Modern Mechanical Engineering, 2023, 13, 55-61
https://www.scirp.org/journal/mme
ISSN Online: 2164-0181
ISSN Print: 2164-0165

A Comparative Study on the Truck Frame Stiffness with Solid and Beam Element FEA Models

Shengyong Zhang*, Zack Gertzen, Adam Manering, Cristian Jongkind

Department of Mechanical and Civil Engineering, College of Engineering and Sciences, Purdue University Northwest, Westville, USA
Email: *syzhang@pnw.edu

How to cite this paper: Zhang, S.Y., Gertzen, Z., Manering, A. and Jongkind, C. (2023) A Comparative Study on the Truck Frame Stiffness with Solid and Beam Element FEA Models. *Modern Mechanical Engineering*, **13**, 55-61.
https://doi.org/10.4236/mme.2023.133004

Received: May 27, 2023
Accepted: July 25, 2023
Published: July 28, 2023

Abstract

Truck frames should be designed and fabricated with enough rigidity to avoid excessive deflections. Finite element analysis (FEA) plays an important role in all stages of frame designs. While being accurate, 3D solid element FEA models are built upon frame configuration details which are not feasible in the preliminary design stage, partially because of limited available design data of frames and heavy computation costs. This research develops 1D beam element FEA models for simulating frame structures. In this paper, the CAD model of a truck frame is first created. The solid element FEA analysis, which is adopted as the baseline in this study, is subsequently conducted for the stiffness of the frame, Next, beam element FEA analysis is performed for validating the feasibility of the beam element FEA model by comparing the results from the solid and beam element FEA models. It is found that the beam element FEA model can predict the frame stiffness with acceptable accuracy and reduce the computation cost significantly.

Keywords

FEA, Truck Frame, Structural Stiffness, 3D Solid Element, 1D Beam Element

1. Introduction

Most trucks use ladder-type frames with two side members and multiple cross members. One main function of truck frames is to sustain both static and dynamic loads: the side members support vertical and side loads such as engine, transmission, fuel tank, battery box, suspensions, body and cargo. The cross members provide torsional rigidity and reduce the degree of side member twisting. All frames should be designed and fabricated with enough rigidity to avoid excessive deflections.

FEA plays an important role in all stages of frame designs and drives the design path at a much faster pace by evaluating the behaviors of a frame under operating conditions and identifying the potential failures early in the preliminary design stages [1]. FEA static structural analysis was conducted for an idealized ladder type chassis with two side members and a few cross members [2]. Conventional structural steels and aluminum alloys were compared for lightweight design of the chassis. It was concluded that material substitution reduced the chassis weight by 68%. The effects of material replacement, member thickness change and member cross section change on the responses of a chassis using FEA analysis were studied [3]. It was concluded that the 8 mm thick Advanced High Strength Steel (AHSS) chassis was better than the original 5 mm thick steel chassis, and T section was preferred to box section from weight reduction point of view. Shape and size optimization of side rails of a large hybrid truck chassis were performed for weight reduction [4]. The front and rear suspensions and relevant truck components were integrated in the chassis FEA models. Vibration mode analysis and stress analysis for multiple load cases were performed on the entire model to evaluate the constraints in the optimization problem. It was concluded that a mass reduction of about 13.25% with respect to the baseline model was achieved. A shell element FEA model was developed for analyzing a truck chassis [5]. Static responses of the chassis under different loads were conducted from stress and deflection analysis. Vibrational response was carried out as well for natural frequencies and vibration modes. It was concluded that both deformation and stresses were within the allowable limits of the chassis materials. FEA analysis was performed to compare the three different materials (grey cast iron, AISI 4130 steel and ASTM A710 steel) in their applications of a heavy vehicle chassis [6]. It was concluded that AISI 4130 steel was superior to structure steel based on stress and stiffness analysis. Different cross-sectional shapes were compared as well in this study. Shell element FEA models were used for investigating the potential lightweight design of a truck chassis [7]. Topological optimization was used in this study under different loading cases. It was concluded that a 20.97% weight reduction was achieved without compromising structural strength.

