STM JOURNALS

International Journal of
Advanced Control and
System Engineering

ISSN: 3049-1452
Volume 3, Issue 1, 2025
January–June
DOI (Journal): 10.37591/IJACSE

https://journals.stmjournals.com/ijacse

Research **IJACSE**

Optimization and Structural Integrity of Truck Chassis Using Finite Element Analysis

Jigar Patel[1], Reena Trivedi[2,*], Bharat Modi[3]

Abstract

The chassis of a truck is a crucial component, and it acts as a framework for the body and other components of the truck. It must also be strong enough to withstand pressures like shock, twisting, vibration, and more. When building a chassis, having enough bending stiffness for better handling characteristics is just as important as strength. Deflection, maximum stress, and maximum equilateral stress are crucial design factors for the chassis. This project focuses on the efforts made to improve the vehicle chassis. To determine stiffness and determine how to minimize weight while increasing stiffness, sub-modeling and static analysis is performed. The sub-model's boundary represents a section extracted from a larger, less detailed version of the complete model, known as the coarse model. The boundary conditions, such as the movement or displacement of structural parts, are derived from the displacement values calculated in the coarse model. This approach allows the sub-model to analyze a specific, more refined area while maintaining its connection to the overall structure's behavior. Therefore, using real parameters, a correct finite element model of a truck chassis must be created. Applying appropriate loading and boundary constraints to the truck chassis model. For finite element analysis, pre-processing is done on the Hypermesh 21.1, and analysis is carried out in OptiStruct package.

Keywords: Finite element analysis, sub-modeling, truck chassis, optimization, OptiStruct analysis

INTRODUCTION

To ensure that heavy-duty vehicles remain safe, long-lasting, and operate effectively, a truck's chassis strength and performance are crucial. The chassis must withstand a lot of stress when the truck is moving, including twisting, bending, and continuous vibrations, because it is the primary component that supports the weight of the entire vehicle. The primary support component of the car is the chassis, sometimes referred to as the “frame.” It sustains all the vehicle's stress under both static and dynamic circumstances. It is comparable to the skeleton of a living being in a vehicle. The French language is where the word “chassis” first appeared. Every vehicle, including cars, trucks, and two-wheelers, have a chassis and frame. But depending on the type of vehicle, its shape fluctuates [1, 2]. The following are the functions of the chassis:

1. Supports or bears the vehicle's body weight.
2. Provides room and a mounting point for different vehicle aggregates.
3. Supports the weight of the car's numerous systems, including the engine, gearbox, etc.
4. Supports both the luggage and a load of passengers.
5. Is able to withstand the strains brought on by poor road conditions.
6. Is able to withstand strains as the vehicle accelerates and brakes.

***Author for Correspondence**
Reena Trivedi
E-mail: reena.trivedi@nirmauni.ac.in

[1]Student, Department of Mechanical Engineering, Institute of Technology, Nirma University, Ahmedabad, Gujarat, India
[2]Associate Professor, Department of Mechanical Engineering, Institute of Technology, Nirma University, Ahmedabad, Gujarat, India
[3]Professor, Department of Mechanical Engineering, Institute of Technology, Nirma University, Ahmedabad, Gujarat, India

Received Date: February 01, 2025
Accepted Date: February 15, 2025
Published Date: February 25, 2025

Citation: Jigar Patel, Reena Trivedi, Bharat Modi. Optimization and Structural Integrity of Truck Chassis Using Finite Element Analysis. International Journal of Advanced Control and System Engineering. 2025; 3(1): 31–37p.

© STM Journals 2025. All Rights Reserved

INTRODUCTION TO SUB-MODELING

A finite element method called sub-modeling is used to improve the accuracy of a particular area of a model. The cut-boundary displacement method and the defined boundary displacement method are other names for sub-modeling. The sub-model's boundary, which symbolizes a cut through the coarse model, is known as the cut boundary. Boundary conditions for the sub model are displacements computed on the coarse model's cut boundary. The foundation of sub-modeling is St. Venant's principle, which asserts that the distribution of stress and strain is only changed in the vicinity of the load application regions if an actual force distribution is substituted with a structurally equivalent system. This suggests that the consequences of stress concentration are localized around the concentration; as a result, reasonably accurate results can be estimated in the sub model if the borders are sufficiently distant from stress concentration. A sub-model example is shown in Figure 1.

