Anand Aggarwal / International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com
Vol. 2, Issue6, November- December 2012, pp.1317-1322

Design Modification & Analysis for Suspension Frame Using Finite Element Analysis

Anand Aggarwal
(as Masters in Technology –Machine Design project submitted to Desh Bhagat Engineering College, Gobindgarh)

ABSTRACT

Structural analysis is aimed towards decreasing the weight while maintaining structural integrity and strength. It may be a cross field functionality such as civil dams, bridges, flyovers, power transmission towers OR components related to one engineering discipline like automotive where much physical testing is done along with computer simulations to arrive at results. Field data and past records for failure are also very essential as without these there is no historical data supporting the performance of the products. While all the above information is hardly available at one place, effort has been made to analyze a structural component by changing its design keeping stress within limits.

I. INTRODUCTION

Suspension frame was modeled where dimensions were drawn from an existing model and two models were created. First is an as existing model of the frame while second was modified so as to reduce some manufacturing time by simplifying the shape of an important part.

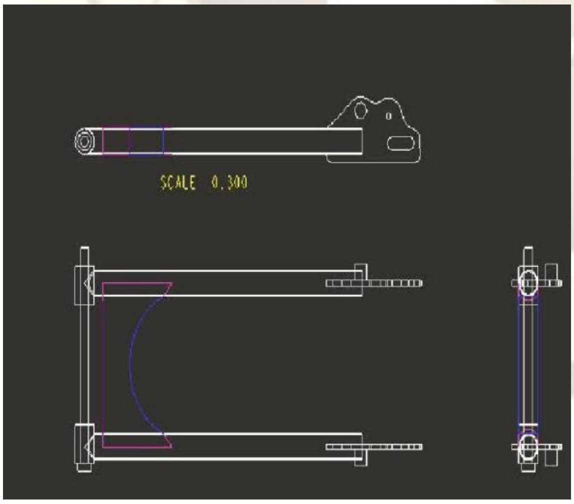


Fig 1.Existing model 2D Layout

II. MOTORBIKE CHASSIS

A common motorbike chassis is a complex assembly of different components like suspension frame, shocker-spring assembly, sheet-metal central frame, bolt connecting the suspension frame with the rest of the chassis.

Due to strength requirements, chassis is made from tubular cross section.

It is the backbone of the motorbike which carries weight of all members and puts all components in its proper position.

The aim was to structurally modify a design so as to make it simpler to manufacture while maintaining strength.

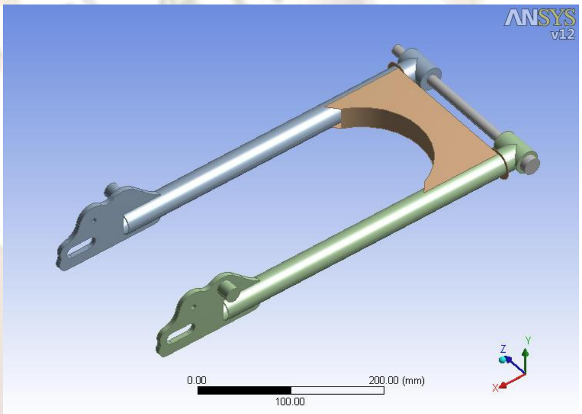


Fig2. Existing Model 3D as generated into workbench

III. ANALYSIS LAYOUT

Workbench is a very user friendly technology from Ansys where all the required components of an analysis can be defined at one place like geometry,engineering data containing information about materials being used , meshing and post processing results.

TABLE: contact regions

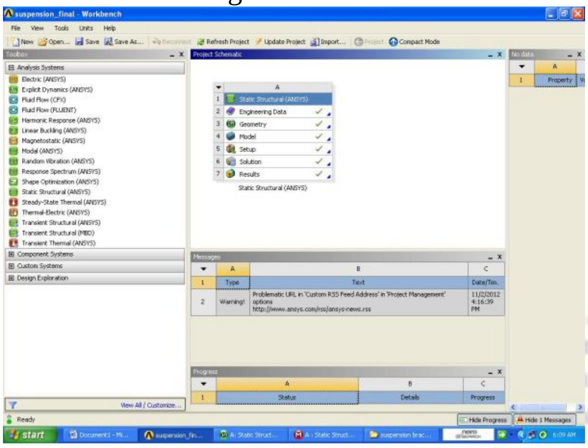


Fig 3. Analysis layout in Ansys

Further the model is imported into workbench geometry from 3D CAD and since all the parts are welded together of the suspension frame, Bonded connections are used to simulate the joints which are automatically picked up when a model is imported. Four additional manual contact regions are defined which are shown in the following images

