

Fatigue Life Estimation of Chassis Frame FESM Bracket for Commercial Vehicle

Shivakumar M.M¹, Nirmala L²

¹M-Tech Student, Dept. of Mechanical Engineering, K.S Institute of Technology, Bangalore, India

²Assistant Professor, Dept. of Mechanical Engineering, K.S Institute of Technology, Bangalore, India

Abstract: Automotive chassis is an important part of an automobile. The chassis serves as a frame work for supporting the body and different parts of the automobile it should be rigid enough to withstand the fatigue, shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate low damage and higher fatigue life. To improve the fatigue life and damage reduction are the two important criteria for the design of the chassis. This work deals with the analysis of the truck chassis. Structural systems like the chassis can be easily analyzed using the finite element techniques with a proper finite element model of the chassis. The work also deals with the static analysis, modal analysis of the chassis and fatigue analysis of FESM (FRONT END SUSPENSION MOUNTING) BRACKET of chassis, on the user requirements and improving its payload by changing the shape and thickness of the bracket which is attached to the present chassis frame. The FEM analysis is carried out with various alternatives. The truck chassis is modeled in CATIA V5 and the finite element modeling is carried out using Altair's pre-processing tool HYPER MESH. Static analysis is performed by using MSC NASTRAN/ANSYS to determine the high stress region, and also the maximum displacement. fatigue analysis is performed by using N-Code to determine the damage and life of the FESM bracket. The results are viewed through Altair's post-processing tool HYPER VIEW. it is observed that maximum stress induced is 155.7mpa which is less than the allowable stress of steel 205mpa, natural frequencies of six global modes were below 100 Hz, varying from 20 Hz to 90 Hz. Almost all of the truck chassis designs were based on these frequency ranges to avoid the resonance during the operating condition, the fatigue life of existed FESM bracket of the chassis is 13009.879 cycles has improved to 16857.432cycles and of damage of existed FESM bracket of the chassis is 1.149 it has been reduced to 0.602.

Keywords: chassis, static analysis, modal analysis, fatigue analysis.

1. Introduction

A chassis is defined as a structural unit that will support the full load of the vehicle drive train, body and all ancillary components. It is the backbone of the vehicle. A vehicle without body is called Chassis. The chassis Frame is made up of long two members called side members Riveted/welded together with the help of number of cross members together forms an integral structure for the support of all chassis equipment and payload The components of the vehicle like Power plant, Transmission System, Axles, Wheels and Tires, Suspension, Controlling Systems like Braking, Steering etc., and also electrical System parts are mounted on the Chassis frame. It is the main mounting for all the components including the body. So it is also called as Carrying Unit. The chassis frame used in this project is a

ladder frame chassis made out of structural steel. Thus using the chassis frame many mounting can be installed/mounted without considerable change in the overall design; this is also will be an economical. Construction of a heavy vehicle chassis is the result of careful design and rigorous testing. Consequently, a modification to the chassis frame should only be attempted after consultation with the vehicle manufacturer or engineer experienced in commercial vehicle chassis modifications to ensure that the proposed modification will not be detrimental to the vehicle's safety or performance. Because various manufacturers have individual design concepts and different methods of achieving the desired performance standards for the complete chassis, not all chassis components are interchangeable between various makes and models of vehicles.

