International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 11, November - 2013

Weight Reduction Techniques Applied to Control Arm Using Optistruct

[1]T. Chandra Sekhar [2]Dr. P. V. R Ravindra Reddy [3]Sk. Shakur

[1]P.G.Student, [2]Professor, [3]P.G.student, *Mechanical Engineering Dept., Chaitanya Bharathi Institute of Technology, Hyderabad-75, AndraPradesh, India.*

ABSTRACT

In the quest for reduced vehicle mass without sacrificed integrity, Computer Aided Engineering (CAE) topology optimization software was investigated and utilized in the design of the Indian Maruti Suzuki vehicle as a means to determine the optimum material distribution within a component for a given set of loading and boundary conditions.

This work looks at the design of a front suspension control arm component using modern topology optimization techniques and compares the end product to that of the 2010 model control arm component, which was designed using more traditional techniques.

A hydraulic load cell system was created to simulate the vehicle suspension forces and was used to physically test the original and optimized parts to failure. Through the use of Altair Optistruct topology optimization software, the same hydraulic loads are applied to the virtual component and checked the results.

Control arm was designed in 3D modelling software then imported in to Altair Hyper mesh for pre-processing; to solve Altair Radioss was used. Altair Optistruct is used for optimum material distribution in the component to get weight reduction. After getting reduced component from Optistruct again modified model is analyzed by using same loads and constraints.

1 INTRODUCTION

Weight reduction has become a primary concern in automotive industry. In fact, safety standards and emission regulations impose conflicting performance targets that need to be satisfied at the same time. While the respect of the safety standards pushes the automotive design process towards heavy weight solutions, environmental issues and handling call for a resolute vehicle weight reduction.

Over the last twenty years the average vehicle weight has steadily risen due to the improvements in safety and the growth in number of the vehicle features. This brought to the increase in the aluminium content of vehicles with the aim of restraining their weight, and also to a growing interest towards composite materials, even though their application is still limited to parts of some high performance prototype vehicle for cost reasons.

Apart from the quest for better materials, remarkable weight saves can be also obtained by adopting a new approach in design involving optimization techniques. Optimization is a powerful tool for systematic design in mechanics; it can lead to sensible improvements that could not be achieved with a simple trial-and-error approach.

In order to apply these techniques a suitable parameterization of the investigated problem together with the definition of the objectives and the targets which are sought are needed. The optimization algorithm iteratively generates new samples. .Topology optimization is a non-traditional optimization technique, particularly suitable for solving structural mechanics problems at an early design stage using finite elements analyses. It aims at finding the optimum material distribution within the domain given by a finite elements mesh. In a different way than more traditional algorithms, it has the peculiarity that it can change the topology of the object by virtually digging holes in the domain at locations where the algorithm, from local gradient computations, thinks it is less needed. This is made possible by adopting a parameterization based on a fictitious element-by-element material density. Various ways for formulating such an optimization problem exist.

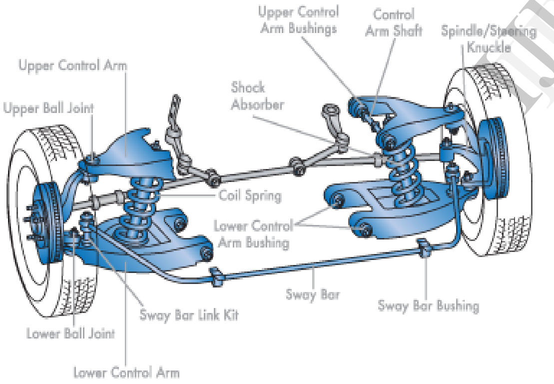


Figure 1 Diagram of a control arm suspension

2 MODELLING OF CONTROL ARM

2.1 MODELLING USING CATIA V5R19

The CATIA V5 R19 3D model and 2D drawing model is shown below for reference. Dimensions are taken from Maruti Suzuki OEM. The design of 3D model is done in catia v5r19 software, and then to do test we are using below mentioned software's.