Contact region 1	auto
Contact region 2	auto
Contact region 3	auto
Contact region 4	auto
Bonded 1 to 1 part	manual
Bonded 2 to 2 part	manual
Bonded 1 to 1 lower	manual
Bonded 2 to 2 lower	manual

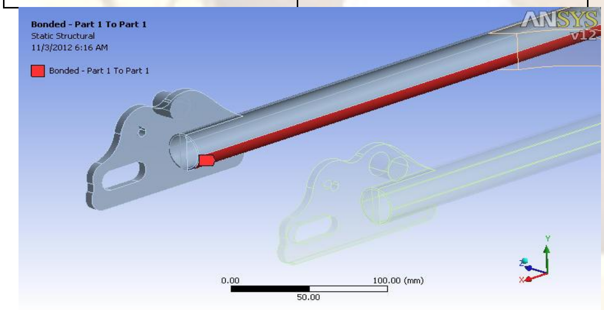


fig4.

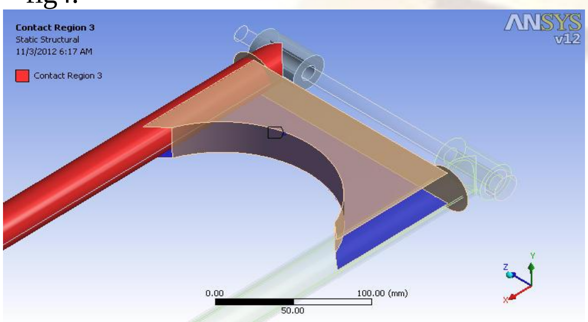


Fig 5.Bonded part 2 to 2 lower surface
Auto contact - bonded

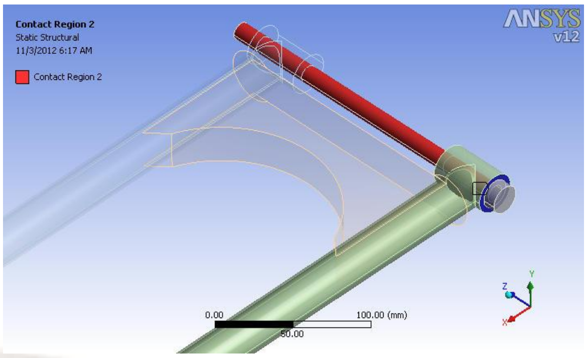


fig 6 .

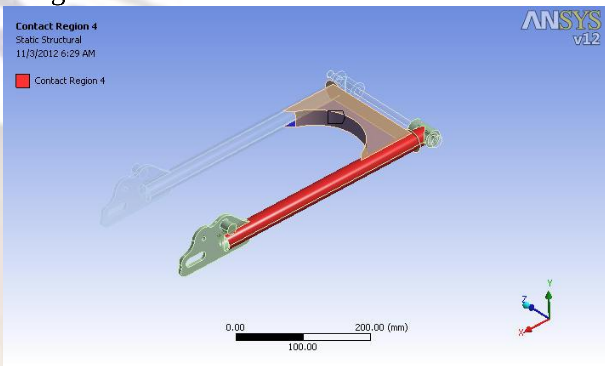
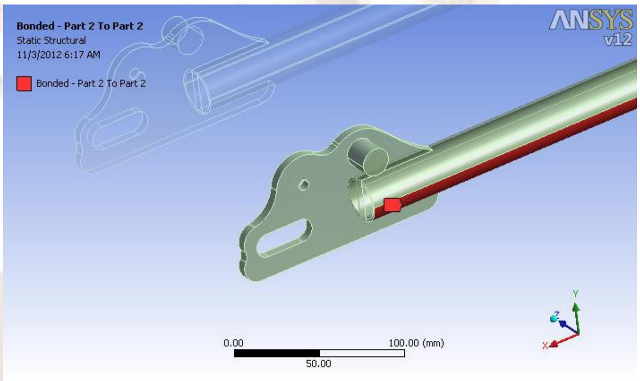
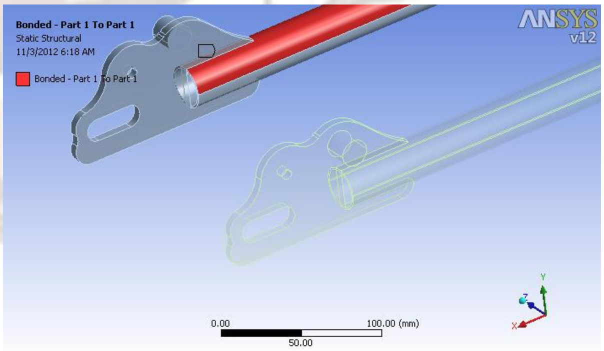