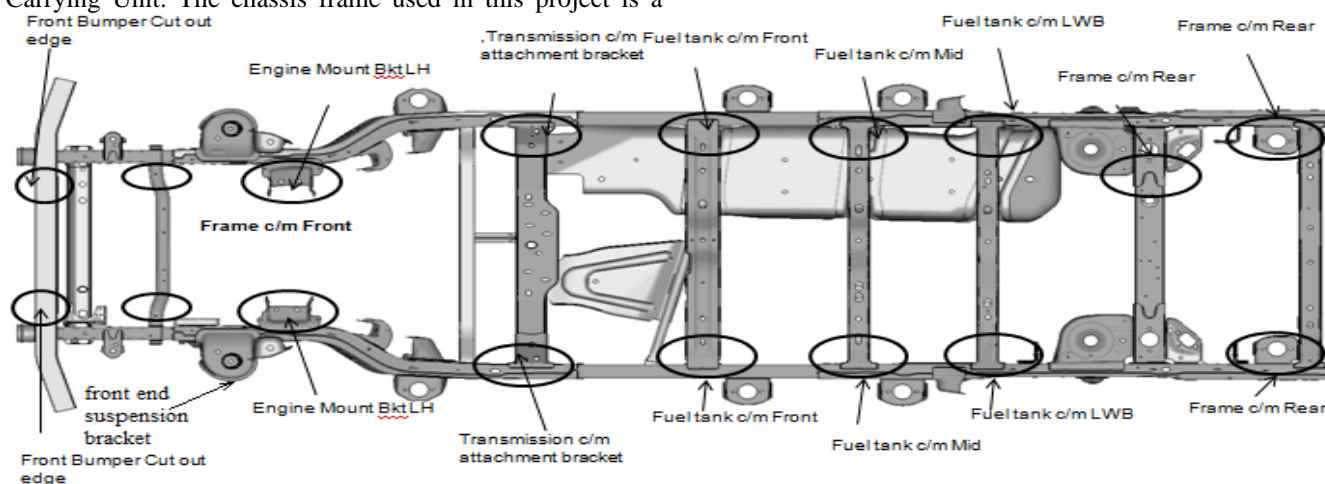


Figure 1.1: Parts of chassis

Volume 3 Issue 8, August 2014

www.ijsr.net

This project work is focused on Re-design of chassis by changing the shape and thickness of front end suspension mounting bracket (FESM), based on end user requirement to withstand the different super structure loads, the same structure will be statically analyzed by using FE analysis method to verify the reinforcement of chassis frame and also we will study the Modal behavior & Natural frequencies of the modified chassis frame.

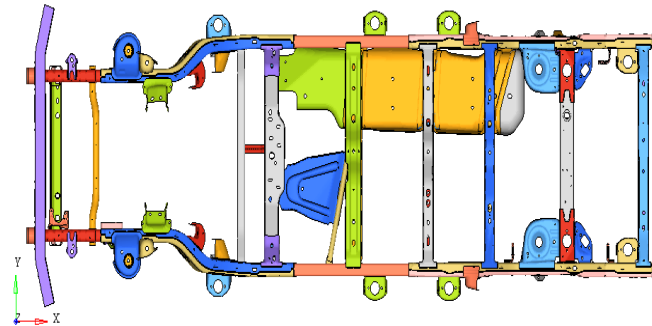


Figure 1.2: Complete chassis assembly

2. Geometrical configuration of the chassis

2.1 3D Model of existed chassis

The model is constructed by using CATIA V5R21, The explode view of the model as shown in Figure 2.1.

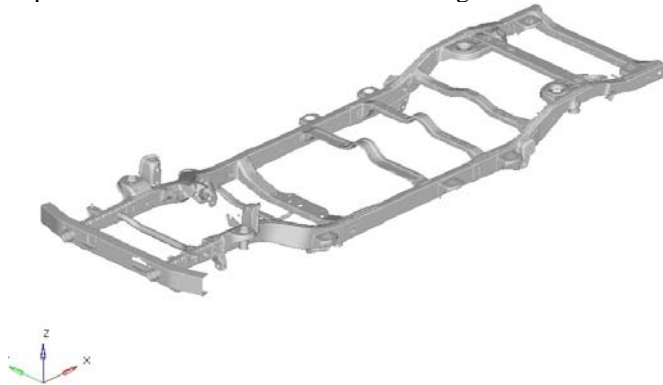


Figure 2.1: 3D Model

The total length of the chassis is 4400mm and width of the chassis is 1162mm. The mass of link is 332 kg.

2.2 Material properties of chassis

The mechanical properties of steel alloy material,
 Type of material: MS264_050_Tower
 Properties: Young's modulus= 2×10^5 MPa
 Density= 7.85×10^{-9} kg/mm³
 Yield strength = 410 MPa
 Poisson's ratio=0.3

3. Finite Element Method

Import the IGS file to Hypermesh and clean up the geometry. In meshing shell mesh was done on whole surface of the model.