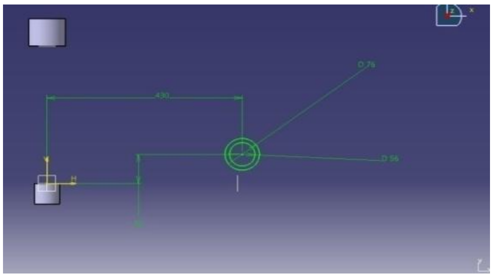


Figure 2 Control arm sketch

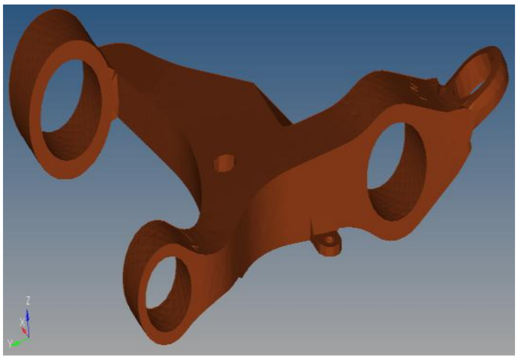


Figure 3 control arm component

2 .2 MESHING

TETRA-MESHING

Once the geometry was cleaned, the design space volume was filled with tetrahedral elements using the auto-mesh features of Hyper Mesh. This was done with a volume-tetra element with a nominal minimum size of 10 mm.Total number of nodes is 4710 and elements are 18345.

2.3 MATERIAL

Aluminium 7075-T0 material is used for control arm, the material mentioned in hyper mesh.

- Aluminum 7075-T0
- Young's Modulus (E) = 71700 MPa
- Poisson Ratio (μ) = 0.33
- Density = 3.09e-09 kg/mm^3

- **Mass of the control arm 9.414 kg**

2.4 CONSTRAINS

Table 1: Boundary conditions

Control Arm Fixing Locations	A	B	C
DOF Constrained	3(z is fix)	23(y and z is fix)	123(x,y,z translation is fixed)

In the above figure clearly mentioned boundary conditions for control arm. Left side fixing point is in z direction which is arrested in z direction. At right side of the bolt fixing location is in y and z direction. Centre hole location is fixed in x, y and z direction which doesn't move in 3 translation direction. The fixing points are assumed from the physical model and applied in virtual software and analyzed.

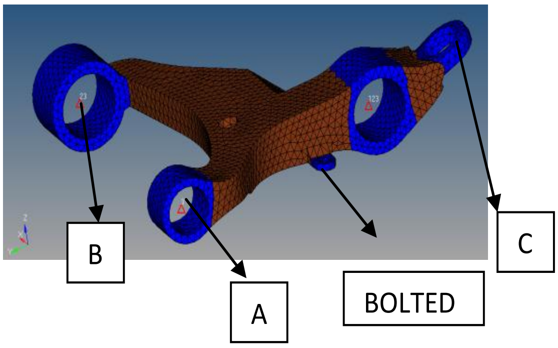


Figure 4 Boundary conditions.

2.5 LOADS

We have updated the finite element model to include these load vectors applied to the rigid spiders of nodes A and B, C. Furthermore, two additional "out of plane" 2 load cases were created in which first load cases will have steering rod applied load and torque and in second load force and torque is applied to the control arm.

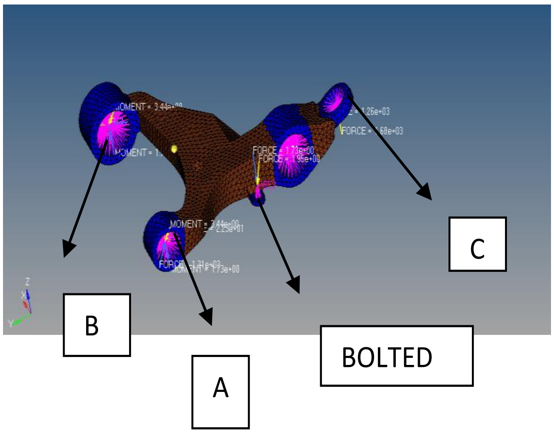


Figure 5 Forces and moments applied to control arm.

Force at three locations is different based on the physical effect to control arm force and moments are applied.

Load case 1 describes below for reference, at point 'A' force is 223 N about y axis and moment is 3.44 N, mm/deg about z axis. At point 'B' force is 303 N about x axis and moment is 3.44 N,mm/deg about z axis and point 'C" force is 168 N about z axis up words and clip point force is added at bolted location in load case 1 is 1.95 N about z axis down words.

