SHAPE OPTIMIZATION OF FRONT AXLE BRACKET BY USING FINITE ELEMENT ANALYSIS

[1]PANKAJKUMAR.J.M.
Department of Mechanical Engineering, Rungta College of Engineering and Technology, Raipur
Chhattisgarh, India
pankajsir@gmail.com

[2] NARENDRA YADAV
Department of Mechanical Engineering, Rungta Engineering College, Raipur, Chhattisgarh, India

[3]ANKUSH TIWARI,
Department of Mechanical Engineering, Rungta Engineering College, Raipur, Chhattisgarh, India

ABSTRACT: The objective of this paper is to analyse the proposed optimum design of tractor front axle bracket for varying load conditions. Due to brittleness of the cast component material, it is subjected to sudden failure under dynamic load condition causing potential danger. The front axle bracket is designed for high strength and stiffness. The existing design has no field failure reports and hence its parameters were taken as basis for comparison with results of the proposed designs. Based on the finite element analysis results, optimum design was carried out for the bracket for weight optimization and easy manufacturability. The proposed design was evaluated for selected worst load cases of the existing design. The finite element analysis of proposed models yielded displacement and stresses close to the existing design. The increase in displacement was not significant and the proposed design met the structural requirement. It was also observed that the proposed design has significant reduction in weight and did not involve welding, thereby significantly saving manufacturing cost. The assembly components like smaller diameters bearing, smaller knuckle size, etc were also found to be cost effective. The present work showcases the use of finite element analysis as a method for reduction of cost in terms of materials & manufacturing.

Keywords: Front Axle Bracket, Finite Element Analysis, Maximum Von Mises Stress, Weight optimization

1. INTRODUCTION

Tractor users demand low cost and light weight components for fuel efficiency and cost effectiveness. To meet the performance targets, manufacturers go for effective use of materials, easy manufacturability, weight efficient components and reduced design cycle time. Tractors work on difficult conditions than other machines and its components should have high safety of factor [1] .Front Axle of Tractor needs very good design as this part experiences the worst load condition of the whole tractor [2].The front axle bracket requires a properly designed support with high strength and stiffness. Due to brittleness of the Nodular cast iron component material, it is subjected to sudden failure under dynamic load condition causing potential danger.

The von Mises stress also known as the maximum distortion energy criterion, is often used in determining whether an isotropic and ductile metal will yield when subjected to a complex loading condition. Von Mises stress is compared with the material's yield stress, constituting the von Mises Yield Criterion [3]. Analysis is carried out on the front axle bracket to assess the stress and displacement in the system. The shape optimization is done on the component for optimum material distribution. Solid works software [4] is used as methodology for optimum designing of front axle bracket.

2. FINITE ELEMENT ANALYSIS

Finite element analysis is a computational tool for performing engineering investigation. It uses mesh generation technique for dividing a complex problem into smaller elements and a coded finite element algorithm software program [5]. It is useful for components with complex loadings, material properties, and geometries where analytical solutions cannot be obtained [6]. The bracket optimizing is very complex due to its shape and is difficult to design through mathematical calculations[7]. ANSYS 15 [8] software is used for finite element method for static analysis is used in the present study. By analyzing the weakest link and the difference in strength between each node for the overall structure in the condition, it can provide a theoretical basis and direction for the design and optimization of geometric entities [9].

3. METHODOLOGY

The existing front axle support bracket has been taken from Mahindra tractor (Model-Arjun 265 DI) [10] /for the analysis. Existing weight is 31.28 kg and its material is nodular cast iron. The total load acting downward on the bracket is 15000 N (considering the engine weight with mountings). Structural analysis is carried out for static condition. The main objective of this study is minimization of volume with manufacturing feasibility for given strength and life with less design cycle time.

3.1 CAD Model Generation

Solid modeling of component has done with the help of Solid works software. The software is a Para solid-based solid modeler which utilizes a parametric feature-based approach to create assemblies and models. The identified parameters refer to the constraints whose values decide the geometry or shape of the model or assembly. Fig. 1 shows the CAD Model with hole position and boss feature of Front Axle Bracket.