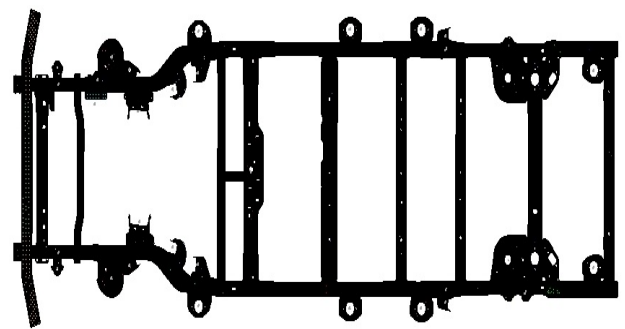


Figure 3.1: Meshed model of chassis

3.1 Loading and boundary condition

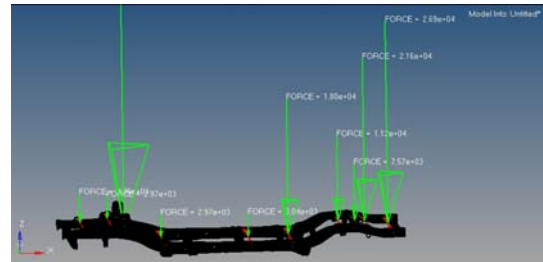


Figure 3.2: Load applied in negative Z direction

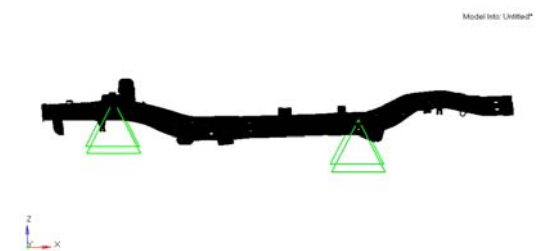


Figure 3.3: Constraints applied at suspension region

Figure 3.2, 3.3 Shows that loading and boundary conditions are applied on the chassis. As for the loading condition the point load applied on the chassis in negative Z direction and chassis is constrained at region of front and rear axle support. Also loads and constraints are applied on chassis to compare the results with theoretical values.

3.2 FEA Results of chassis

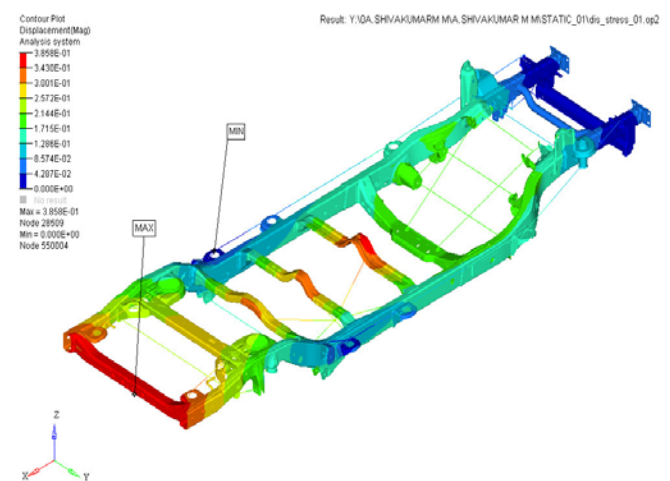


Figure 3.4: Maximum Displacement of the chassis for the given load condition is 0.3858mm.

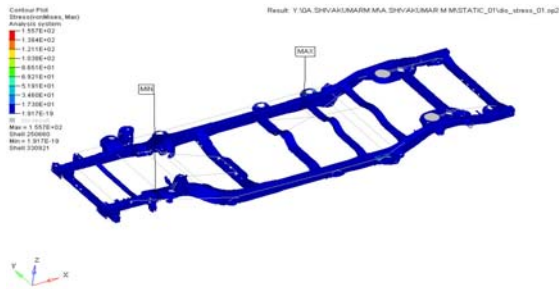


Figure 3.5: Stress contour for the chassis, from the figure it can be observe that maximum stress of 155.7 MPa.

Maximum stress=155.7MPa
 Maximum Displacement=0.3858 mm
 As the material is ductile, maximum Distortion energy theory (Von mises theory) is considered for determining the criterion for failure. Taking a factor of safety of 2.0, the allowable stress for the material is 205 MPa, based on Y.S of 410 MPa.

$$FOS = \frac{\text{critical stress}}{\text{allowable stress}}$$

$$FOS = \frac{410}{205} = 2.0$$

The maximum stress of the chassis for the given load condition is 155.7 MPa.

$$FOS = (\text{Critical Stress}) / (\text{Allowable Stress})$$

$$FOS = \frac{410}{155.7} = 2.6$$

Hence, the design of the chassis frame is considered adequate (safe) with a Factor of safety of 2.6.

