OPTIMIZATION STUDY ON TRAILER ARM CHASSIS BY FINITE ELEMENT METHOD

Mohammed Fouzan K[1], Raghavendra K[2]

[1]*P.G. Student, Mechanical Engineering Department, B.I.T.M., Bellary*
[2]*Assistant Professor, Mechanical Engineering Department, B.I.T.M., Bellary*

Abstract:

Chassis is the important part of an automobile. It supports the body and different parts of an automobile. Chassis consists of engine, brakes, power train, steering system and wheels mounted on a frame. The frame is the main part of the chassis on which remaining parts of chassis are placed. The chassis should be rigid enough to withstand the twist, shock, stresses, vibrations and bending moments to which it is subjected while vehicle is moving on road. The trailer arm chassis frame has to carry and sustain the heavy loads which are applied on it. Hence it is very important to design and analysis of the trailer arm chassis frame. The design of trailer arm chassis is carried out by taking the base model structure of chassis as a standard. The optimization technique to redesign Chassis (Frame) of a trailer is carried out here. The trailer had dimensional limits and must be able to reduce the overall size and shape and still lift the same amount of load. Different load cases with given boundary conditions & loadings are used. To improve the performance of the chassis finite element analysis is carried for various alternatives. The analysis is performed by varying the thickness, shape and material to get the best possible design of the chassis. Static analysis is carried out for both basic and modified designs to determine the high stress regions, maximum displacement, and normal stress at critical positions of the chassis. Normal modal analysis for base model is carried out to find the first natural frequency. Normal modal analysis is carried out for all modified designs to improve the first natural frequency of the chassis in order to avoid the resonance. The whole challenging task, starting with pre processing, analysis and post processing is completed using Altair's HyperMesh, Abaqus and HyperView FE package.

Key Words: Trailer arm chassis, Static analysis and Modal analysis.

----------***----------

1. INTRODUCTION

Transportation industries place a major role in the economy of modern industrialized and developed countries. The total and relative volume of goods carried on trailers is dramatically increasing. To support the vehicle a structural unit is used known as chassis. The chassis safely supports the weight of the vehicle components and transmit the loads that result from longitudinal, lateral and vertical acceleration that are experienced in a racing environment without failure [3]. Chassis is an Frȩnch term and was initially used to denote the frame parts or basic structure of the vehicle. A vehicle without body is called chassis. The chassis provides the structure which connects the front and rear portion of the vehicle without excessive deflection. After the invention of automobiles a little attention was paid to the design of chassis frame. The other components of the chassis like gears and axles were of much importance. The first chassis was made up of wood and ash being the materials used. The cross members were forged iron and joined on the frame [2]. The difficulty has been increased when accidents tends to occur more with other failures which were directly related with the poor design of the chassis. Then the automobile engineers thought of building the chassis frame with more strength and less in weight. Now-a-days the well designed chassis frame has increased the tendency to take the more loads and holding the parts with the proper alignment. The number of mountings on the chassis has increased to greater degree which will relieve the stress to a greater extent when the vehicle moves on the rough roads.

2. TRAILER ARM CHASSIS

A good chassis is the one which has the optimum stiffness requirements in order to allow a safe drive under the most adverse road and traffic conditions. The static and modal analysis techniques are significantly used for automotive chassis structure design. Static analysis is carried out to find the characteristics of chassis at static condition so that stresses can be found out at the initial loading condition of the chassis. The dynamic characteristics of the chassis are studied in order to find out the natural frequency to avoid the resonance which occurs during the working condition of chassis. Resonance occurs when the excitation frequency of the chassis becomes equal to the natural frequency [1]. Therefore investigation on dynamic characteristics is carried out to control unpleasant noise and vibration by designing a chassis whose excitation frequency is well within the natural frequency. The type of chassis used in this project is a trailer arm chassis. This project work is mainly focused on design and optimization of trailer arm chassis. The chassis is designed by using modeling software and is analysed by using finite element analysis. The chassis is designed by varying thickness, shape and material.