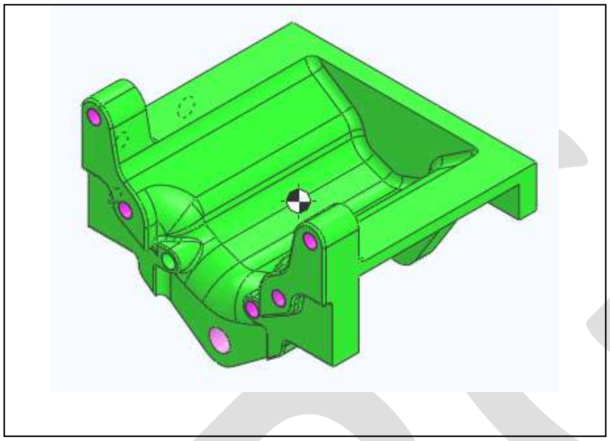

Fig. 1 CAD Model with hole position and boss feature of Front Axle Bracket

3.2 Analysis of Existing Front Axle Bracket (FAB)

The analysis of existing front axle bracket is divided into three areas; preprocessing, solution and post processing.

3.2.1 Preprocessing

The analysis front support bracket (FAB) of the tractor is done in ANSYS 15 software in the static structure analysis work bench. Static analysis is used to determine the stresses, strain, displacements, and forces in structures or components caused by loads that do not induce significant damping and inertia effects. Steady load and response conditions are assumed (vary slowly with respect to time). Table 1 shows the size of Nodes and Elements of FAB. The location of forces applied on FAB and meshing of FAB is shown in Fig. 2 and 3.

Table 1 Size of Nodes and Elements of FAB

	Mesh		Element type	
Sr.No.	Entity	Size		
1	Nodes	266840	Connectivity	Statistics
2	Elements	147493	TE10(Tetrahedron element)	47493

