jaw designed in solid works was imported in Hypermesh. Then geometric cleanup and meshing has done. Meshed model of fix jaw consist of 44053 nodes and 202579 elements. All are 3D Tetramesh (volumemesh).Tetra elements give enhanced result as compared to other types of elements, therefore the elements used in this analysis is tetra elements.

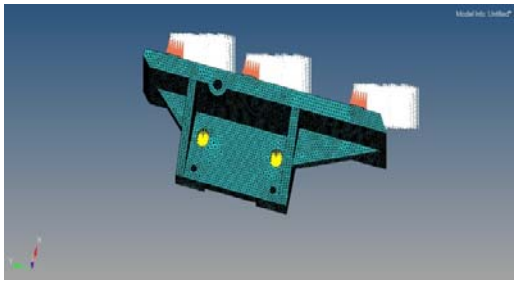

Figure 2: Fix jaw with meshing, loading & boundry Condition.

Ductile iron material has used for fix jaw. Calculated forces and boundary conditions were applied on meshed model in Hypermesh as shown in figure 2. Static analysis was performed by using optistuct. Viewed Result in Hyperview. All specification such as material property and result shown in table1.

4.1 Design parameters

1) Forces acting on jaws

Rear vice jaw has two jaws one is fixed another is movable. Hydraulic cylinder attached to movable jaw which applies force to jaw to clamp the work piece. Inner diameter of hydraulic cylinder is 80.125mm, while maximum pressure of hydraulic oil on cylinder is 22kg/cm2 .here we calculate total force applying on fix jaw [6].

P=F/A
F=P*A
F=22*(3.142*(4.00625)2) =1109.44 kg.
F=1109.44*9.81 = 10884 N
Maximum load applying on fix jaws is 10884N.
Considering factor of safety (F.S.) 1.5 then
F= 10884*1.5 =16326N.

Table 1. Material properties and specification of jaw

Parameter	*Description*	*Fix jaw*
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 jaw before optimization	12.1 kg

5. Results and Discussions

a) Static analysis

Static analysis was performed by using optistuct solver. From figure3, it is observed that the maximum displacement Developed is 0.116mm for fix jaw. Stress developed is 97.82 N/mm^2. Which is lower than the Yield strength. Hence, design is safe that is the values of maximum stresses are acceptable as compared to yield strength so design constraints For Optimized fix jaw is to maintain displacement value lower than 0.116 mm.

Table 2: Analyzed results for fix materials

Description	*Fix jaw*
Max displacement in mm	0.116 at node 7888
Von Mises stress (N/mm^2)	97.82

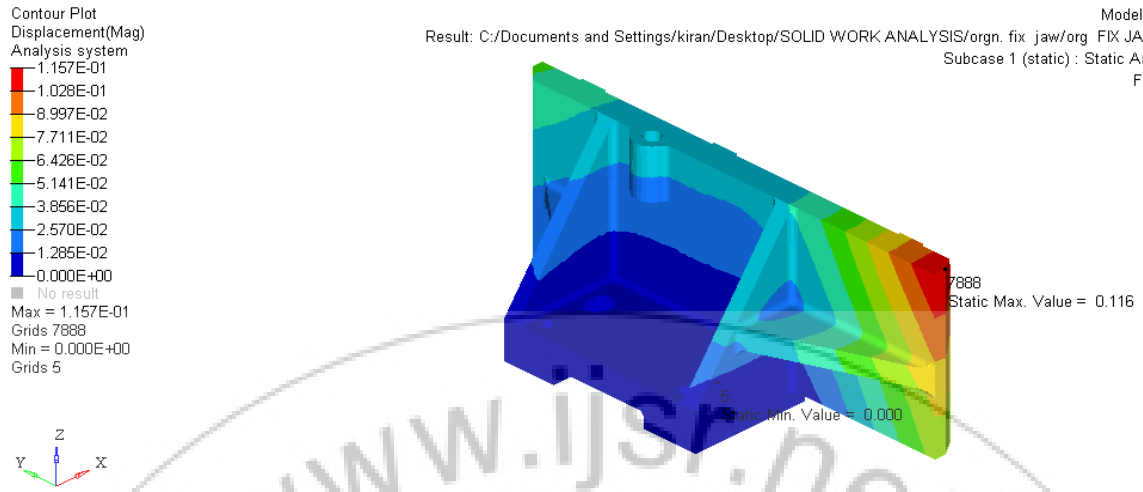

Figure 3: Max. Displacement of fix jaw

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 and setting up two manufacturing constraints 1. Minimum member size control (8 mm) 2. Draw direction type (single).Design response was volume Fraction, weighted comp., displacement. Optimization design constraints were volfrac for that upper bound 0.30 and Displacement constraint was upper bound 0.116mm for fix jaw.

Finally design objective was Minimum weight compliance. Run optimization by using optistuct Solver. Finally viewed result in hyper view.