For Load case 2 point 'A' force is 131N about y axis and moment is 1.73 N, mm/deg about z axis. At point 'B' force is 243 N about y axis and moment is 1.73 N,mm/deg about z axis and point 'C' force is 126 N about x axis down words. Bolted location in load case 2 force is 1.73 N about z axis.

3 ANALYSES

3.1 LOAD STEP PANEL

In the load step panel we are giving the defined loads and we are going to load step panel and in that different loads are giving .we can apply the loads with different load cases .so in this analysis we are giving two sub cases.

3.2 BASE MODEL RESULTS AND DISCUSSIONS

Displacement for load case 1

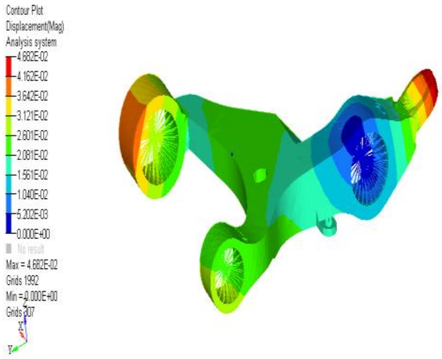


Figure 6 Displacement profile of control arm

The maximum displacement of control arm in load case 1 is 0.046 mm at point B. Point refers from the figure 4

Displacement for load case 2

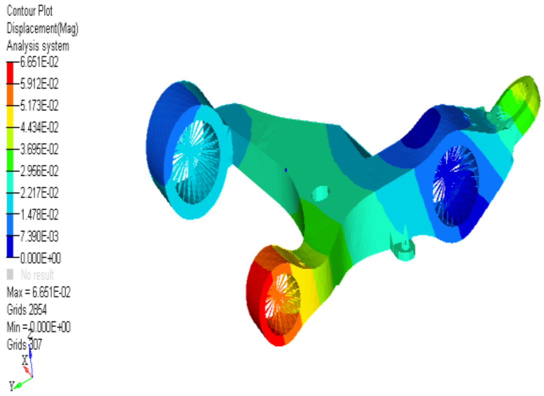


Figure 7 Displacement profile of control arm

The maximum displacement of control arm in load case 2 is 0.065 mm at point A Point refers from the figure 4.

Von Mises stress for load case 1

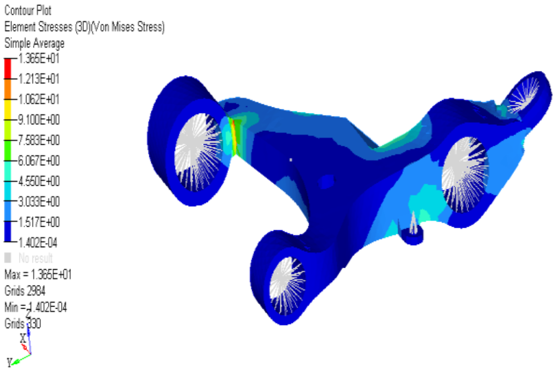


Figure 8 Von Mises stress profile of control arm

The maximum von mises stress are in load case1 is 13.65MPa at point B .Point refers from the figure 4

Von Mises stress for load case 2

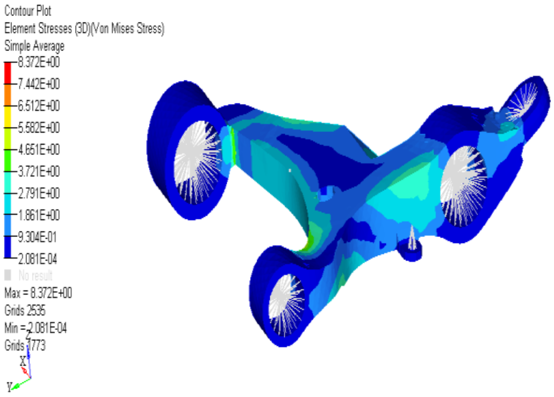


Figure 9 Von **Mises stress profile of control arm**

The maximum von mises stress is in load case2 is 8.37 MPa at point A .Point refers from the figure 4

After solving the problem in radioss will get result file which shown in above figures for load case 1 and load case 2, output file and h3d file for viewing the results in Altair hyper view. Results are viewed in

Altair hyper view for seeing the stress and displacement of the component we can assume that model is safe or not. By seeing above results model is safer which not crossed the yield point of aluminium 7075-T0 material is 103 MPa. Stress and displacement is very less for static analysis.