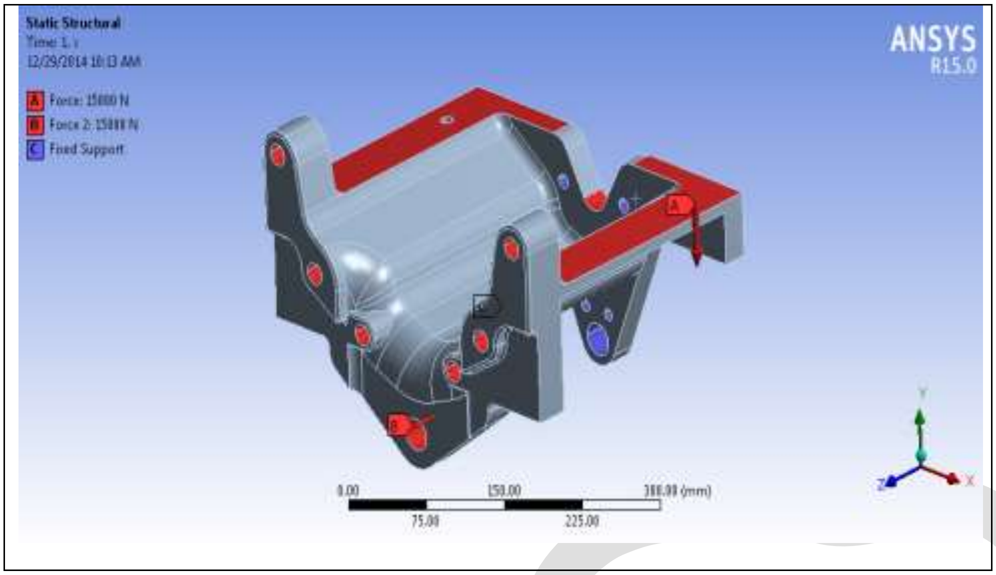

Fig. 2 Location of forces applied on FAB

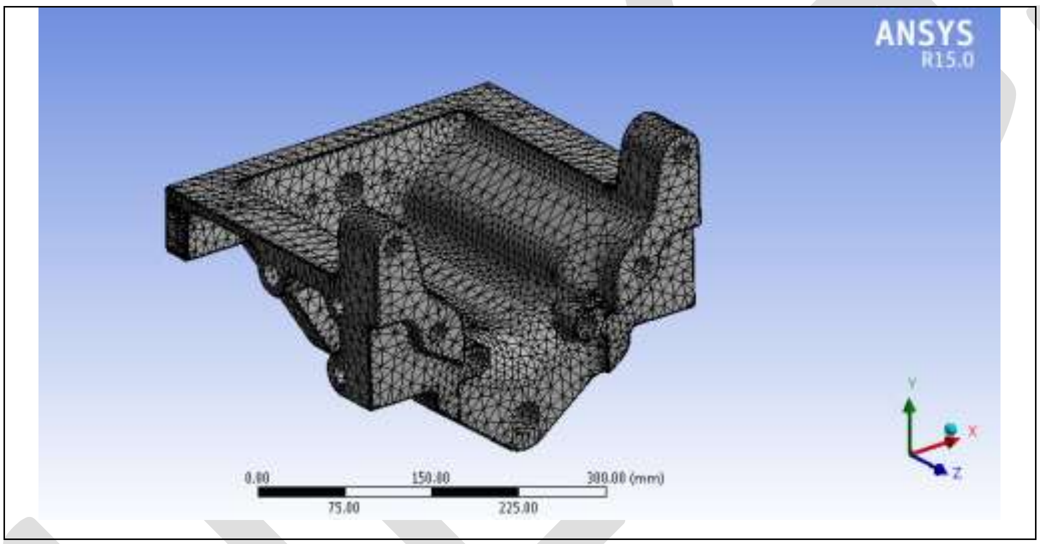

Fig. 3 Meshing of FAB

3.2.2 Solution

The results after computation are shown in Fig. 4 and 5.

3.2.3 Post processing

Reviewing the results, the values of stress and deflection obtained for front axle bracket are shown in Table 2

Table2: Results of Existing design

S.N.	PARAMETER	VALUE
1	Max Von Mises Stress	20.50 MPa
2	Max Deflection	5.2784 e-004 mm
3	Mass	31.287 Kg