Steps Involved for Optimization

Figure 2 shows the methodology flowchart for optimization.

- For the first step, we have to perform baseline run with the full assembly and load cases.
- After that, sub-model run is performed forces and moments applied at the end nodes and validating with the global model results.
- Perform the analysis of sub-model with loading and boundary conditions.
- Optimization run is performed and get the optimized design.

LITERATURE REVIEWED

Yilmazcoban and Kahraman built a chassis of a car using CATIA V5 software [1]. Meshing of the model was carried out in the Hypermesh software where the element type was set to be SHELL63. A total of 5793 elements were created with the node count of 5932. Material used was steel.

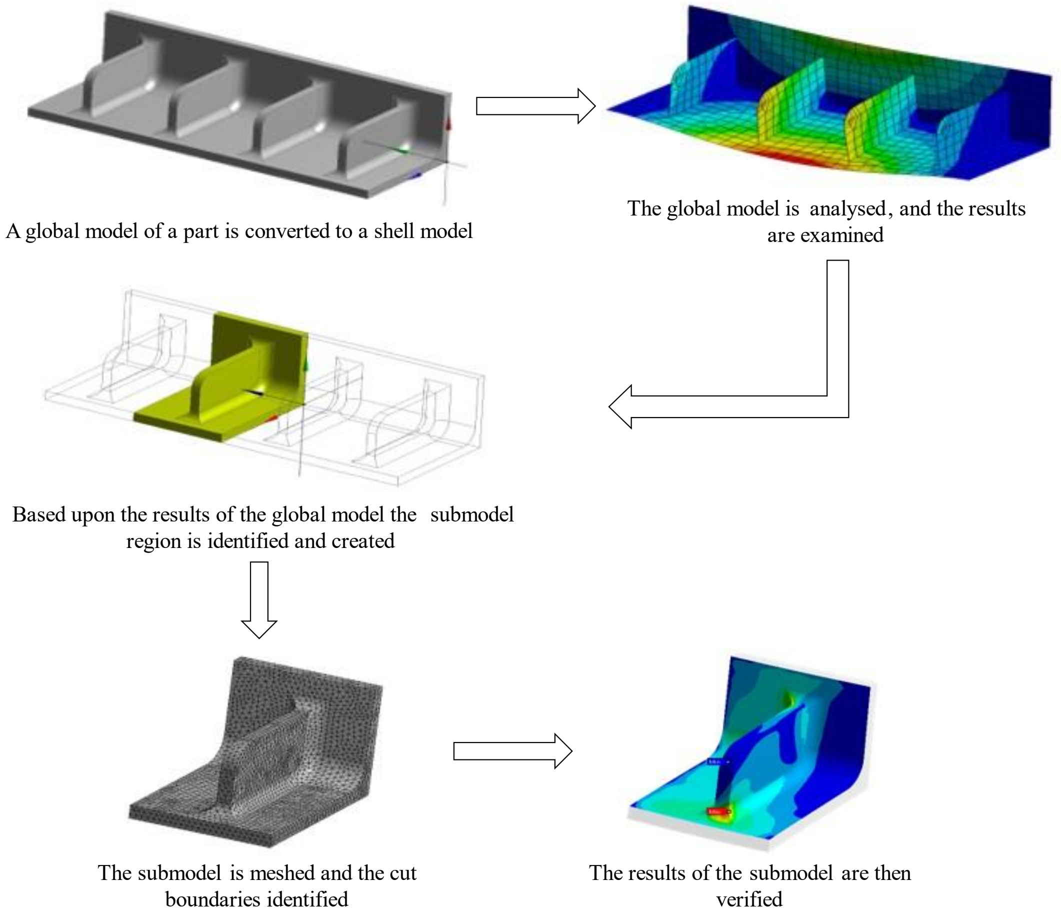

Figure 1. Sub-model example [1].