4 OPTIMIZATION PROCESS

First step is user profile should change to optistruct in hyper mesh interface.

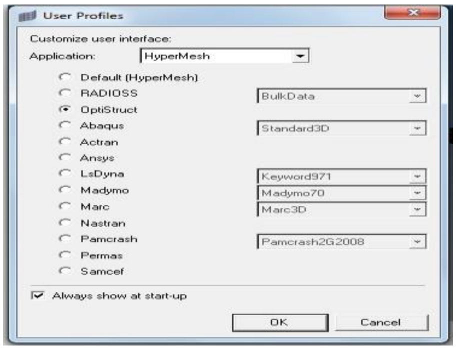

User Profiles
Customize user interface:
Application: HyperMesh
○ Default (HyperMesh)
○ RADIOSS BulkData
● OptiStruct
○ Abaqus Standard3D
○ Actran
○ Ansys
○ LsDyna Keyword971
○ Madymo Madymo70
○ Marc Marc3D
○ Nastran
○ Pamcrash Pamcrash2G2008
○ Permas
○ Samcef
☑ Always show at start-up

Figure 10 User profile to solve optistruct.

4.1Design variable

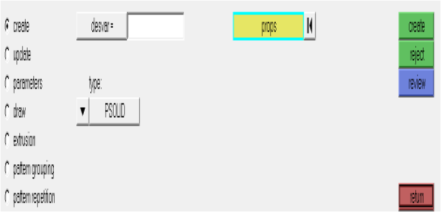

● create
○ update
○ parameters
○ draw
○ extrusion
○ pattern grouping
○ pattern repetition
desvar =
type: PSOLID

Figure 11 Design variable

Design variable should be mentioned in above figure as shown in above figure, in property selection we need to select the design area property were design changes are done using optistruct.

4.2 Draw direction

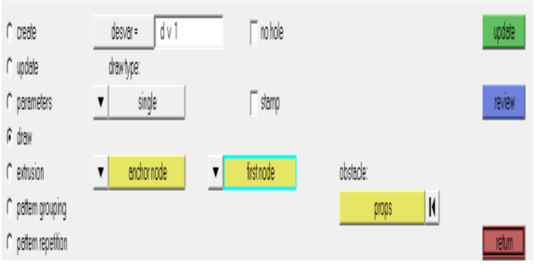

○ create
○ update
○ parameters
● draw
○ extrusion
○ pattern grouping
○ pattern repetition
desvar = d v 1
☐ no hole
draw type: single
☐ stamp
obstacle:

Figure 12 Draw direction panel

Draw direction is mentioned in optistruct by which optistruct will remove the material in that direction only. Figure 13 shows the the draw directions of control arm.

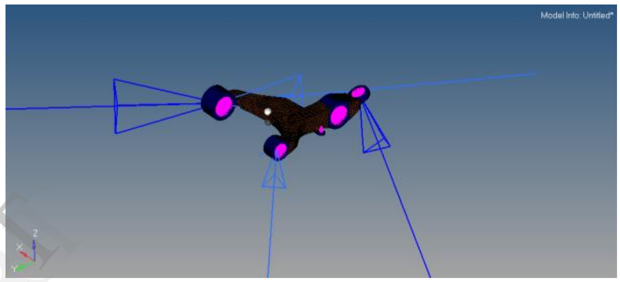
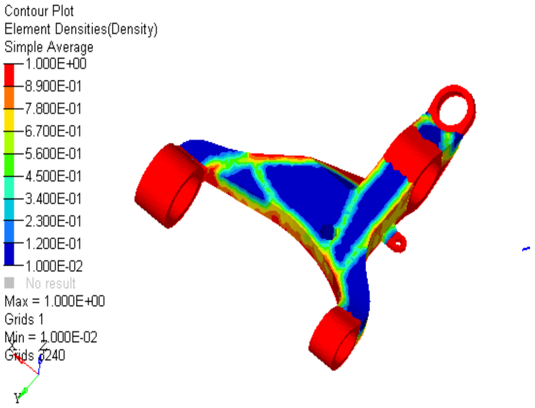