And some other main objectives of work are,

- To study the design of existing trailer arm chassis.
- To generate the geometric modeling of existing chassis.
- To generate the finite element modeling of the chassis.
- To carry out all necessary checks on the model.
- Static and modal analysis is performed on chassis to calculate the stress, displacement, and natural frequencies.
- Recommendation of best suitable Design for Trailer Arm chassis.

It is vital that the correct material and section type is used to ensure that each part will be able to endure the load it will experience; hence proper material grade selection should be done to improve the life of the component.

3. METHODOLOGY OF THE WORK

The procedure followed in order to do the analysis work is done by initially taking the three trailer arm chassis models. Modal analysis is carried out for the three designs considered by meshing the chassis and applying the boundary conditions, where the meshing and analysis is done in ALTAIR-HYPERWORKS V12. Static analysis is done by taking the best model along with the whole assembly and a static load is applied over the structure. Different loading conditions and different materials are considered for the analysis which are done by ABAQUS software as a tool. The obtained results are viewed in HYPERVIEW and graphs are plotted for each of the results obtained.

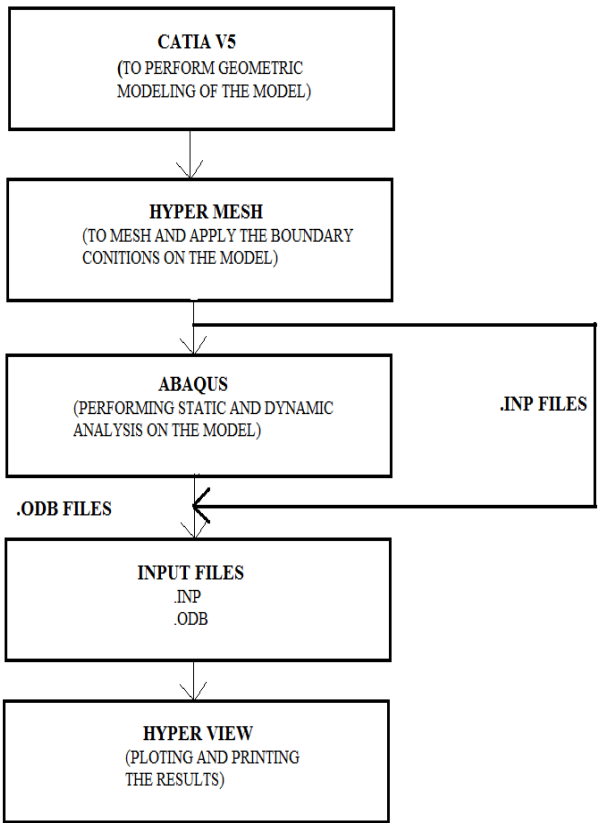

Figure 1: Flow chart of Chassis analysis

4. MATERIAL PROPERTIES AND DESIGN OF CHASSIS

In this topic concepts of trailer arm chassis are studied in order to set the boundary condition to obtain better accuracy in the results after analysis, also selection of materials and their properties are discussed in brief.

4.1 Model selection and study

Three trailer arm chassis models are taken for the analysis study which is represented in the following figures.

Figure 2: Trailer arm chassis model 1

The above figure shows the model1 taken for the analysis work.

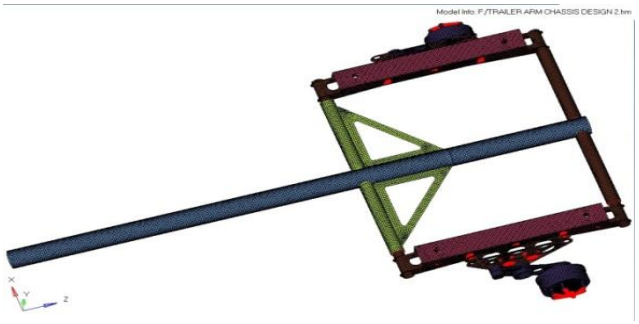

Figure 3: Trailer arm chassis model 2

Figure 4: Trailer arm chassis model 3

The chassis models shown in the above figures are considered for the modal analysis in order to find the natural frequencies and the mode shapes, among which the model with the highest natural frequency are subjected to static load with different loading conditions and different material considerations.