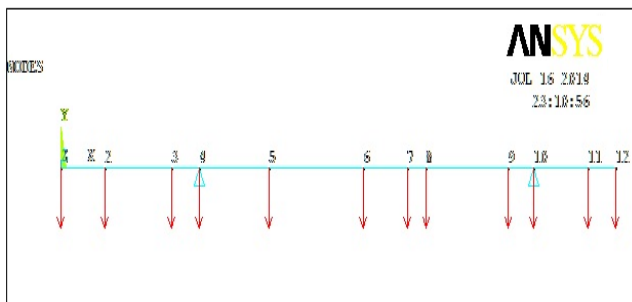


Figure 3.6: Load pattern

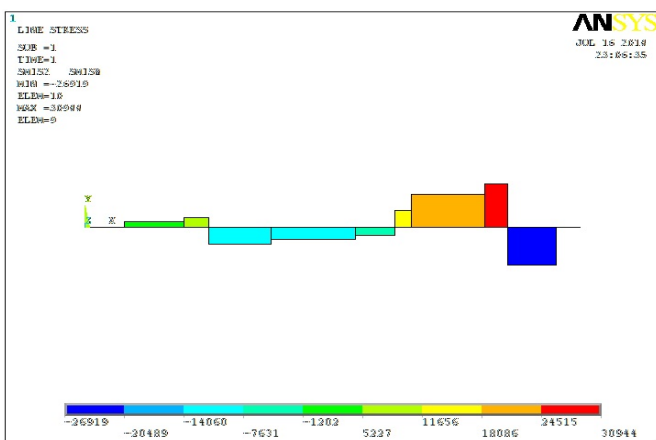


Figure 3.7: Shear Force Diagram, the Max shear force at element 8 is 30944.

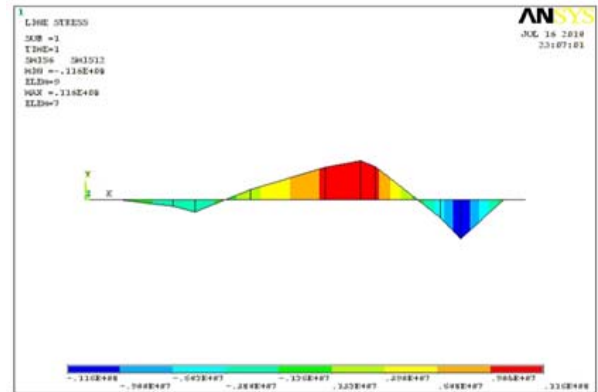


Figure 3.8: Bending Moment Diagram, the Max bending moment at element 6 is 11630.

4. Theoretical Results

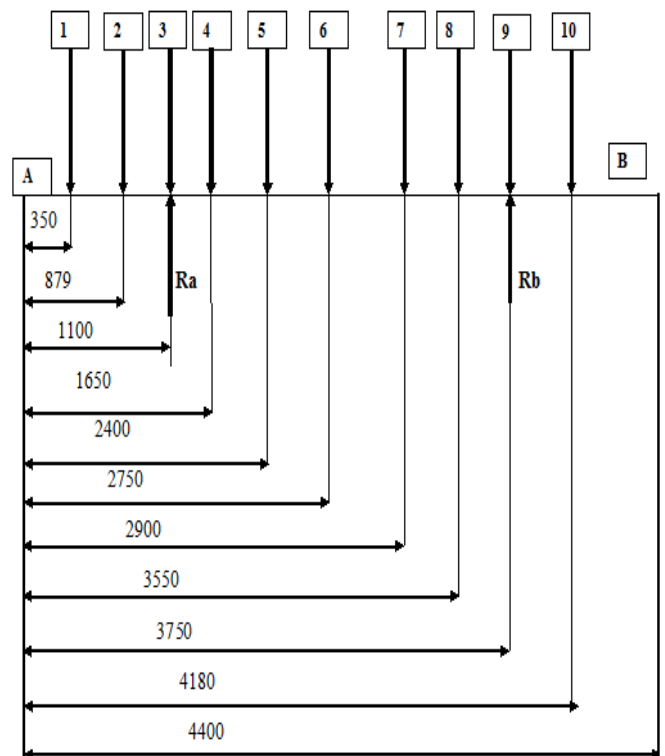
4.1 Load condition

- Load at 1-Cabin Weight-393 kg
- Load at 2-engine weight/2-303 kg
- Load at 3-Weight of Truck distributed to Front axles-4500 kg
- Load at 4-engine weight/2-303 kg
- Load at 5-b-pillar bracket-310 kg
- Load at 6-fuel tank-1835 kg
- Load at 7- c-pillar bracket- 1140kg
- Load at 8- Rear suspension-772 kg
- Load at 9-Weight of Truck distributed to Rear axles-2200kg
- Load at 10-rear frame-2744kg

Constraints:

- Ra-Front axle support.
- Rb-Rear axle support.