Figure 13 Draw direction

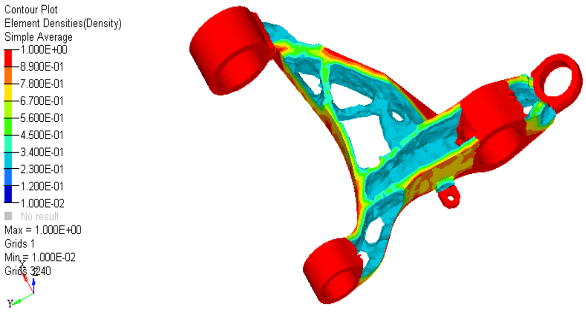
4.3 OPTIMIZED MODEL RESULTS

Based on above setup the optimized results



(a)

This is complete optimized model given by the software after removal of the huge material from the main geometry with consecutive iterations, by these we can tell that unnecessary material has been removed.



(b)

Figure 14 (a) Before optimization model (b) Complete optimized model

4.4 RE-DESIGN OF THE OPTIMIZED MODEL AND PRE-PROCESSING METHODOLOGY

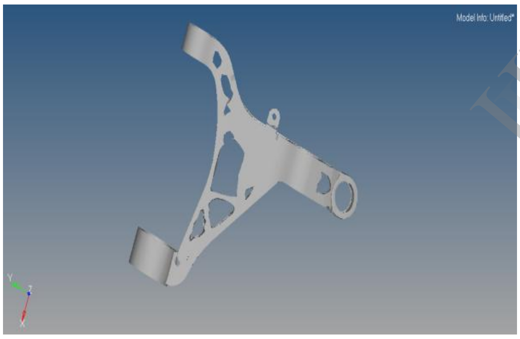


Figure 15 Mode **of optimized control arm**

Basic reference model is changed to the above design after applying the optistruct application to that. Design changes had been generated in hyper mesh using osssmooth option. Be that the figure4.21 is generated in catia. The design space was filled with tetrahedral elements using the auto mesh feature of hyper mesh. This was done with a volume tetra element with a nominal minimum size 10mm and the curvature and proximity adaption enabled to refine the mesh in the regions of more complex geometry.

Figure 16 shows optimized control arm meshed with tetra elements.

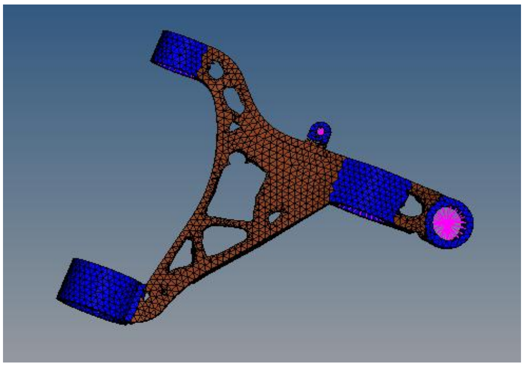


Figure 16 Meshed mode of optimized control arm

To this redesign model we have to assign material, thickness, load step and run the base run analysis as did for reference model.

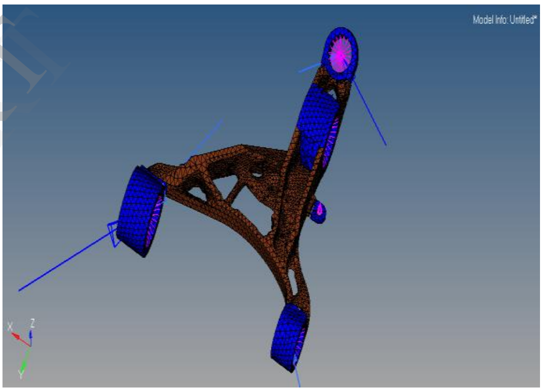
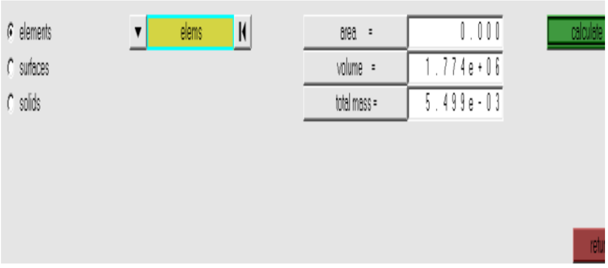


Figure 17 Forces and fixing location of optimized model

As same as base model loading conditions and forces we applied to the new concept model, which is shown in the above figure 17. Moments and forces are taken from methodology chapter.