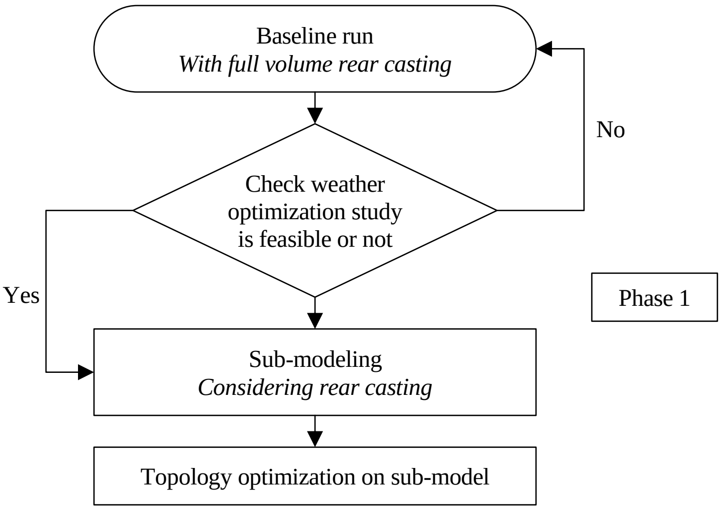

Figure 2. Methodology flowchart for optimization.

They have performed self-weight analysis, bending analysis, and torsion analysis. Several sub-assemblies, such as the powertrain, nuts, bolts, and name plate, were not taken into consideration for the research because they do not contribute anything of importance to the study.

Mathivanan et al. used a 2019 Chevrolet Silverado pickup truck for their investigation [3]. They imported the model from the ANSYS software. They tested the crash analysis for different sections and different materials. Following the comparison, they concluded that, under static crash analysis, the channel frame portion is safer and lighter under ideal load and speed conditions.

Agrawal and Razik used TATA 1612 chassis to calculate the stress of given boundary and loading conditions. They have calculated the pressure acting on the parts manually [4].

Solid model of chassis is made through CATIA software and the material used is mild steel which contains 0.05% to 0.3% of carbon. First, they performed the static analysis and checked the total deformation and directional deformation. Modal analysis was performed and they checked different mode shapes and regarding that they have changed the thickness of the C-section wherever less load was acting and where there was less deformation [4].

Vijayakumar et al. performed the stress analysis of the static force loaded heavy duty truck chassis [5]. They prepared the TATA 407 fire truck model in the PRO-E software. The material used was steel and carbon fiber [5]. ANSYS workbench was used to mesh the model and for the analysis to check the equivalent stress, extreme elastic strain and whole deformation. Modal analysis was also performed for both the materials from that they have concluded that carbon fiber material can reduce 80% of the weight.

Kurdi et al. used Hino model truck chassis [6]. ASTM low alloy steel A 710 C was used to make the chassis. There are 37697 nodes and 101466 elements in the meshed truck chassis model [6]. The element type was 3D stress, and its shape was tetrahedral. The chassis aperture that meets the bolt is where the Von Mises stress is highest.

The stress analysis of a truck chassis with riveted joints was examined by Karaoğlu and Kuralay using finite element method [7]. The side member thickness can be increased locally to alleviate side member strains, according to numerical results. Increasing the length of the connection plate could be a useful substitute if changing the thickness is not feasible.

The goal of authors is to assess a car's chassis for a 10-ton vehicle. Pro-E is used for modeling, and ANSYS is used for analysis [8]. Analytical calculations of the chassis' overhangs for stresses and deflections are made, and the findings are compared to those produced by the analysis program. The natural frequency of the chassis is also determined using modal analysis, which shows that it is higher than the excitation frequency. When the results of finite element analysis and theoretical calculations are compared, it is found that they fall within the parameters of the material [3].

Mathivanan et al. [3] has analyzed the lightweight chassis for different composite materials. Where they have used steel, E-glass and aluminum alloy materials. The behavior for steel and aluminum alloy is Isotropic while E-glass is orthotropic. After analyzing the results, they have concluded E-glass is the best suitable for the chassis material as it has less weight but strength is very high as compared to other two materials [3].

To reduce the weight of the TATA 2516TC chassis frame, Patel et al. investigated and improved a chassis design using Pro-Mechanical [9]. Using ANSYS software, they first determined the assembly weight, maximum stress, strain, and displacement for the existing chassis sections (C, I, and box sections). They then adjusted the dimensions of the existing C-sections and repeated the process, concluding that the existing “C” sections outperform all other sections in terms of stress, displacement, strain, and shear stress, except for weight. The redesigned “C” section has a lower weight than all the other parts that are being examined in this research. Lastly, 105.50 kg (11%) less weight is saved per chassis assembly by using the redesigned “C” section [10, 11].

