ASEE 2022 ANNUAL CONFERENCE
Excellence Through Diversity
MINNEAPOLIS, MINNESOTA, JUNE 26TH-29TH, 2022 ASEE

Paper ID #36700

Multi-Material Optimization of a Simplified Railcar Truck Stand

Raghu Echempati (Professor)

Raghu Echempati is a senior professor of Mechanical Engineering at Kettering University, Flint, MI with expertise in Mechanisms Design, Applied FEA, Mechanical Engineering Design, Metal Forming Simulation and Automotive Lightweighting and Joining Technologies. He has over 3 decades of academic teaching, industrial consulting and applied research in the areas of expertise mentioned earlier. He worked as a faculty intern at Bosch, General Motors and GEMA (Chrysler Div). He established several study abroad programs in Germany and Australia and taught at HTWG, Konstanz, and Brazil. He was a Fulbright scholar to teach in Thailand at KMUTT (Bangkok) and at IIT Delhi (India). He also received Erskine Scholar Fellowship to teach in New Zealand. He gives keynote and invited talks at several international universities. He has supervised around 200 bachelor and master student theses. He has published over 180 technical articles, journal and conference papers of repute.

Multi-Material (MML) Optimization of a Simplified Railcar Truck Stand

Abstract

The objective of the work presented in this paper is to gain an understanding of the structural analysis and of a simplified railcar truck stand using the computational (CAE/FEA) and math tools. Railcar truck stands are used in railway industry and consist of several complex shaped members that are welded together. They are used during maintenance operations to support one end of a freight car or a commuter car, for example, to change the wheel bearings, etc., while the other end is on the rail track. From a safety perspective, the stand needs to be designed carefully, but at the same time due to their possible large volume of production, this structure needs to be optimized from strength and cost perspectives besides other parameters such as long life, etc.

The work carried out in this paper is based on one of the term projects of a mezzanine level mechanical engineering (ME) elective course on Lightweighting and Joining of Structures. The 11-week duration class (including the final exam week) at Kettering University consisted of both senior undergraduate and graduate ME students. The prerequisites for the course include mechanics, CAE, design, material science and finite element analysis (FEA). For the analysis carried in this paper, the currently used all-steel railcar truck stand has been redesigned and modeled as a simplified 3D space frame using standard tubular (pipe section) members. Although the simplified model does not represent in any way the actual stand used in the railcar industry, it is anticipated to serve the same purpose as the original stand as stated in the first paragraph. One of the course learning objectives (CLOs) is to model a given real physical system ready for analysis, and this simplification addresses to some extent the stated CLO. Modeling of 1D and 3D structural frames along with the underlying assumptions, and their limitations has been discussed in the class following real life examples available in the standard textbooks on CAD and finite element analysis. Traditional strength, buckling and impact analyses of the simplified 3D model of the frame have been carried out in this work under various loading and constraint conditions. Further, virtual experiments for the optimal design and material makeup of the various truck stand designs analyzed using the NX CAE tool has been carried out and the results compared with the results from 1D simulation have been compared with an earlier work that used the AutoDesk simulation tool. The main design variables in these stands are the geometry, material and safety factor. The teaching and learning outcomes of the work along with the safety and ethical issues have been discussed. It is hoped that through this study the students develop a clear understanding of assumptions made in the CAD and FEA course topics on frames and how they address the CLOs.

Introduction and Literature Review

Structural analysis of space frames is not a new subject. There are numerous textbooks and research papers available on this topic [1-4]. In addition, several CAE tools have been used for structural analysis of trusses and frames [5-7]. Only few references are provided in this paper as an example. However, this list by no means is complete. The work reported in this paper is based on an earlier work published in ASEE by the current author along with the other co-authors [8], and the work presented by one of the students at MS&T conference [9]. The senior elective course is offered as an independent study few times with enrollment each time of no more than 4 to 6. For the class project, both NX and SolidWorks CAE tools have been used for the simulation [10]. Several MATLAB files for 3D truss and frame analysis are also available at MATLAB Central File Exchange [11 - 12], however, these codes have to be understood thoroughly to avoid any errors, and used carefully to verify the preliminary CAE results. In view of this, it is always desirable to write one's own MATLAB code, but it takes time to assemble the local and global stiffness matrices in FEA.