Solid finite elements are frequently used by automatic mesh generators in FEA software. While being accurate, solid element FEA models are built upon frame configuration details which are not feasible in the preliminary design stage, partially because of limited available design data of frames. In addition, solid elements have at least four or eight nodes for each tetrahedral or hexahedral element, resulting in an extremely heavy computation cost. This research develops beam element FEA models for simulating frame structures. Compared to solid element FEA models, beam element FEA models are based on beam elements which have two or three nodes for each element, leading to a significant reduction in the FEA model size and computation time. As a result, application of beam element FEA models will accelerate frame design process through optimum iteration. In this paper, the CAD model of a truck frame is first created. FEA analysis based on solid elements is subsequently conducted for the stiffness of the frame, which is adopted as the baseline in this study. Next, beam element FEA analysis is performed for validating the feasibility of the beam element FEA model by comparing the results from the solid and beam element FEA models. Finally, conclusions are drawn.

2. Frame CAD Model

Figure 1 shows the frame used in this study, which includes mounting brackets, welds, attached nuts, etc. Unlike stress analysis, which is a function of local geometry, stiffness analysis has neglected relevance to these detailed small parts. Removal of these small parts will simplify the CAD modelling without sacrificing the accuracy of the following stiffness analysis. In addition, a continuous domain is decomposed into discreated elements and nodes in the FEA analysis. Even though the decomposition process is performed by the computer automatically, manual intervention is needed to generate a good mesh. The small parts in the frame shown in Figure 1 are features related to manufacturing and assembly, but extraneous in the FEA simulation. These insignificant features affect the FEA model complexity adversely in terms of model sizes, the efficiency of modeling processes, and the accuracy of simulation results. For example, a fillet can increase the number of elements and nodes greatly. Therefore, it is desirable to examine the small features in the frame model to see if the model should be simplified before it can be adopted as the computational domain in the simulation. In this study, the above-mentioned features have been suppressed and cleaned up for creation of a mesh solvable within a reasonable time. The cleanup frame CAD model is shown in Figure 2.

3. 3D Solid Element FEA Analysis for Frame Stiffness

The frame CAD model shown in Figure 2 is imported in ANSYS for stiffness analysis. The side members and cross members are meshed with solid elements (either hexahedral or tetrahedral elements) as shown in Figure 3. Each hexahedral solid element has eight nodes, and each tetrahedral solid element has four nodes. In the stiffness analysis, the frame is sufficiently constrained at the four locations where the suspensions are mounted. Two concentrated forces with the same magnitude of 1700 N are applied downward at the middle points of the two side members. The frame stiffness is defined to be the quotient of the applied net force and the corresponding average deflection at the acting points, which is 1328 kN/m from the FEA analysis. Figure 4 shows the deformation of the frame under the applied forces.

4. 1D Beam Element FEA Analysis for Frame Stiffness

FEA beam elements are suitable for simulating structures composed of slender members, such as the frame structure in this study. Beam elements can resist both forces (lateral and axial) and moments (twisting and bending) which is also in line with the loading conditions in this study. In addition, compared to solid elements which have at least four or eight nodes for each tetrahedral or hexahedral element, a beam element has two or three nodes, resulting in a significant reduction in model size and computation cost.

Figure 1. Original truck frame configurations.

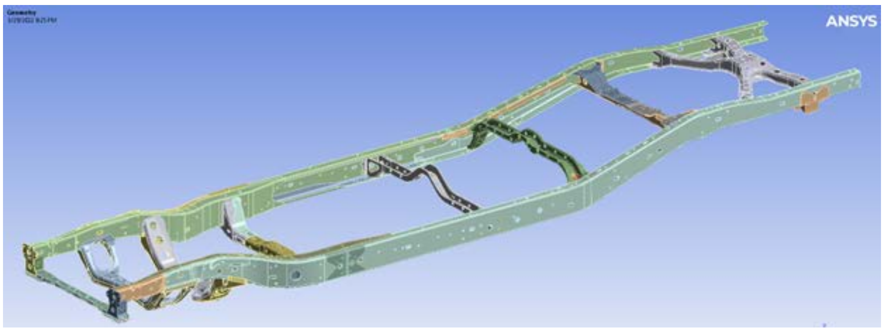


Figure 2. Frame CAD model after cleanup of insignificant features.