4.2 Load Pattern



4.3 SFD AND BMD

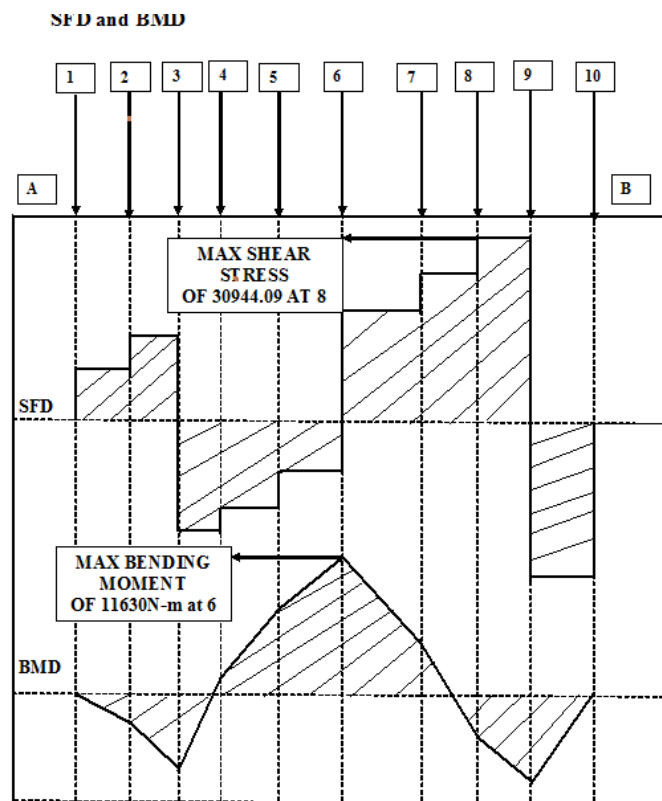


Figure.4.1 Shear force diagram and bending moment diagram for the given load condition, Maximum shear stress is 30944.09 N and maximum bending moment is 11360N-m.

Table 4.1: Validation Results

Forces and Bending moments	FEA results	Theoretical result
Reaction force (Ra)	62800 N	62.80 KN
Reaction force (Rb)	79445 N	79.44573 KN
Maximum shear force	30944 N	30944.09 N
Maximum bending moment	0.116E8 N-mm	11.63E6 N-mm

5.Modal Analysis

Modal analysis is the study of the behavior of a system, which is defined independently from the loads applied to the system, and the response of the system this is usually used to determine the natural frequencies and mode shapes of mechanical structures. The majority of structures can be made to resonate, i.e. to vibrate with excessive oscillatory motion; resonant vibration is mainly caused by an interaction between the inertial and elastic properties of the materials within a structure. Resonance is often the cause of, or at least a contributing factor to many of the vibration and noise related problems that occur in structures. To better understand any structural vibration problem, the resonant frequencies of a structure need to be identified and quantified and the modal analysis has become a widespread means of finding the modes of vibration of a structure.

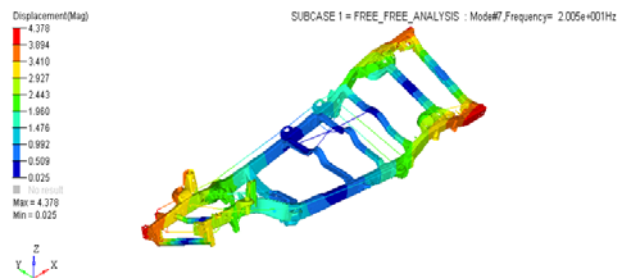


Figure 5.1: First global mode shape

Figure.5.1 First global mode shape of the truck chassis at 20.05Hz. The chassis Experienced twisting mode along z-axis. The maximum translation was at the back end of the truck chassis.

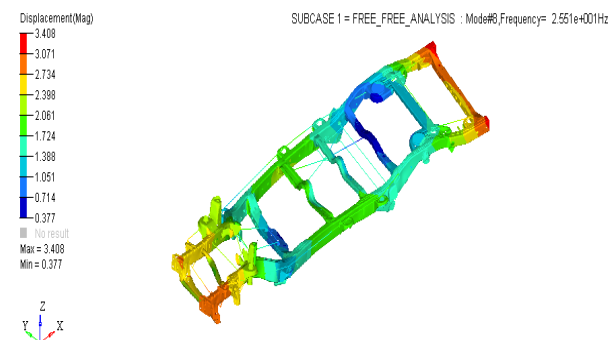


Figure 5.2: Second global mode shape

Figure 5.2 Second global mode shape is shown in Fig.6.9. It was found at 25.51 Hz where the chassis experienced twisting along x direction.