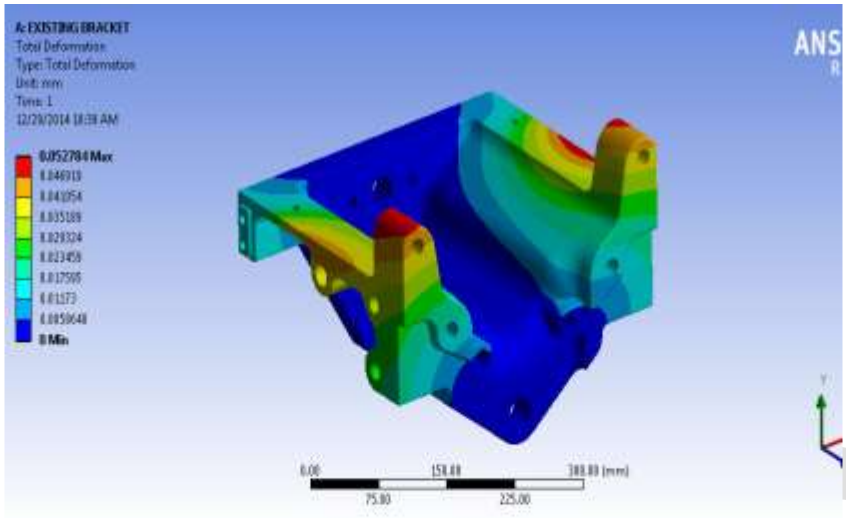

Fig.4 Stress produced in existing design

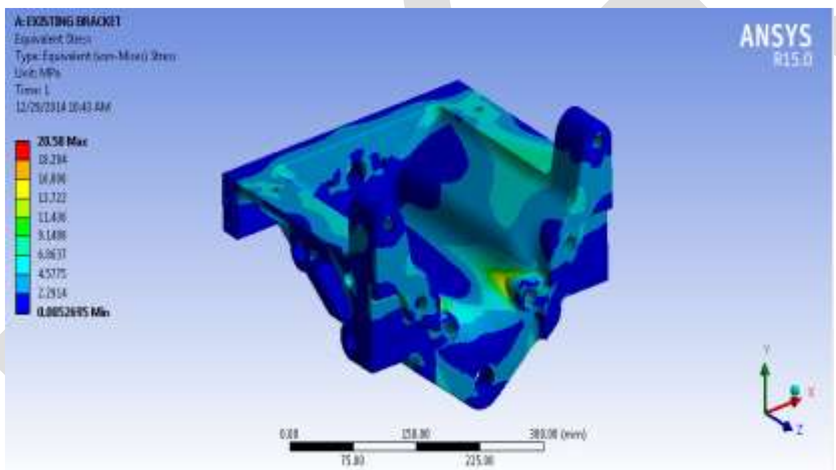

Fig. 5 Deflection generated in existing design

3.3 The uncritical area for engineering scope

Figure 6 shows the locations A, B, C for a possibility of modification. A shows the position of no load region. In this position no direct load is in contact of the tractor engine. B shows the position where the sideways tapering is to be minimized. C shows the region of the thickness of that section.

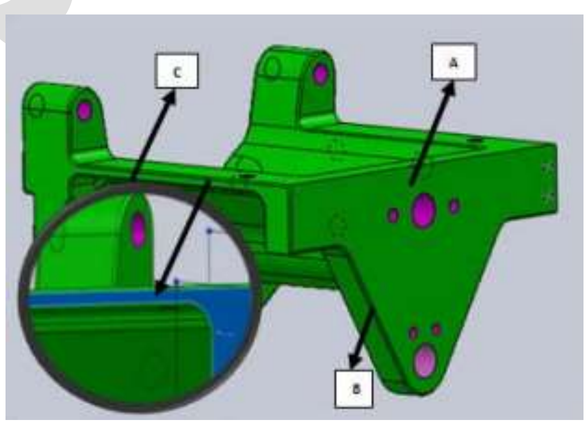

Fig. 6 The uncritical area for engineering scope

4. DESIGN OPTIMIZATION

The existing design has been discussed in CAD Model generation. The proposed designs are discussed under Optimized Design I, II and III

4.1 Optimized Design I

In this design a slight section is cut from the critical section A as shown in Fig. 7. For analysis of stress, the load applied in this design is 15000 N acting vertically downwards and same load acting at front direction over the entire span of the bracket. The front axle support is subjected to tensile load on the upper span of the bracket. The maximum stress of front axle support is monitored and it is not more than the allowable stress. The deflection generated is shown in the Fig 8.