In order to fully understand the structural analysis using FEA, the concepts of modeling, free body diagrams (FBD using Newtonian mechanics) must be well understood. Normally, those students taking an elective course such as the one mentioned in this paper that is based on using CAD and FEA will have had some introduction into the analysis of structures, the forces and moments in beams, and the coordinate transformations, etc. however, due to the possible time gap between taking several of these prerequisites, it would be best if these concepts and principles were reviewed to ensure that the class, as a whole, is up to speed and can proceed at the same level and pace. In this context, using a CAD/CAE tool for the FEA will help to enhance the classroom activities. Usually, many students at Kettering university use the CAD, and to some extent, the CAE tools in several other core courses, so little or no navigation of the CAD is needed although a brief coverage of the FEA theory and navigation is needed. With the current advancements in technology, several online videos that are specific to the particular FEA usage enhances the students' confidence to use such tools and reduces the learning cycle time.

As mentioned before, this project was assigned during the 2nd week of the 11-week Summer term at Kettering University. For this particular elective, there were only 3 ME students – one undergraduate who is familiar with using the NX CAE software who worked individually on the project, while two graduate students who are familiar with SolidWorks worked together as a group. There were other group assignments carried throughout the term besides the final project. Obtaining a good-looking CAD model with applied boundary conditions in no way compromises the need to understand the problem being approached or the impact of the assumptions used to build and analyze the model. On the other hand, preparing a good-looking coarse or fine FEA mesh may be relatively easy, but in no way guarantees the user an accurate analysis. Therefore, obtaining and accepting a solution given by a CAE tool may be easy and satisfying, but it cannot deliver a true understanding of the real-life implications of the designs. This is where a teacher needs to help the students to interpret the results correctly through discussion of results, and to make sure that they make sense. In this context and within the purview of Bloom's Taxonomy, space frame analysis definitely occupies a higher level of understanding and builds on the application of the basic knowledge gained in the fundamental applied mechanics courses such as Statics, Mechanics of Solids and Finite Element Analysis. Size (geometry) and multi-material (MML) optimization of the members of the 3D frames to satisfy the strength and factor of safety (FOS) constraints is very challenging and require one to use a math and/or a CAE tool.

Problem statement

Conduct virtual experiments for the optimal design and material makeup of a stand structures using math and CAE/FEA tools. Multi materials using a combination of standard steel and aluminum tubing (scheduled pipe sections) is to be analyzed such that the 1-D truss or beam element analysis has been carried out in the math and the CAE tool that you used. Another goal is to obtain nearly the same factor of safety (FOS) in all members of the structure for the same given loading and constraint conditions. Time permitting, repeat the same work using 3D finite element analysis.

Modeling and structural analysis

As mentioned before, Railcar truck stands are used during maintenance operations to support one end of a freight or commuter railcar (example, Chicago Transport Authority, CTA). All the members of the existing stand are bulky, and are usually welded together. They are optimally designed to meet the safety standards dictated by the railcar industry. Finally, these stands are optimized from strength and cost perspectives. The photograph on the left in Figure 1 shows the original design used by the CTA [13], while the schematic of the railcar, wheels and the proposed box stand are shown on the right. The railcar is lifted on one side and the stand inserted in place while the maintenance crew removes the rigid axle with the wheels to carry out the periodical repair work. The original design shown (left) in the photograph is certainly sturdy and safe for use. However, they are not very strong should there be any torsional loads, besides being heavy and expensive. Thus, the goal of the project work assigned is to propose new and simple design that is lighter but safe to use.

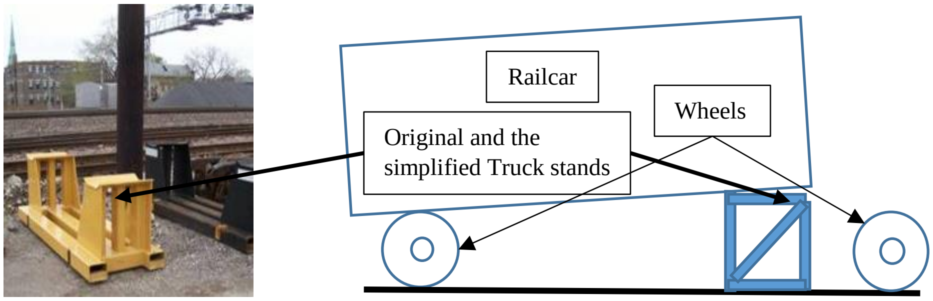


Figure 1: Original design of a railcar truck stand [left, 13] and the schematic of the proposed simplified design (right)

Figure 2 shows the sketch of the CAD model of the preliminary design of the newly proposed box frame using AutoDesk (left) [8] and NX (middle). The sketch on the right in Figure 2 shows the different circular sections used for the members isolated at the top joint (node) in the finite element analysis (FEA). As mentioned before, this design can be used in place of the original truck stand that is expected to hold the weight of the railcar. However, more design modifications are needed to meet the safety, specific design and joining standards of the railcar industry. The overall dimensions of the new design are similar to the original truck stand and they are in mm as shown on the sketch.