DESIGN OF CHASSIS

The chassis in use here comes from a combine harvester. The most advantageous multi-crop harvester in its class is the new Ideal for harvesting minor crops like wheat or barley, specialized crops like rice or sunflower, or huge grains like corn, soybeans, and a variety of pulse crops. Figure 3 shows the full assembly of the feeder.

The chassis which we have used is consist of many sub-assemblies named as adapter, cradle, housing, etc. The whole feeder assembly is about 1500 kg in weight. It consists of 180 kg of cradle sub-assembly. Figures 4 and 5 shows the housing and sub-model of chassis, respectively. Cradle is the front portion of the feeder where the crop is collected. After that there is another sub-assembly called adapter.

TYPES OF MESH USED FOR FINITE ELEMENT MODELLING

For casting we have used first order hex mesh. CHEXA elements are used to model this casting. More complex parts can be meshed with the second order tetra element. Here we have meshed the gear box and some clamps with second order tetra element. The sub-model of the chassis is shown in Figure 5. The second order tetra element is CTETRA. Sheet metal parts or the parts where the thickness is constant is modeled with 2D Quad elements. Here, the OptiStruct quad elements is named as CQUAD4. It utilizes five integration points.

CONNECTORS

In some cases, the weld region is modeled with node-to-node connection and in some cases, it is modeled with 1D rigids.

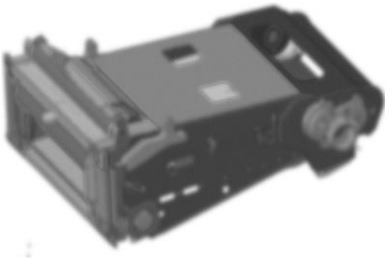

Figure 3. Feeder assembly.

Figure 4. Housing sub-assembly of chassis.

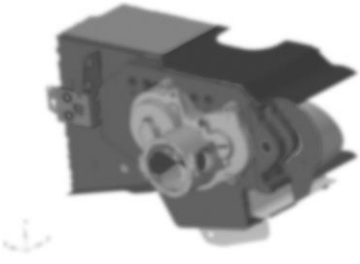

Figure 5. Sub-model of chassis.

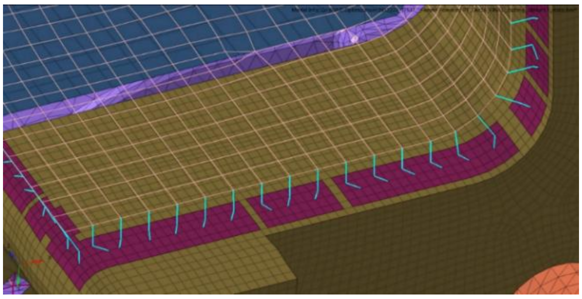

Figure 6. Weld connections.

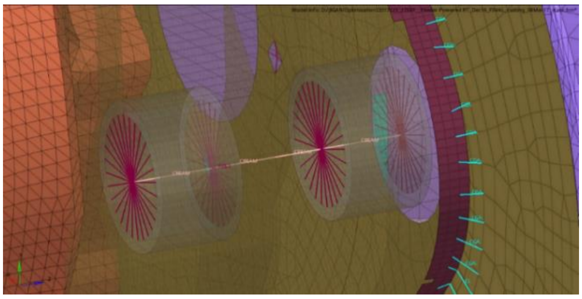

Figure 7. Bolt connections for load transfer.

Figure 6 shows the weld connections between two parts and Figure 7 shows the bolt connections for load transfer. When a hard link is sought between two nodes in a model, an element formed in that space is called a rigid element. The letter R is printed at the centroid of rigid elements, which are element config 5 and appear as a line between two nodes. Also, the front portion of the feeder assembly is modeled with the CBEAM element. We must directly transfer the load to the cradle and adapter sub-assembly through this front feeder head.