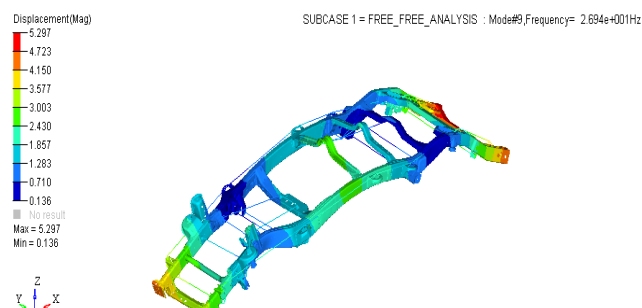


Figure 5.3: Third global mode

Figure.5.3 Third global mode at 26.94Hz, Where the chassis experienced bending at the Front and rear portion of the chassis in z-direction.

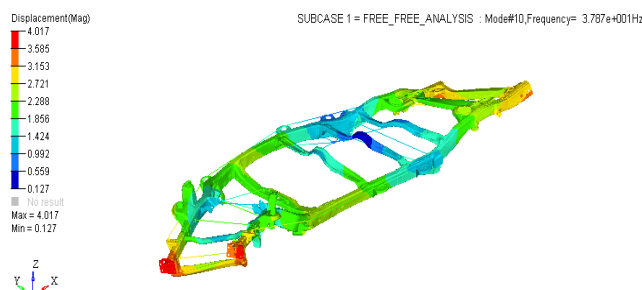


Figure 5.4: Fourth mode

Figure 5.4 Fourth mode at 37.87 Hz, where chassis experienced twisting about lateral direction.

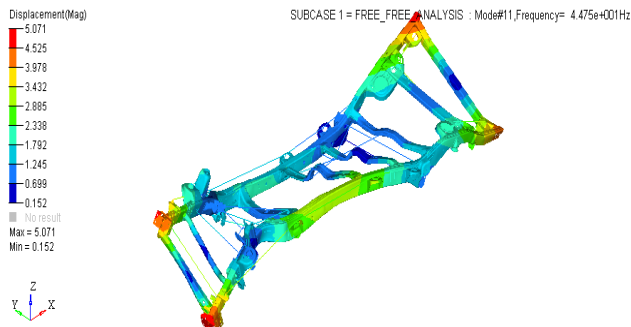


Figure 5.5: Fifth global mode

Figure 5.5 Fifth global mode at 44.75Hz, Where the chassis experienced twisting at the Front, middle and rear portion of the chassis in z-direction

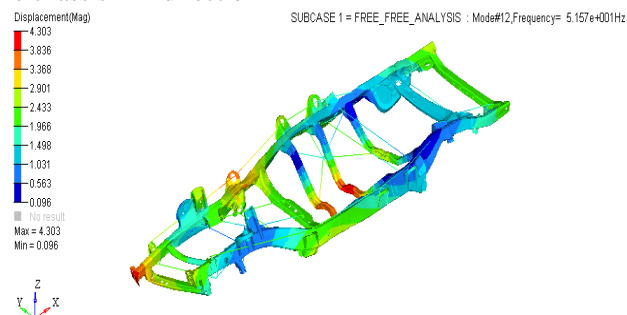


Figure 5.6: Sixth global mode

Figure 5.6 Sixth global mode at 51.57Hz, where the chassis experienced bending at the Middle portion of the chassis in z-direction.

6. Fatigue Analysis

Front-end suspension bracket (FESM) of the chassis subjected to cyclic load during vehicle riding. When riding the fluctuating load leads adverse effect on bracket and it fails withstand the damage strength and life before reaching its yield strength of material. Hence to achieve the bracket must be less damage and more life cycles. In this project for fatigue analysis n-Code software has used.