Fig 7





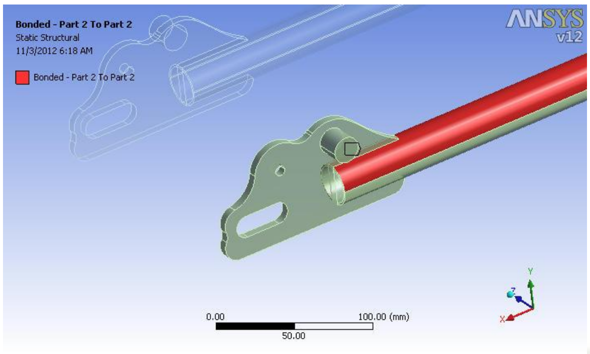
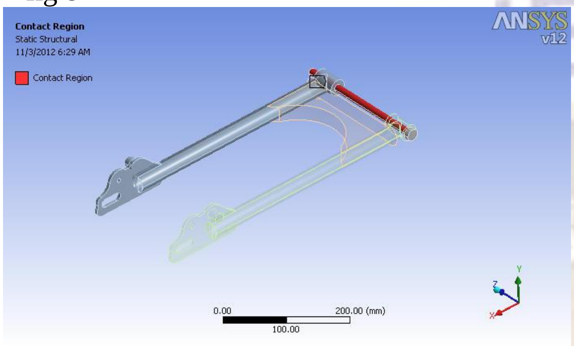


fig 8



IV. MATERIAL PROPERTIES USED: this is the same in both models

Young's modulus:	200,000 Mpa
Poisson ratio	0.3
Density	7850 Kg/m3
Tensile yield strength	250 Mpa
Ultimate Tensile Strength	460 Mpa

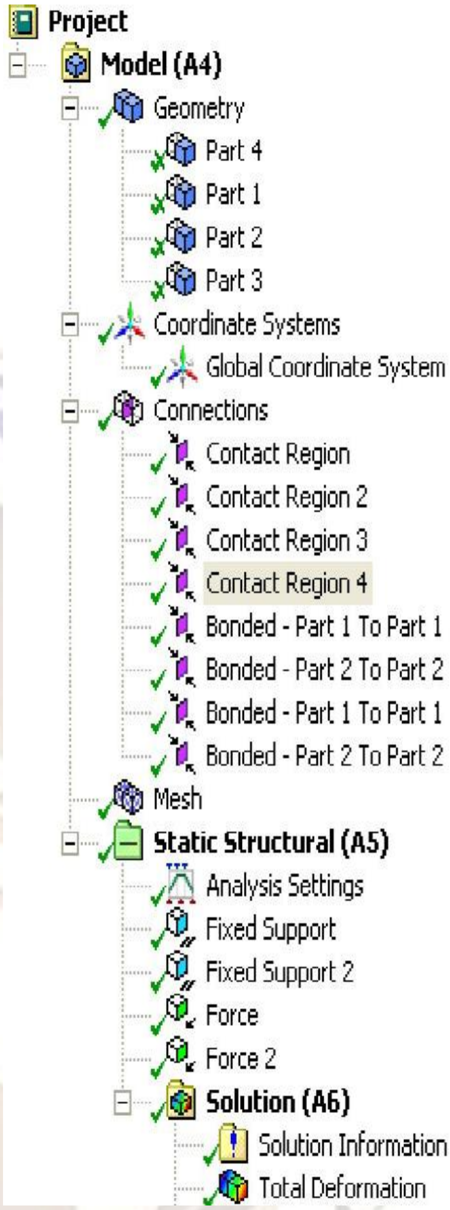
V. BOUNDARY CONDITIONS AND LOADS

A suspension frame acts like a beam in actual which is supported from front and rear with the help of a bolt and from rear on rear axle. Loads are transferred to it from suspension, the main load being that of person riding the motorbike. We have additionally taken the load as two persons of weight 75 KG each i.e.