Using Hyper mesh interface, Tool-page-count-Fe entities and select on the displayed option to get list of nodes and elements. Weight of the optimized control arm is mentioned in figure 4.24 is 5.49 Kg's.



● elements
○ surfaces
○ solids

elems

area = 0.000
volume = 1.774e+06
total mass = 5.499e-03

Figure 18 Mass of optimized model

Figure 18 shows hyper mesh interface, tool-page-mass calculation- click on calculates. It displays mass value as per density value given in material property.

4.5 RESULTS OF OPTIMIZED CONTROL ARM

Displacement for load case1

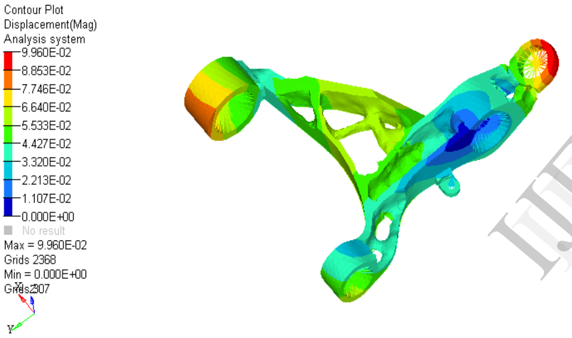


Figure 19 Displacement profile of optimized control arm

The maximum displacement of control arm in Load case 1 is 0.099 mm at point B .Point refers from the figure 4.

Displacement for load case 2

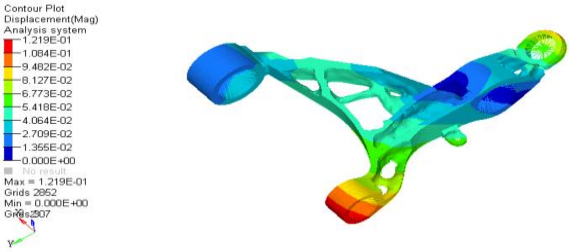


Figure 20 Displacement profile of optimized control arm

The maximum displacement of control arm in Load case 2 is 0.121 mm at point A .Point refers from the figure 4

Von Mises stress for load case 1

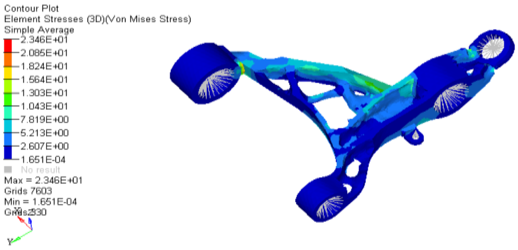


Figure 21 Von Mises stress profile of optimized control arm

The maximum von mises stress are in load case 1 is 23.46 MPa at point B.Point refers from the figure 4

Von Mises stress for load case 2

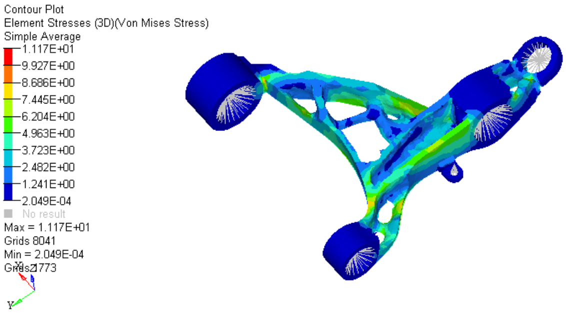


Figure 22 Von Mises stress profile of optimized control arm

The maximum von mises stress is in load case 2 is 11.17 MPa at point A. Point refers from the figure 4.