4.2 Material

It is important to choose the correct material and section type is used to ensure that each part will be able to endure the load it will experience. The type of material to be used for chassis depends on its application requirements and operating conditions. The material is decided on the basis of its rigidity, strength, cost, durability and reliability. The material selected should also be easy to fabricate. Fabrication plays an important role in material selection. As strength of material increases, cost of fabrication increases.

Steel started to be used as material for decoration in automobile. However, in recent years, it is mostly used as material for the chassis. It is because those stainless steels with good performance of high temperature characteristics and high corrosion resistance[4]. Material used for structural analysis of Chassis is listed in Table1.The isotropic material properties used in a FEA analysis are as follows.

Table 1: Mechanical properties of materials used in Chassis analysis

Sl. No	Mechanical property	Steel	Cast Iron	Wrought Iron
1	Young's Modulus	$2.10x10^5$ N/mm^2	$1.20 x10^5$ N/mm^2	$1.90 x10^5$ N/mm^2
2	Poisson's Ratio	0.30	0.28	0.30
3	Density	$7.89x10^{-9}$ Ton/mm^3	$7.20x10^{-9}$ Ton/mm^3	$7.75x10^{-9}$ Ton/mm^3
4	Yield Stress	370 N/mm^2	130 N/mm^2	210 N/mm^2

4.3 The boundary conditions is set for the models as shown in the figure 5,6,7.

The proper specification of boundary conditions is just as important for dynamic analysis as it is for static analysis. The improper specification of the boundary conditions leads to incorrect answers. One such improper specification of boundary conditions is forgetting to fully constrain the structure [5]. Unlike static analysis, for which an under constrained model does not run, an under-constrained model does run in dynamic analysis. We should perform a Normal mode analysis first and verify that there are no unwanted rigid-body modes.

The boundary condition is set for the model1as shown in fig.5, which indicates that it is in the fixed condition since it the fixed part in the whole assembly of the part.

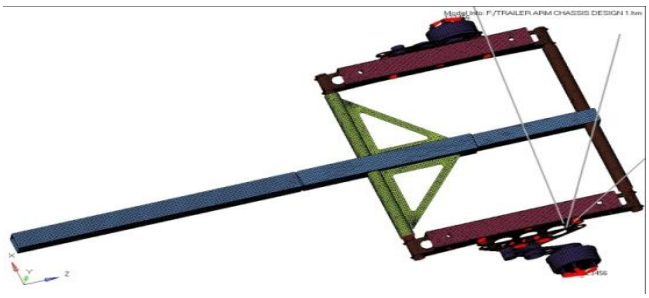

Figure 5: Boundary condition applied on chassis model 1

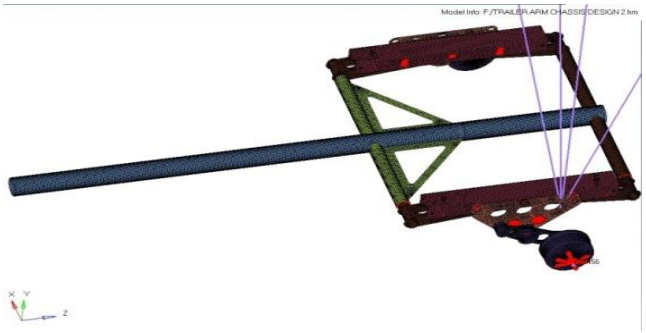

Figure 6: Boundary condition applied on model 2

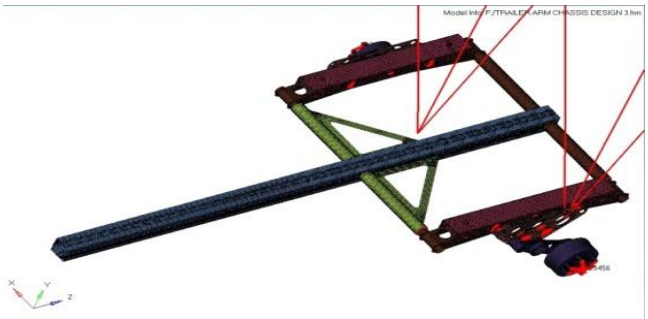

Figure 7: Boundary condition applied on model 3

Once the boundary conditions are applied on the models, the natural frequencies and mode shapes are found which are listed in the below tables.