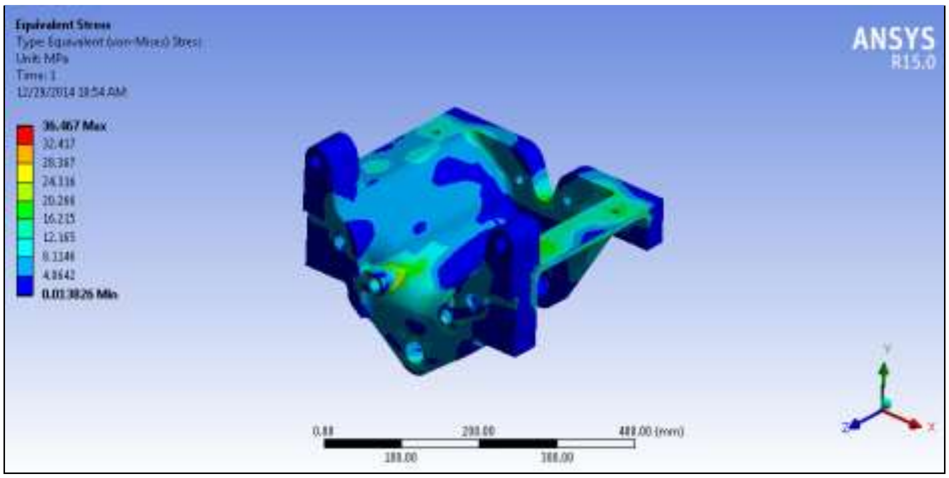

Fig.7 Stress Produced in Design I

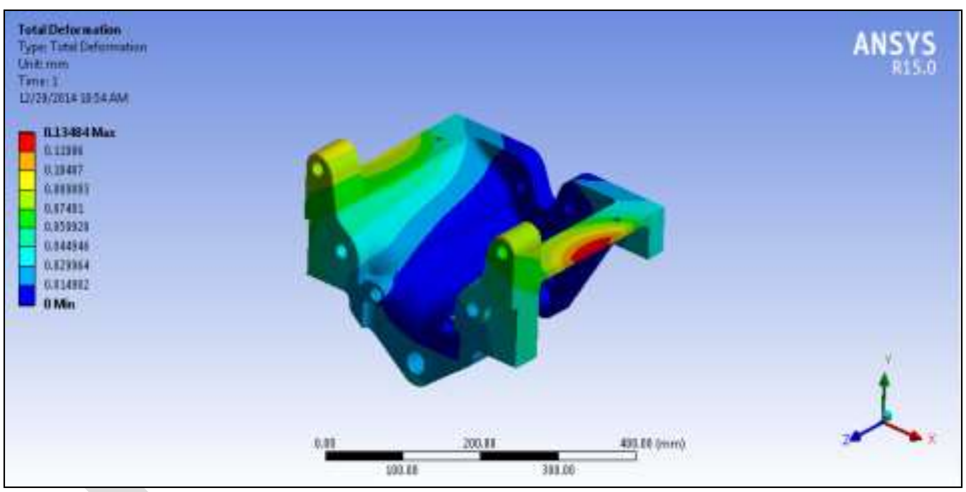

Fig. 8 Deflection generated in Design I

Table 3: Results of optimized design I

S.N.	PARAMETER	VALUE
1	Max Von Mises Stress	36.46 MPa
2	Max Deflection	11.984e-002 mm
3	Mass	30.921 Kg.

4.2 Optimized Design II

In this design, section B is considered for optimization. The arc length is reduced and the angle is tapered by which considerable amount of weight is reduced. The amount of stresses and deflection increases but it is in safe limits. The stress generated and the deflection is shown in Fig. 9 and 10.