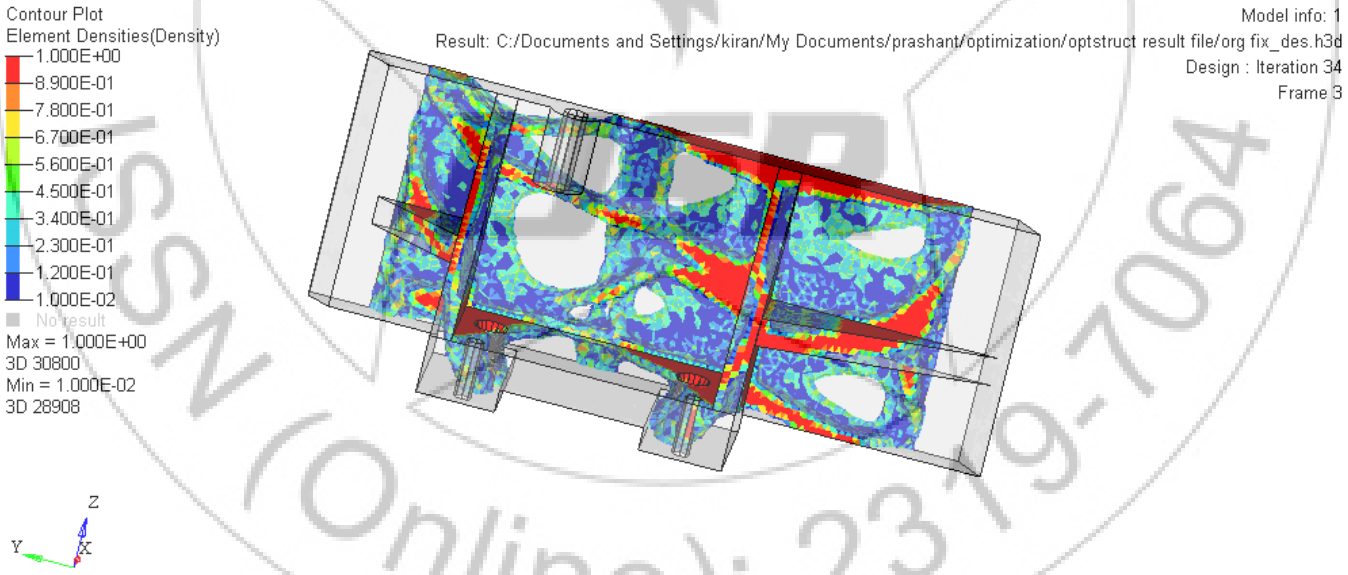

Figure 4: Hyper view element densities of fix jaw

Figure 4 shows element density for fix jaw. Optistruct identifies material distribution pattern throughout jaw 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 numbers of iterations, when solution converges the density pattern of component a region with lower density indicate that it can be removed without hampering safety of component. So by removing the material from these design space of component objective of reducing weight of component will be fulfilled with all design constraints. A conceptual design can be imported in a CAD system using An iso-surface generated with OSSmooth, which is part of OptiStruct. This IGES model imported in solid works makes changes as per manufacturing aspect. Figure 5 Shows CAD model of fix jaws resp.

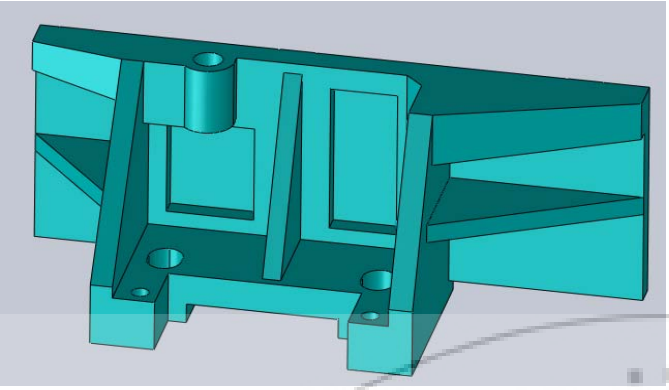

Figure 5: CAD model of optimized fix jaw

Again conduct analysis on newly optimized fix jaw model. Setup all meshing, boundary and loading condition. Cross Check that displacement and stress of optimized model do not exceed value Initial model. Figure 6 Shows displacement result of optimized fix jaw. Displacement of optimized fix jaw is 0.106mm (<0.116mm of current model). Then Check the magnitude of displacement and stress of optimized model has lower than initial model.