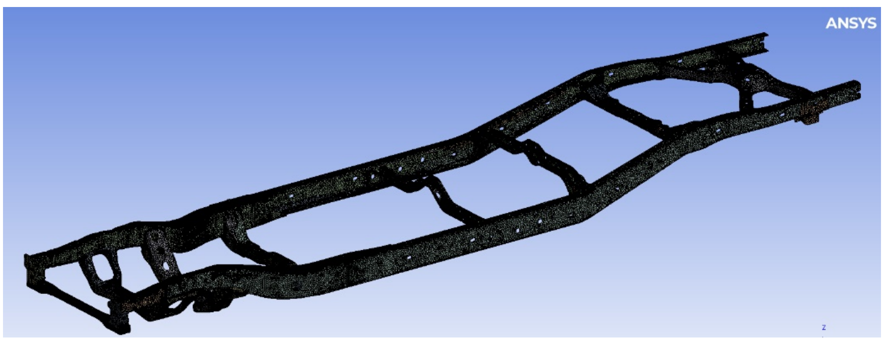
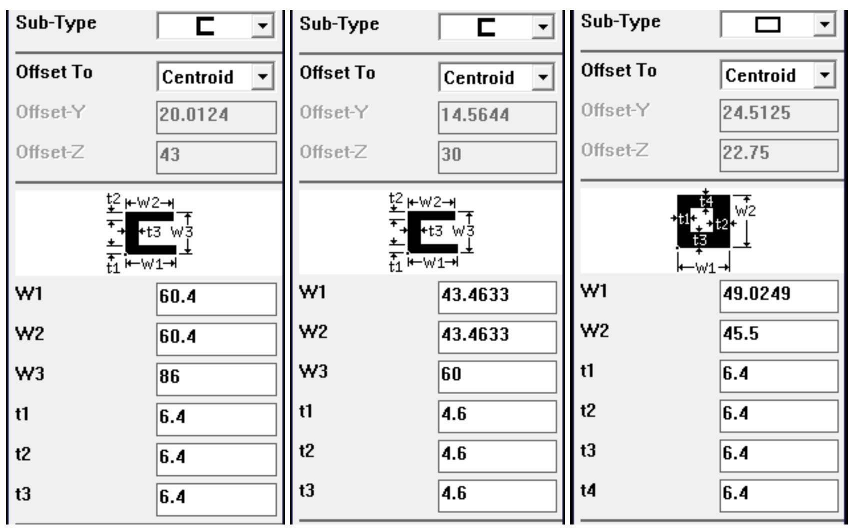


Figure 3. Solid element FEA model for stiffness analysis.

Figure 4. Frame deflection from the solid element FEA analysis.

The beam element FEA model of the frame is built in the ANSYS APDL module. Keypoints are first specified for all frame members according to their configurations, followed by creation of lines connecting each pair of adjacent keypoints. The side members and cross members have different cross-sectional shapes and dimensions which can be obtained from its CAD model. As an example, some cross-sectional properties are defined in Figure 5: each side member has a C-shaped cross section with the dimensions shown in Figure 5(a), the cross-sectional properties of the two X-shaped cross members are shown in Figure 5(b), and so forth. The material properties in this study are as following: Young's modulus E = 200 GPa, Poisson's ratio ν = 0.3, yielding stress σ_y = 250 MPa, density ρ = 7850 kg/m^3 and ultimate stress σ_u = 450 MPa.

Figure 6 shows the FEA model of the frame meshed with quadratic three-node beam elements, BEAM189, in ANSYS. Again, two concentrated forces with the same magnitude of 1700 N are applied downward at the middle points of the two side members. The frame stiffness is calculated to be 1141 kN/m. Figure 7 shows the deformation of the frame under the applied forces.



(a) (b) (c)

Figure 5. Cross-sectional shapes and dimensions of (a) the side member, (b) X-shaped member, and (c) member next to the X-shaped member.

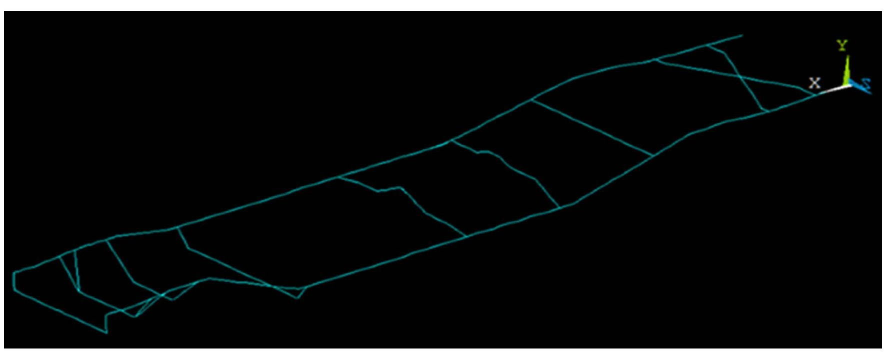


Figure 6. Beam element FEA model for stiffness analysis.