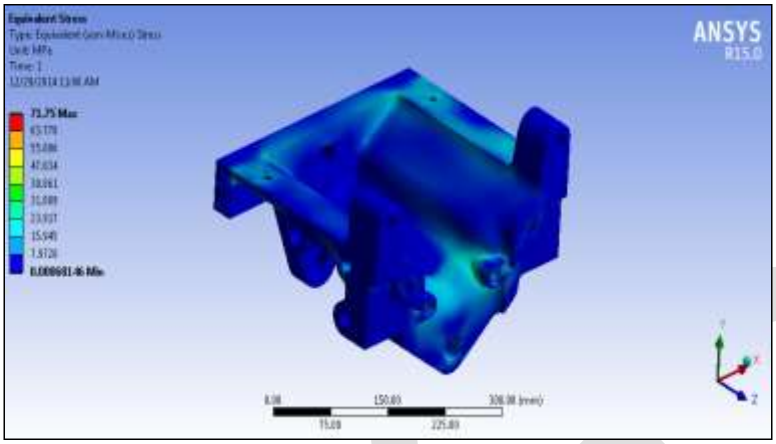

Fig. 9 Stress generated in Design II

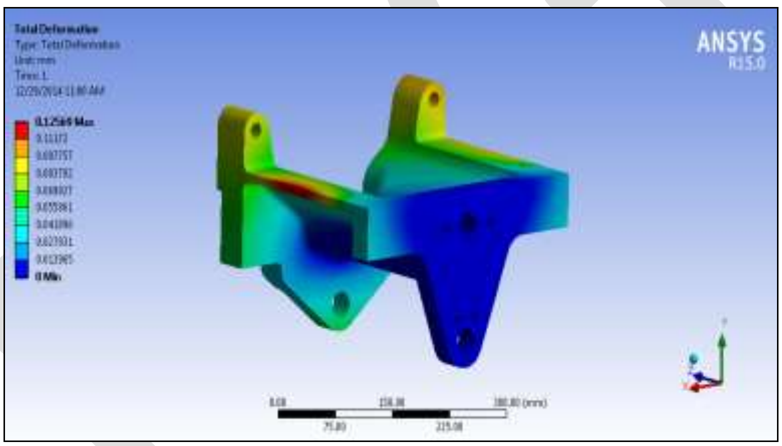

Fig. 10 Deflection generated in Design II

Table-4 Results of optimized design II

S.N.	PARAMETER	VALUE
1	Max Von Mises Stress	71.75 MPa
2	Max Deflection	12.569 e-002 mm
3	Mass	30.45 Kg.

4.3 Optimized Design III

The third and final design includes the modification of design I & II with reduction in thickness of the front and rear wall. It is reduced from 55 to 50mm and 30 mm thick wall is changed to 25mm.The stress generated is shown in Fig. 11 and the generated deflection is shown in Fig. 12.

Table-5: Results of optimized design III

S.N.	PARAMETER	VALUE
1	Max Von Mises Stress	45.89 MPa
2	Max Deflection	15.00e-002 mm