Table 2: Mode Shapes on Different Natural Frequencies of chassis model 1

SL. No	Modes shapes	Description	Natural frequency (Hz)
1	1^{st} Mode	Bending mode	17.96
2	2^{nd} Mode	Linear mode	20.41
3	3^{rd} Mode	Linear mode	44.38
4	4^{th} Mode	Bending mode	53.43

The table 2 shows the different values of nautral frequencies and mode shapes obtained for model 1.

Table 3: Mode Shapes on Different Natural Frequencies of chassis model 2

SL. No	Modes shapes	Description	Natural frequency (Hz)
1	1^{st} Mode	Bending mode	19.66
2	2^{nd} Mode	Linear mode	21.51
3	3^{rd} Mode	Linear mode	44.10
4	4^{th} Mode	Bending mode	51.85

The table 3 shows the different values of nautral frequencies and mode shapes obtained for model 2.

Table 4: Mode Shapes on Different Natural Frequencies of chassis model 3

SL. No	Modes shapes	Description	Natural frequency (Hz)
1	1st Mode	Bending mode	18.19
2	2nd Mode	Linear mode	20.76
3	3rd Mode	Linear mode	44.64
4	4th Mode	Bending mode	48.02

The table 4 shows the different values of nautral frequencies and mode shapes obtained for model 3.

Table 5: Natural Frequency of chassis for Different materials

Sl no	Material	First Natural Frequency in Hz
1	Steel	19.66
2	Cast Iron	18.55
3	Wrought Iron	19.64

The table 5 shows the values of nautral frequencies For different materials.

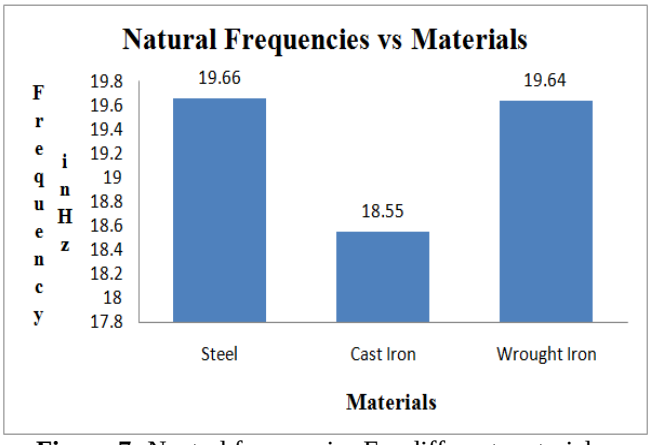

Figure 7: Nautral frequencies For different materials

By comparing design iterations of normal mode analysis the first fundamental natural frequency of chassis design 2 is higher . When compared to masses of the chassis, the design 2 gives the optimum mass and the natural frequency for design 2 satisfies the design standards so design 2 is considered further analysis such as static analysis.

4.4 Static Analysis

Static analysis is employed to know the maximum stress, displacement developed on the chassis and .The name Static indicates that the load on the chassis is constant, and load is not varying with respect to time [6]. Before analysis the structure is meshed for finite meshing in HYPERMESH software. Static analysis is performed with 5g, 5g,-10g loads applied on the chassis in x,y,z directions respectively. The boundary conditions are applied and all degrees of freedom (DOF) on the chassis are constrained. The displacements of chassis for different materials are separately obtained and compared at different loading conditions. The materials considered for the analysis in order to fond the stress and displacement at different loading condition are Steel, Cast iron and Wrought iron.
The obtained results are tabulated for stress, displacement for chassis with different materials are as listed below.