6.1 Design Modification of FESM Bracket

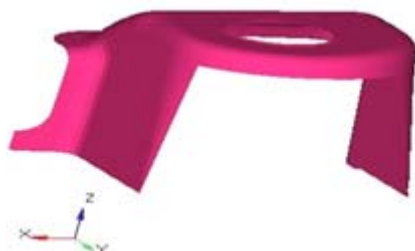


Figure 6.1: Base Design of FESM Bracket

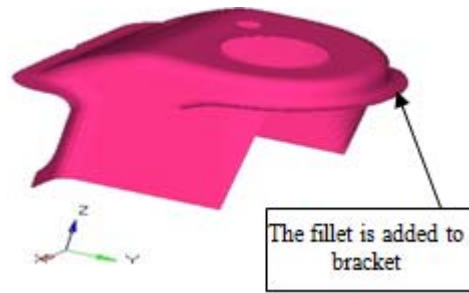


Figure 6.2: Modified Design of FESM Bracket

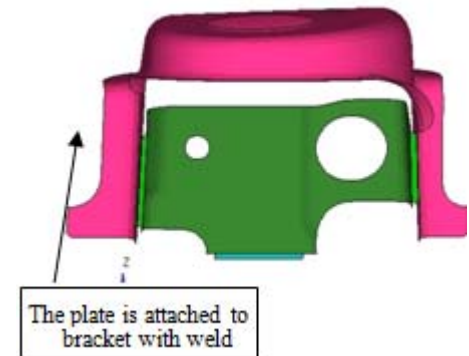


Figure 6.3: Final Design of FESM Bracket

6.2 Boundary and loading conditions for fatigue analysis

For fatigue analysis unit load was applied in each direction (X, Y, Z direction) at both the ends of chassis frame as shown in Fig 6.4. Fatigue analysis was carried out due where the high stress concentration region there will be more crack initiation will occur, it was consider inside of the solid also but always crack was initiated from on the surface. Since first take face of the solid and apply constraints and load. In Hypermesh, deck was prepared with unit load and solved in MSC- Nastran. Finally road load data, material data and Nastran output files like .OP2 and .input files like BDF were input to the n-Code design life software to get damage and life, repeats.

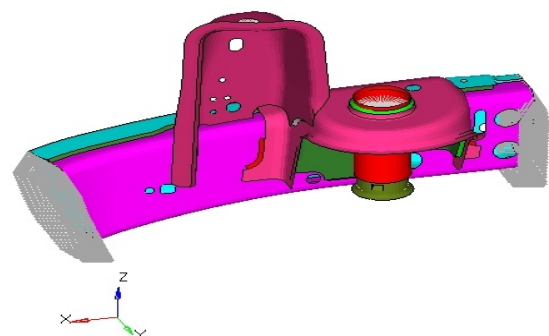


Figure 6.4: Chassis frame is constrained at both the ends

6.3 Area of interest for life calculations

Similar type of design of FESM brackets is considered for both Left and right side of chassis. Hence their needs to be calculate the life calculation on both sides of chassis frame.

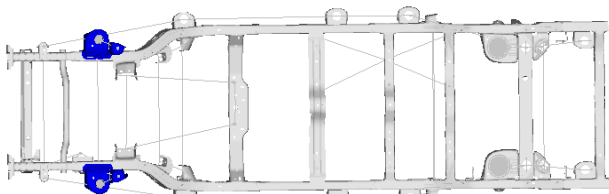


Figure 6.5: Area of interest for life calculations.

6.4 Damage results

Damage of existed FESM bracket

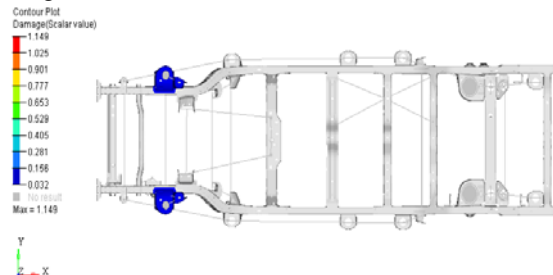


Figure 6.6: Damage of existed FESM mounting bracket

From above plot damage of FESM bracket was found to be 1.149. It is greater than one. Since this bracket is going to fail.

Damage of modified FESM bracket

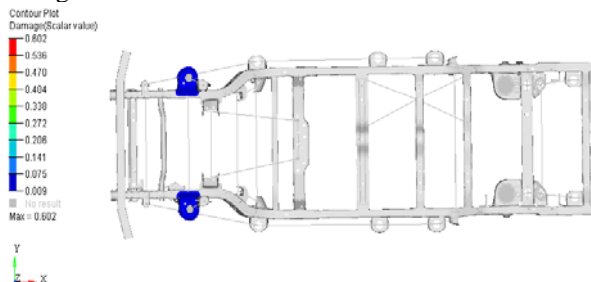


Figure 6.7: Damage of modified FESM mounting bracket

From the Fig 6.7 damage of FESM mounting bracket was found to be 0.602. It is less than one. Since this bracket is safe.

6.5 Life, repeats results:

Life, repeats of existed FESM bracket

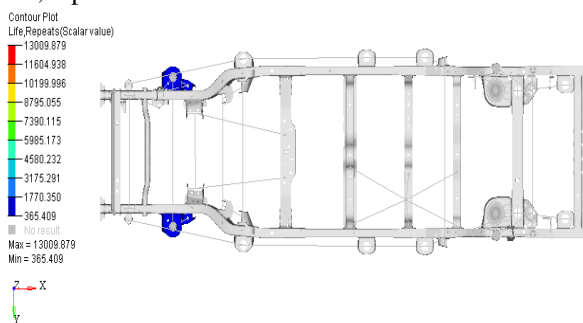


Figure 6.8: Life repeats of existed of FESM mounting bracket

Figure 6.8 life, repeats of the existed FESM mounting bracket were found to be 13009.879 cycles.