3	Mass	26.39 Kg.

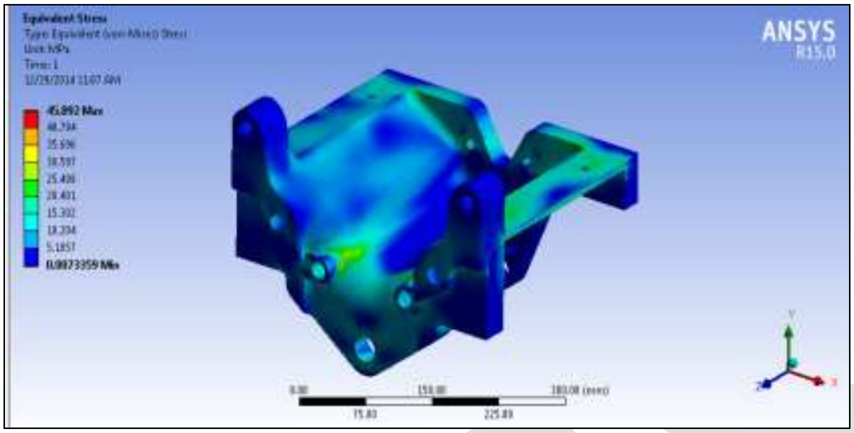

Fig. 11 Stress generated in Design III

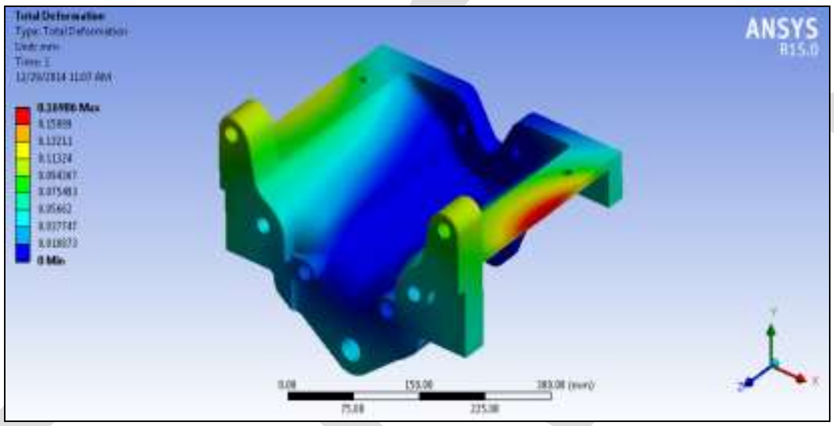

Fig. 12 Deflection generated in Design III

Table – 6: Analysis of the Results

S No.	PARAMETER	Existing Design	Design- I	Design - II	Design - III
1	Max. Von Mises Stress, MPa	20.5	36.46	71.75	45.89
2	Max Deflection, mm	5.27e-004 mm	11.9e-002 mm	12.5e-002 mm	15.0e-002 mm
3	Mass, Kg	31.287 Kg.	30.921 Kg.	30.45 Kg.	26.39 Kg.

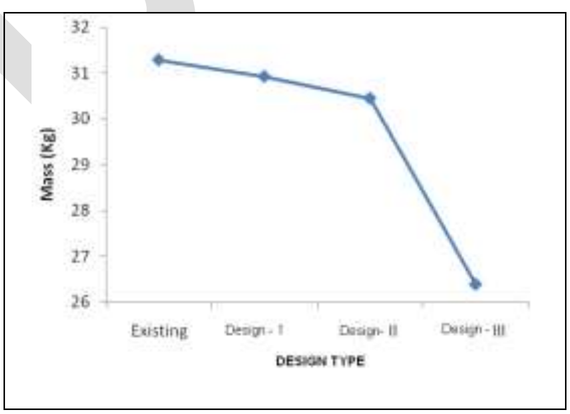

Fig. 13: variations in weight in different Designs

CONCLUSION

The three different designs have been adopted to analyze the optimum configuration for the purpose of reducing the weight of FAB without affecting the functional quality of the bracket. The results of design I, II & III are tabulated in Table-6. The variations in weight in different Designs are shown in Fig. 13. The design I results show that the amount of maximum stress was developed in area near to the right flat plate. Design I and II separately gives near about 1 and 2 percentage reduction in the weight but it shows higher amount of stress so this method is not considered from design point of view. Moving forward to design III, the reduction of material by combining design I and II is implemented along with reduction in thickness of both the walls. It gives better result with less amount of stress. Nearly 15 % weight has been reduced (weight reduced to 26.39 Kg from existing 31.287 kg). The amount of Von Mises stress is increasing from 20.5 to 45.89 MPa but it is in safe limits. The optimization analysis by design III is manufacturing feasible for given strength and life with less design cycle time.

REFERENCES:

[1] Javad Tarighi et al, *Static and dynamic analysis of front axle housing of tractor using finite element methods*, Australian Journal of Agriculture Engineering, 2(2): 2011, pp 45-49.

[2] http://www.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/confpaper/2002-Int-ANSYS-Conf-106.PDF.

[3] http://www.continuummechanics.org/cm/vonmisesstress.html.

[4] https://www.solidworks.com.

[5] http://en.wikipedia.org/wiki/Finite_element_method

[6] http://www.engr.uvic.ca/~mech410/lectures/FEA_Theory.pdf.

[7] Shree Dewangan et al, *Shape Optimization Of Front Axle Support Of Tractor*, Int. Journal of Engineering Research and Applications, Vol. 4, Issue 4, 2014, pp.70-73

[8] http://www.ansys.com

[9] Dong Fang Hu, Wen Hui Liu, *Finite Element Analysis of the Tractor Front Axle Housing Based on ANSYS*, Applied Mechanics and Materials (Volumes 105-107), 2011, pp 168-171

[10] http://www.mahindratractor.com