5 RESULTS COMPARISON FOR TWO DESIGNS

5.1STRESS COMPARISONFOR TWO MODELS

Table 2: Stress comparison of base model and optimized model

BASE MODEL LOAD CASE	OPTIMIZED MODEL LOAD CASE
Case 1 13.65 MPa	Case 1 23.46 MPa
Case 2 8.37 MPa	Case 2 11.17 MPa

The above table shows the comparison of stress of two designs, which is below the yield point value of aluminium 7075-T0 material. The yield point of aluminium 7075-T0 is 103 MPa.

5.2 DISPLACEMENT COMPARISON FOR TWO MODELS

Table 3: Displacement comparison for two models

BASE MODEL LOAD CASE	OPTIMIZED MODEL LOAD CASE
Case 1 0.046 mm	Case 1 0.099 mm
Case 2 0.065 mm	Case 2 0.121 mm

The above Table shows the displacement comparison of two models. One is base model and new optimized model. Displacement is below 1mm.

5.3 WEIGHT REDUCTION AND COMPARISON OF BASE MODEL

Table 4: Weight comparison of base model and optimized model

MODEL TYPE	WEIGHT OF THE CONTROL ARM
Base Model	9.41 Kg
Optimized Model	5.49 Kg
Percentage of Weight Reduction	3.92/9.41 *100 41.65 % Reduction

41.65 % of weight is reduced from the base model. As comparison of stress and displacement with base model which is very less for the optimized model.

6 CONCLUSIONS AND FUTURE SCOPE OF WORK

6.1 CONCLUSION

In this work, topology optimization approach is presented to create an innovative design of control arm. Final comparison in terms of weight and component performance illustrates that structural optimization techniques are effective to produce higher quality products at a lower cost.

The control arm is been used to reduction of the vibration created by car wheels. The control arm is made up of aluminium 7075-T0 material. In this work the weight reduction of control arm is taken under the consideration without varying the performance of the component. Firstly the process of the structural optimization involves the variation of weight from base to new model, which resulted to the 41.65% weight reduction of the existing industrial component.

The optimized control arm displacement changes in load cases below the 1mm and the von mises stress are below the yield point 103 Mpa .Weight of the control arm was reduced 41.65% and the optimized model weight was 5.49 kg..

6.2 FUTURE SCOPE OF WORK

The future work focuses on the cost reduction of the material without varying the weight of the component. After the careful analysis of the better material the product is further undergone to topology optimization using hyper works software. The manufacturability of the component is been analyzed using the Altair radioss and optistruct analysis. Future work is cost analysis of materials, which is having less cost that material is applied for the physical model.

REFERENCES

1 .SAE International. “Collegiate design deries: Formula SAE Series.” 2006.

2. Fornace, L.V., Davis, A.E., Costabile, J.T., Hart, J.D., Arnold, D., “Traction control systems: FSAE Vehicle acceleration optimization.”

3. Taylor, Rob. “F-35 Joint strike fighter structural component optimization Lockheed martin aeronautics company.” Optimization technology.

4. Talke, F.E., “Optimization: Computer aided analysis and design.” Class.

5. Schneider, Detlef, and Erney, Thomas. “Combination of Topology and Topography Optimization for Sheet Metal Structures.” Opticon 2000.

6. Meyer-Pruessner, Rainer. “Significant Weight Reduction by Using Topology Optimization in Volkswagen Design Development.” Optimization Technology.

7.Altair Engineering. “Optistruct 7.0 User’s Guide.” Hyper works 2012.

8. Hughes, Thomas J.R., “The Finite Element Method, Linear Static and Dynamic Finite Element Analysis.” New York: Dover Publications, 1987.

9. “Webster’s New World Dictionary.” New York: The World Publishing Company, 1967.

10. Wikipedia. “The Free Encyclopedia: Bell Crank Definition.” 2006.

11. Wright, P., “Formula 1 Technology.” Warrendale: SAE International, 2001.

12 .Altair Engineering. “Altair Motion View: Pre-and Post Processing for Multi- Body Dynamics, Volume I.” Hyper works Training Manual, 2004.

13. Altair Engineering. “Altair hyper mesh: Introduction to FEA: Pre-Processing.
Volume I.” Hyperworks Training Manual, 2004.