Load = 75kg *2= 150 Kg= 1500 N= 750 N on each of the face having revolute joint with spring system.

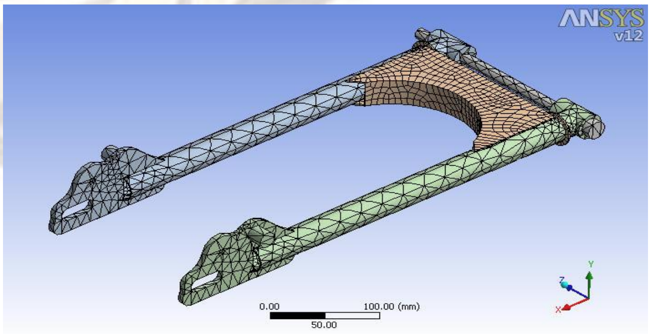
The solution was obtained for the maximum deformation and von mises stresses.

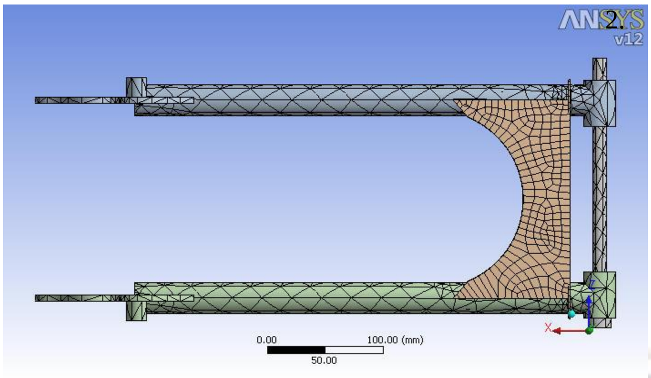
The solution tree looks like the following in the original scenario when all data has been input into simulation.