Table6: Stress and displacement on chassis for different materials

Sl no	Material	Stress in Mpa	Displacement in mm
1	Steel	234	6.74
2	Cast Iron	246	7.52
3	Wrought Iron	243.40	6.76

The stress and the displacement values obtained for Steel is comparatively very low to that of the cast iron and Wrought iron material.

Material wise comparison also done to choose best material and to optimize the design. Following figure shows material wise comparison.

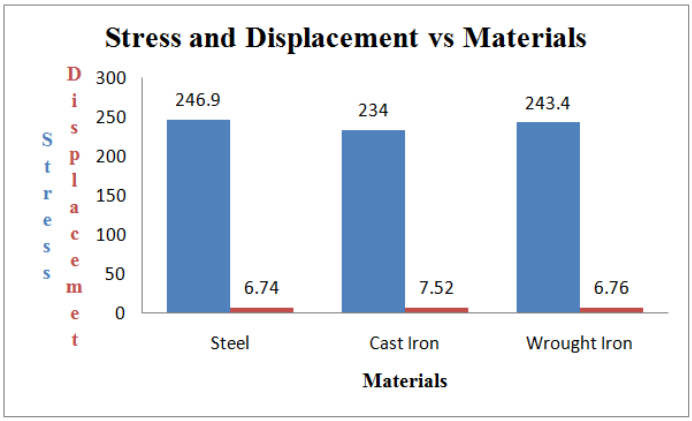

Figure 8: Stress and displacement for different materials

The above graph shows the material wise comparision for Steel, Cast iron and Wrought iron and also shows the Stress and displacement vs materials, Where the Stress and displacement for Steel is lesser.

5. RESULTS AND DISCUSSIONS

The natural frequency obtained for model 2 is 17.96Hz and for the model2 is 19.66 Hz for model 3 is 18.19Hz of which both the 1st modes are bending modes, hence it can withstand further conditions. The natural frequency with higher values is chosen because it has less mass compared to model1 and model 3.
Further Static analysis was carried out to find stresses and displacements at 5g, 5g,-10g loading conditions for the materials Steel, Cast iron, Wrought iron. In these obtained

values when we refer to graphs or table, we can clearly see that Steel has less stress and displacement values as compared to Cast iron and Wrought iron. Steel can withstand maximum loads with less displacement and stress.

6. CONCLUSION

In this study work optimization study of Trailer arm chassis of is carried out to improve the performance on 5g, 5g,-10g loading condition has been done by considering different materials like Steel, Cast iron, Wrought iron. Initially three chassis designs are selected based on the vehicle requirement and considered the same designs for the optimization process to reduce weight of the models. For meshing of the models HYPERMESH software and for the analysis ABAQUS was used. Analysis of models was done at 5g, 5g,-10g loading conditions for different materials by setting appropriate boundary conditions. From the analysis results it has been concluded that Steel is the best material to manufacture the selected Chassis design because it can withstand maximum loads with less displacement and stress.

7. REFERENCES

[1]. Mehdi Mahmodi –K. "Stress and Dynamic analysis of optimized trailer chassis" Original Scientific Paper, pp. 599-608, issn: 1330-3651, 3(2014).

[2]. Ahmad O Moaaz, Nouby M. Ghazaly. "A Review of the Fatigue analysis of the heavy duty truck frames" American journal of engineering and research, vol:3, issue:10, pp.01-06. 2014.

[3]. Haval Kumar Asker, Thaker Salin Dawood , Arkan Fawzi Zaid. "Stress analysis of standard truck chassis during ramping on block using FEM, ARPN journals of engineering and applied sciences, vol:7, no:6 issn:1819-6608 june 2012.

[4]. Shiva Kumar. "Fatigue life estimation of chassis frame bracket for commercial vehicle" International journal of science and research, vol:3, issue: 8, august 2014, issn:2319-7064.

[5]. Hemant B. patil. "Stress analysis of automotive chassis with various thickness"' journal of Mechanical and Civil engineering, vol:6, pp. 44-49, issn: 2278-1684, 2013.

[6]. Monika S. Agrawal. "Finite element analysis of truck chassis", international journal of engineering sciences and research technology, 2(12), pp. 3432-3438, issn:2277-9655 December:2013.