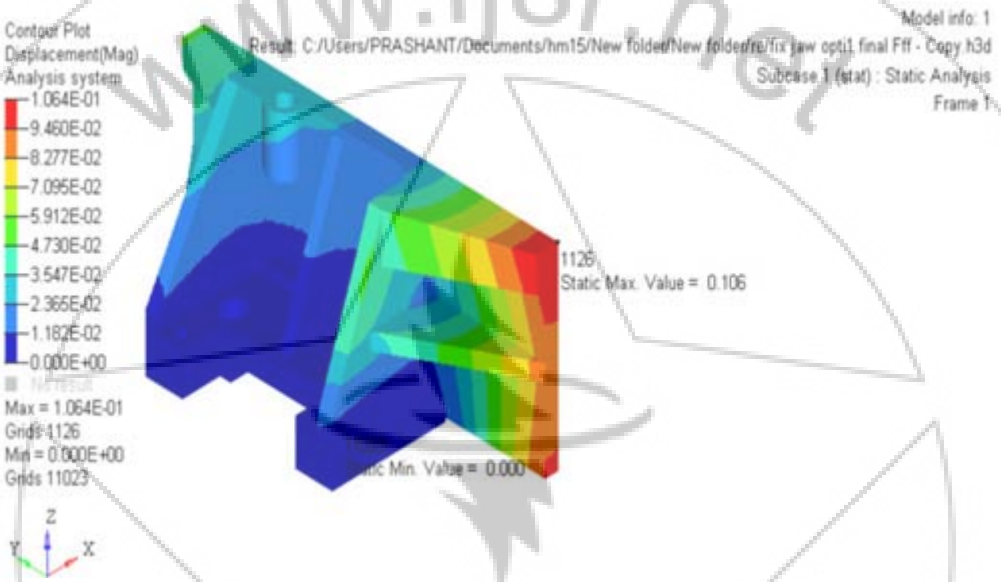

Figure 6: Disp. of optimized fix jaw 0.106mm

Table 3: Analyzed results for fix materials

parameter	*fix jaw **before** optimized*	*fix jaw **after** optimized*
Max. Displacement(mm)	0.116	0.106
Max. Stress (N/mm^2)	97.82	93.89
Weight optimization (kg)	12.1	9.23

Table 4: Total weight and cost saving

Design parameter	*fix jaw after optimized*
Weight reduced (kg)	(12.1-9.23)=2.87kg (24%)
Total cost saving/unit	2.87*85=244/-
Total cost saved/year	**244 *100 = 24,400/-**

6. Conclusions

In this work Forces acting on fix jaw of rear vice of horizontal band saw machine has been calculated with considering factor of safety. CAD model of jaws has been carried out using Solid works software. The static analysis as well as optimization of fix jaws has been carried out in Hyperworks. From the analyzed results, it is concluded that.

- The values obtained for the maximum displacement and von mises stress of optimized model are lower than existing model.
- Topology optimization generates an optimized material distribution for a set of loads and constraints within a given design space. This Optimization reducing weight, manufacturing cost of component fulfilled with all design constraints.
- Weight optimization of rear vice resulted to 24% of weight reduction than existing model. So that company saves 24,400/- per year.
- Finally we reduced cost of fix jaw by 244/- per piece.
- Though this work we can minimize load on resources.

7. Future Scope

We can apply this weight optimization method to all casting parts of horizontal band saw machine which are over design. So we can minimize total Manufacturing cost of machine and it will better to Increase sale in market

References

[1] Vinayak Kulkarni1, Anil Jadhav2, P. Basker3, "Finite Element Analysis and Topology Optimization of Lower Arm of Double Wishbone Suspension using RADIOSS and Optistruct," International Journal of Science and Research (IJSR), Volume 3 Issue 5, May 2014.
[2] 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, No. 3, August 2011

[3] Vivek Zolekar, Dr. L.N. Wankhade, "Finite Element Analysis and Optimization of I.C. Engine Piston Using RADIOSS and OptiStruct",altair technology conference 2013 india.
[4] Rinkle Garg, 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.
[5] Malleswara Swami1, K.Sunil Ratna Kumar2,"Design and structural analysis of CNC vertical milling maches", International Journal of Advanced Engineering Technology, IJAET/Vol.III/ Issue IV/Oct.-Dec. 2012/97-100
[6] SPM tools band saw product manual.
[7] Altair hyperworks.11 user Manual.
[8] Pdf on SAWING MACHINES GENERAL, uhv.cheme.cmu.edu/procedures/machining/ch6.pdf
[9] Tushar M. Patel, Dr. M. G. Bhatt, Harshad K. Patel. „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 (Mar. - Apr. 2013), PP 92-100.
[10] Waqas Saleem, Fan Yuqing, Wang Yunqiao, "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 2008 Vol II IMECS 2008, 19-21 March, 2008, Hong Kong.

Author Profile

Mr. N. V. Hargude has done M.E. (Mechanical-Design Engg.), DBM, MIE. Ph.D. (Pursuing). He is working as Associate Professor & Dean R & D in Padmabhooshan vasantraodada Patil Institute of Technology, Budhgaon, 416034. India

Prashant H. Patil received degrees in Mechanical Engineering from Shivaji University, Kolhapur, India