MATERIAL USED

The material used for the construction of the chassis is usually carbon steel and aluminum. Steel found its way to trucks that demands high durability and reliability. Table 1 presents the mechanical properties of steel and aluminum alloy.

BOUNDARY AND LOADING CONDITIONS

Figure 8 shows the boundary conditions applied on the chassis. Boundary condition is basically to defined the constraints of the degrees of freedom (DOFs). Here, on the bottom left side, it is fixed by 1, 2, and 3 it means it does not translate in X, Y, and Z directions. Whereas on the bottom right side it is fixed in 1 and 2 only, means it can translate in Z direction and also on both side it can rotate in X, Y, and Z direction. It is secured on 1, 2, and 3 on the upper side. All the boundary conditions are defined where the housing is mounted on the full assembly.

The chassis has two types of loading conditions. One is 2G force in downward direction, second is gear box torque of 1222 Nm in +Z direction is applied. Figures 9 and 10 show the loading conditions applied on the feeder assembly.

Design Parameter Selection for Optimization

Optimization analysis is carried out by assigning the design and non-design portion to the chassis as shown in Figure 11. Also assigning the stress constraint to the design portion of the chassis. Configuration is set as topology. The objective is to minimize the volume. Non-design space is selected near the bolting locations, all other locations are selected as the design space.

RESULTS AND DISCUSSION

Sub-modeling methodology is used for quick results of optimization. The procedure described next will be followed for preparing the sub-model

1. Considering the rear casting along with gear box and their nearby connection region for sub-model.
2. After that, we extracted the free body diagram (FBD) forces from the linear static analysis, and applied the same load on Sub model.
3. Static results of the sub-model match with the complete model for all the load cases.

Table 1. Mechanical properties of steel and aluminum alloy.

Material	Mechanical Properties		
	Young's Modulus (GPa)	Poisson's Ratio	Density (kg/m^3)
Steel	210	0.3	7900
Aluminum alloy	703	0.34	2660

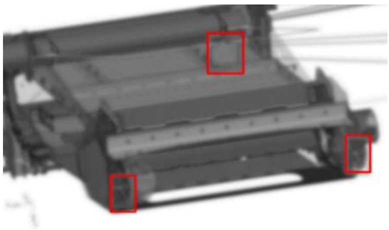

Figure 8. Boundary conditions.

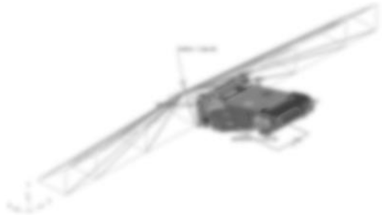

Figure 9. 2G force downward direction.

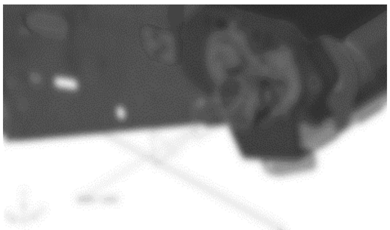

Figure 10. Gear box torque.

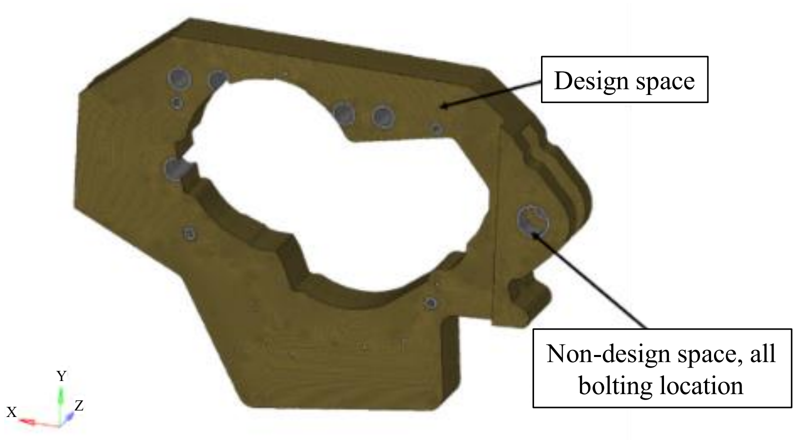

Figure 11. Design parameter for optimization**.**

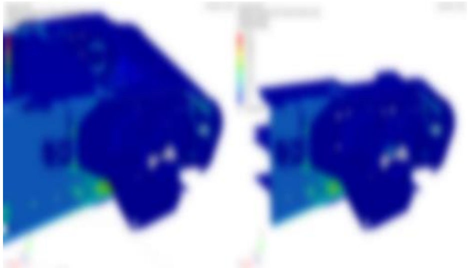

Figure 12. 2G force downward direction.