Life, repeats of modified FESM bracket.

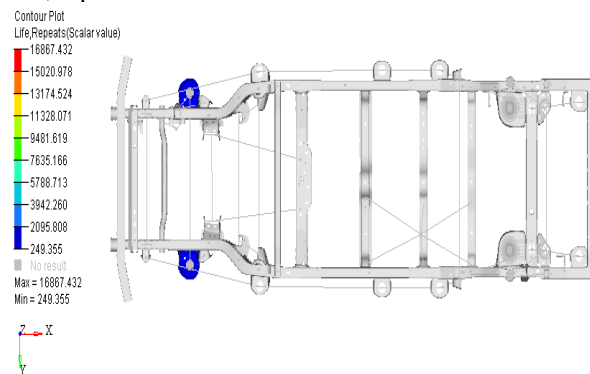


Figure 6.9: Life repeats of modified of FESM mounting bracket

Fig 6.9 life, repeats of modified FESM mounting bracket were found to be 16867.432 cycles.

7. Results and Discussions

- From the static analysis the maximum stress obtained is 155.7 MPa, which is less than the allowable stress of 205 MPa, and static analysis result is validated with the theoretical calculation. And design of the chassis frame is considered with a Factor of safety 2. Table 4.1 shows that shear force and bending moment values with respected diagrams are obtained from FEA analysis are validated with the theoretical calculation.
- From Modal analysis results it is observed that all the natural frequencies of six global modes were below 100 Hz, varying from 20 Hz to 90 Hz. Almost all of the truck chassis designs were based on these frequency ranges to avoid the resonance during the operating condition, Hence the study of Modal behavior of truck chassis had been executed successfully.
- From fatigue analysis, life and damage obtained from existed chassis is 13009.879 cycles, 1.149 and fatigue life and damage obtained from modified chassis is 16857.432 cycles, 0.602 so the Fatigue life of the bracket has been improved and damage reduction had been executed successfully.

8. Conclusion

- From the Modal analysis to predict the dynamic characteristic of truck chassis such as the natural frequency and mode shape. Basically, natural frequency and mode shape are important parameters in design. That is because damage can be occurred if the truck chassis is excited at the natural frequency during operation.
- As the material is ductile, maximum Distortion energy theory is considered for determining the criterion for failure. Taking a factor of safety of 2.0, the allowable stress for the material is 205 MPa, based on Yield strength of 410 MPa. It is concluded that, the maximum stress of 155.7 MPa is well within the allowable limit, hence the design of the chassis frame is considered adequate with a Factor of safety of 2. So we conclude that static analysis result is validated with the theoretical calculation.

- Then reaction forces, shear force and bending moment diagrams are same compared with FEA result so we conclude that static analysis result is validated with the theoretical calculation.
- The fatigue behavior is an important parameter hence the fatigue life of existed FESM bracket of the chassis is 13009.879 cycles has improved to 16857.432cycles and of damage of existed FESM bracket of the chassis is 1.149 it has been reduced to 0.602 hence damage reduction had executed successfully.

References

- [1] Thomas D Gillespie, Fundamentals of Vehicle Dynamics, SAE 1999
- [2] Dave Anderson and Grey Schede, "Development of a Multi- Body Dynamic Modal of a Tractor – Semi trailer for Ride Quality Prediction", International Truck and Engine Corp. 2001.
- [3] Structural analysis of a ladder chassis frame Vijaykumar V. Patel¹ and R. I. Patel²
- [4] Ibrahim I.M.,Crolla D.A and. Barton D.C, "Effect Of Frame Flexibility On The Ride Vibration Of Trucks", Department of Mechanical Engineering, University of Leeds LS2 9JT, U.K. August 1994.
- [5] Izzudin and Abd. Rahman, Roslan, "Application of Dynamic Correlation Technique and Model Updating on Truck Chassis" SAE paper-768665 (2006).
- [6] Murali M.R. Krishna "Chassis Cross-Member Design Using Shape Optimization",\ International Congress and Exposition Detroit, Michigan. February 23-26, 1998.
- [7] "MSC. Nastran for Windows: Quick Start Guide", MSC. Software Corporation, United States of America, 2000.
- [8] J Reimpell & H Stoll, The Automotive Chassis: Engineering Principles, SAE-2000.
- [9] V. Ramamurti, Computer Aided Mechanical Design & Analysis, Tata McGraw Hills-20
- [10] John Fenton, Handbook of Automotive Body Construction & Design Analysis, Professional Engineering Publishing-1998.