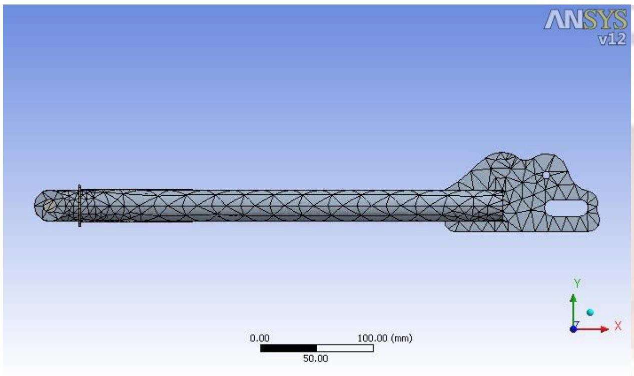


VI. MESHING THE MODEL

Nodes: 10848
Elements: 5619

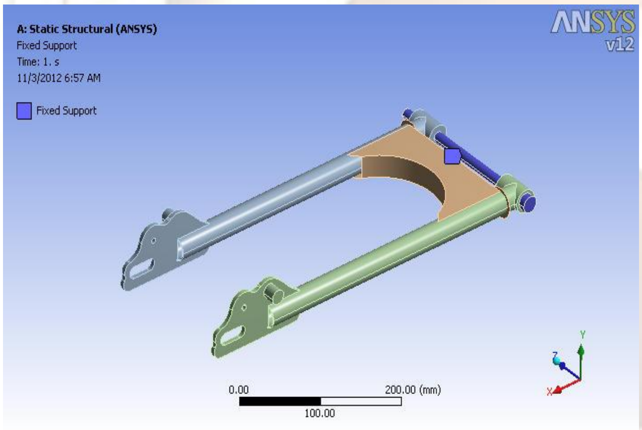




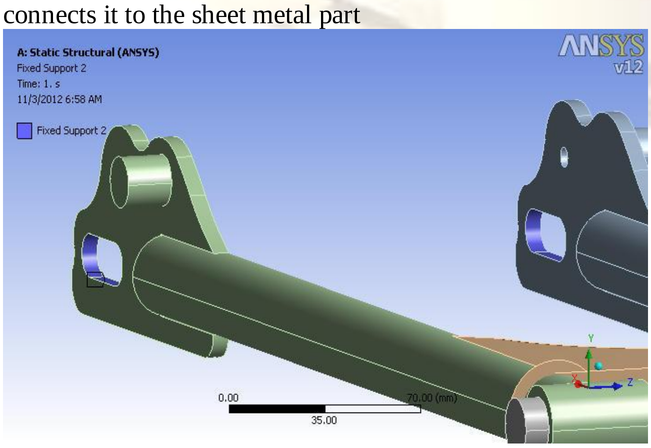


Elements used are default 3D Tetrahedral elements because of the irregular complex shape of the frame. No errors were reported from messages area of the simulation.

BC's: same in both cases



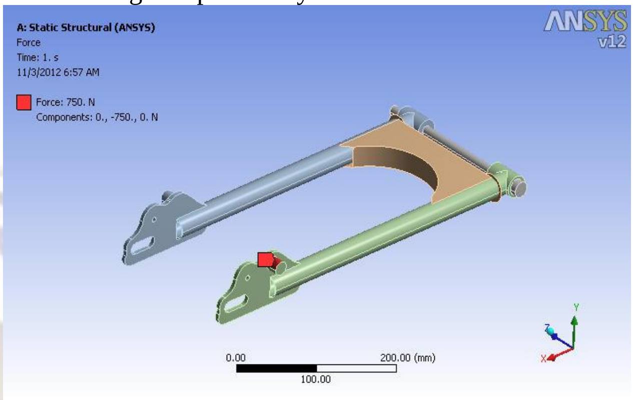
1. The bolt supporting the frame from front end which connects it to the sheet metal part

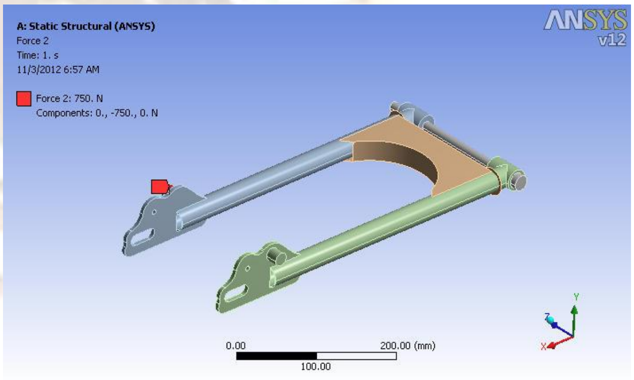


2. The rear end of the frame which connects it to the rear axle of the vehicle

LOADINGS: same in both the cases:

A vertical downward force of 750 N assuming that all the load from both riders is to be born by the frame through suspension system.



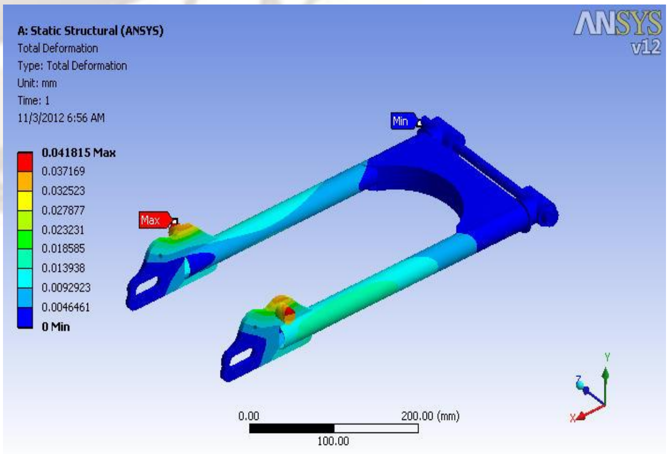


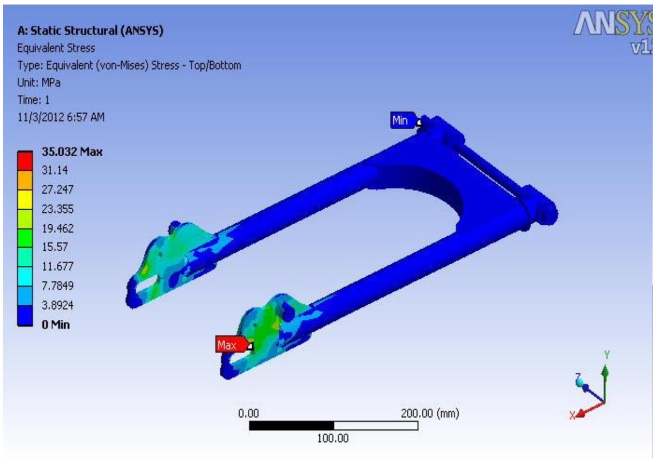
VII. RESULTS FROM ANALYSIS OF THE ORIGINAL KIND:

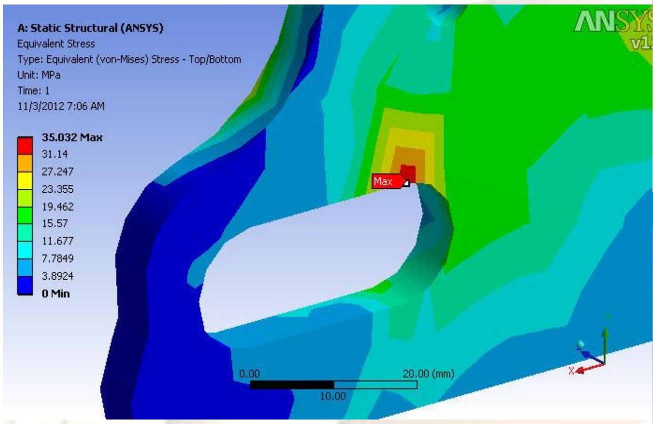
Total deformation .041815 mm

Von mises stress 35.032 Mpa

These values are quite below that of the yield strength even if a factor of safety of 1.5 or 2 is considered.



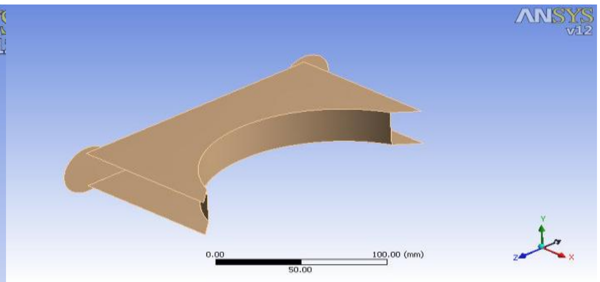


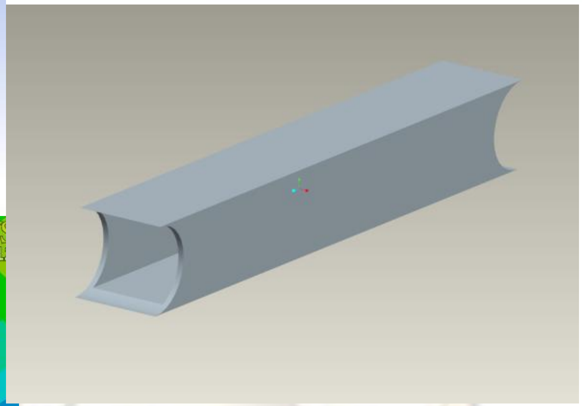


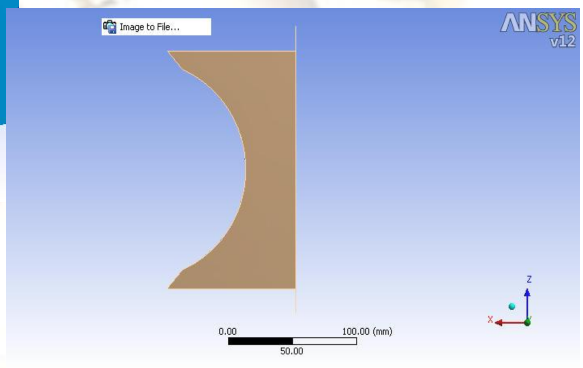
The enlarged image shows the area where maximum value of the stress is concentrated.
As is clear from above that the existing model is safe from all strength and structural consideration irrespective of the exact material being used since we have assumed a common material for structural applications.
Chemical compositions of the material have not been considered and there has been no test of the material.