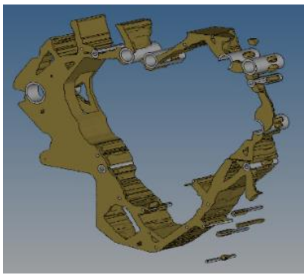

Figure 13. Optimized raw shape.

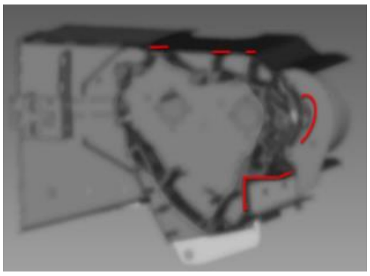

Figure 14. Optimized shape on sub-model of feeder assembly.

Figure 12 shows the sub-model validation with global model. After the sub-modeling, optimization run is done. Where we got the results as per the input, we have given minimize volume. Here element density is set to be as 0.015. and we got the rough design of optimized casting. Figure 13 shows the raw optimized shape of casted part and Figure 14 shows the optimized shape on the sub-model assembly.

CONCLUSION

The sub-model result is verified with the global model results, and it is nearly equal to the global model results. The maximum von Mises stress is around 400 MPa which is crossing the limit of 340 MPa. But we can neglect that stress value because it is stress singularity and it is on the weld region by proper mesh style.

As per optimization result, it is recommended, marking red line location need to do the weld with plate, rest of the location bolt connects are enough. Currently, the optimized shape, we have got is in raw shape. Total wight of the casting component before optimization is around 250 kg and after optimization it is around 200 kg. So, we reduced the weight around 17% without compromising the stiffness.

REFERENCES

1. Yilmazcoban IK, Kahraman Y. Truck chassis structural thickness optimization with the help of finite element technique. Online J Sci Technol. 2011; 1 (3): 1–30.
2. Yasar A, Bircan DA. Design, analysis and optimization of heavy vehicle chassis using finite element analysis. Int J Sci Technol Res. 2015; 1 (6): 1–9.
3. Mathivanan S, Mugesh MK, Praveenkumar K. Analysis and optimization of truck chassis. Int J Eng Res Technol. 2019; 7 (6): 1–4.
4. Agrawal MS, Razik MD. Finite element analysis of truck chassis. Int J Eng Sci Res Technol. 2013; 2 (12): 3432–3438.
5. Vijayakumar MD, Kannan CR, Manivannan S, Vairamuthu J, Tilahun S, Ram PB. Finite element analysis of automotive truck chassis. IOP Conf Ser Mater Sci Eng. 2020; 988 (1): 012114.
6. Kurdi O, Abd-Rahman R, Tamin MN. Stress analysis of heavy-duty truck chassis using finite element method. In: 2nd Regional Conference on Vehicle Engineering & Technology, Kuala Lumpur, Malaysia, July 15–17, 2008. pp. 1–6.
7. Karaoğlu C, Kuralay NS. Stress analysis of a truck chassis with riveted joints. Finite Elements Anal Design. 2002; 38 (12): 1115–1130.
8. Kumar AH, Deepanjali V. Design & analysis of automobile chassis. Int J Eng Sci Innov Technol. 2016; 5 (1): 187–196.
9. Patel AS, Srivastava A. Modeling, analysis & optimization of TATA 2518 TC truck chassis frame using CAE tools. Int J Eng Res Technol. 2016;5(10):70–80.
10. Rajan SD. Sizing, shape, and topology design optimization of trusses using genetic algorithm. J Struct Eng. 1995; 121 (10): 1480–1487.
11. Ary AK, Prabowo AR, Imaduddin F. Structural assessment of an energy-efficient urban vehicle chassis using finite element analysis – a case study. Procedia Struct Integrity. 2020; 27: 69–76.