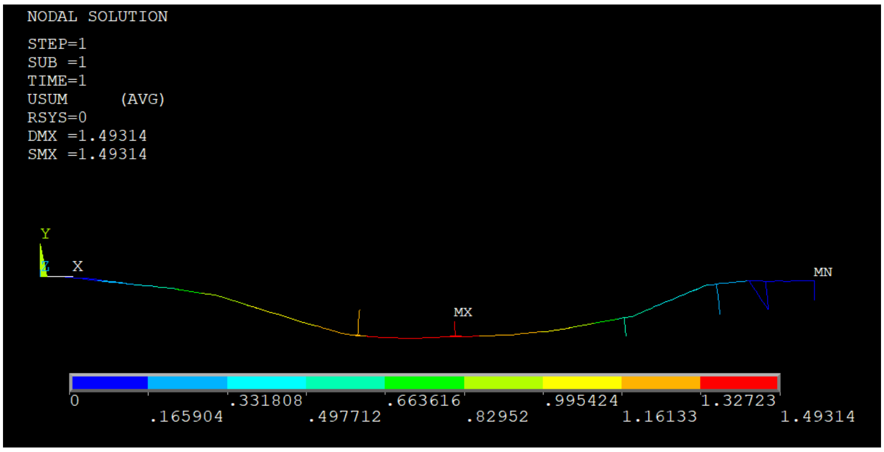


Figure 7. Frame deflection from beam element FEA analysis.

5. Discussion and Conclusion

Although FEA can cope with an object with an arbitrary geometric shape technically, it is a strategy in the FEA modelling to suppress the insignificant features over the object that affect the FEA model complexity adversely in terms of model size and computation cost. A comparative study has been conducted for the stiffness of a truck frame using 3D solid and 1D beam element FEA models. The insignificant features (mounting brackets, welds and nuts for manufacturing and assembly, for example) of the frame have been cleaned up in the CAD model and subsequent solid element FEA analysis. For comparative study, a beam element FEA model is created based upon the frame architecture for validating its feasibility of predicting the frame stiffness. It is concluded that the beam element FEA model can predict the frame stiffness with acceptable accuracy while reducing the computation cost significantly.

Conflicts of Interest

The authors declare no conflicts of interest regarding the publication of this paper.

References

[1] Bi, Z. (2018) Finite Element Analysis Applications: A Systematic and Practical Approach. 1st Edition, Academic Press, Cambridge, Massachusetts. https://doi.org/10.1016/B978-0-12-809952-0.00001-7

[2] Agarwal, A. and Mthembu, L. (2022) Structural Analysis and Optimization of Heavy Vehicle Chassis Using Aluminium P100/6061 Al and Al GA 7-230 MMC. *Processes*, **10**, 320. https://doi.org/10.3390/pr10020320

[3] Garud, R.Y., Tamboli, S.C. and Pandey, A. (2018) Structural Analysis of Automotive Chassis, Design Modification and Optimization. *International Journal of Applied Engineering Research*, **13**, 9887-9892.

[4] De, S., Singh, K., Seo, J., Kapania, R.K., Ostergaard, E., Angelini, N. and Aguero, R. (2021) Lightweight Chassis Design of Hybrid Trucks Considering Multiple Road Conditions and Constraints. *World Electric Vehicle Journal*, **12**, 3.

https://doi.org/10.3390/wevj12010003

[5] Naveen, G. and Prashanth, A.S. (2021) Design and Structural Analysis of Truck Chassis. *International Research Journal of Engineering and Technology*, **8**, 2612-2624.

[6] Sharma, A., Kumar, P., Jabbar, A. and Khan, M.M. (2014) Structural Analysis of a Heavy Vehicle Chassis Made of Different Alloys by Different Cross Sections. *International Journal of Engineering Research & Technology*, **3**, 1778-1785.

[7] Bhoyate, S. and Patel, A.Z. (2020) Weight Optimization and FEA Analysis of Truck Chassis. *International Research Journal of Engineering and Technology*, **7**, 813-822.