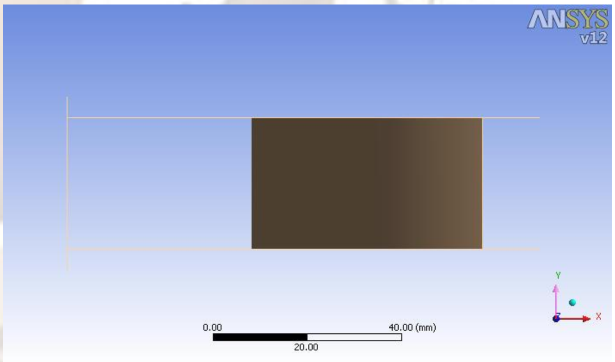
VIII. DESIGN MODIFICATION TO THE ABOVE EXISTING MODEL

After analyzing the above model a design modification has been suggested and studied for analysis so that it reduces the complexity while maintaining the strength and other parameters.
Material properties, boundary conditions, loadings, connections are same as that of the existing design.





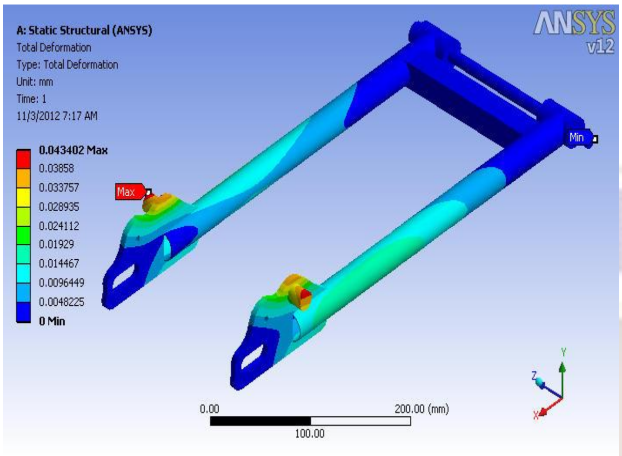


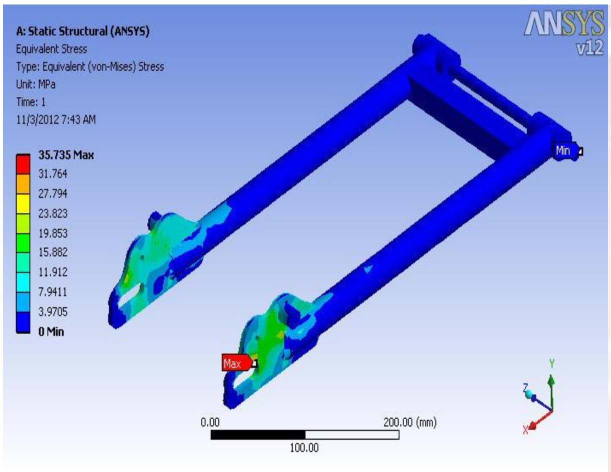


In this central part of the suspension bracket a sheet metal part is used to give support to the members which prevents the part from torsion loads and from coming apart.
This is a typical shape where space for the rotation of tyres is also given and takes manufacturing time to get this into shape.
Instead a simple tubular cross section was proposed which is easily available in the commercial market 25*25*1.6 thickness made of the same material as that of the parent part.

If this part passes the strength requirements then it could possibly reduce time and cost of the suspension frame.

Analysis was performed to see the stresses and deformations with this part as shown in the images below.





Total deformation .043402 mm
Von mises stress 35.735 Mpa

IX. CONCLUSION

Comparison of two designs:

Case	Deformation(mm)	Von mises stress(Mpa)
1	.041815	35.032
2	.043402	35.735

As is clear from the analysis, making a small change to the design could save manufacturing time and costs.

X. FUTURE WORK:

As a extrapolation of the study, torsion characteristics can be studied which will validate the design for the bending and torsion of the suspension frame.

REFERENCES

[1] ANSYS 12.0 user guide: Structural Analysis

[2] Sujatha C&V Ramamurthi, Bus Vibration Study, Experimental Response to Road Undulation, Intl J Vehicle Design , Vol11, no 415,1990

[3] Johanssan & S, Eslund, Optimization of Vehicle Dynamics in Truck by use of full vehicle FE models,IMech E , C466/016/93,1993

[4] Ansys 12.0 Material Library.

[5] Marco Evangelos, Bian Colini, Carlo Brutti, Eugenio Pezuti, Shape Optimization of Structural Design by means of Finite Element Method,XII ADM International Conference, Italy, Sep5th-7th 2001.