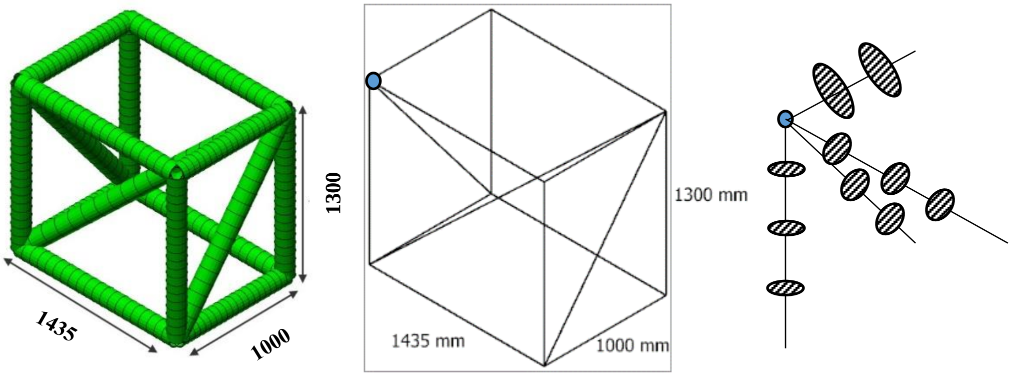


Figure 2: Structure Dimensions Using Scheduled Pipes in AutoDesk (left [8]), NX (middle), and Top left Joint with Circular Sections (right)

Figure 3 (together with Table 1) shows both the 1D line model and the tabular data of the material makeup using structural A36 steel pipes for the stand [14] with yield strength of 250 MPa. The cross-sectional area and other dimensions of the schedule pipes published in the data tables and those that are embedded in the CAE tools and selected by the users can be slightly different due to the pipe thickness, 'flow area' and 'internal area'. This gives slightly different simulation results in the FEA. This difference can be significant if 1D elements versus fully-blown 3D element models are used. Element size (i.e., mesh size) and the type of beam or frame elements (2D planar, 2D grid or 3D) are the other contributors to the discrepancy in the FEA results. This is a good learning outcome for the students to realize that multiple design iterations are needed.

Member	Pipe/dim/internal area
1	5" SCH 40; 2600 mm^2
2	½" SCH 40; 148 mm^2
3	2-½" SCH 40; 1025 mm^2
4	½" SCH 40; 148 mm^2
5	½" SCH 40; 148 mm^2
6	½" SCH 40; 148 mm^2

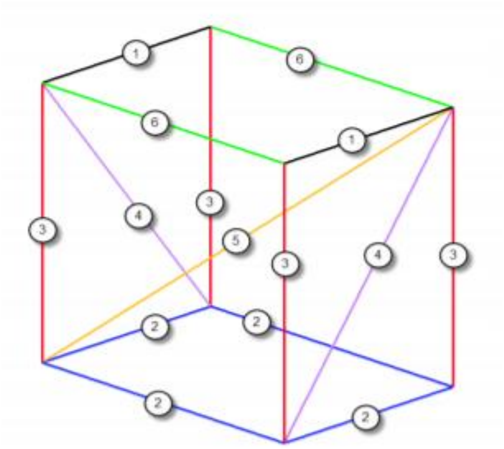


Table 1 (left)/Figure 3 (above): Different Pipe Members used for the simplified box stand of the railcar

Most CAE tools such as OptiStruct [15] have some optimization capability (size, shape, or topology).

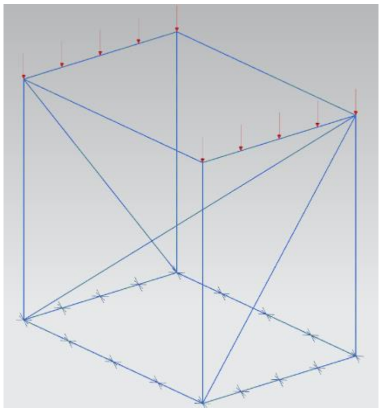
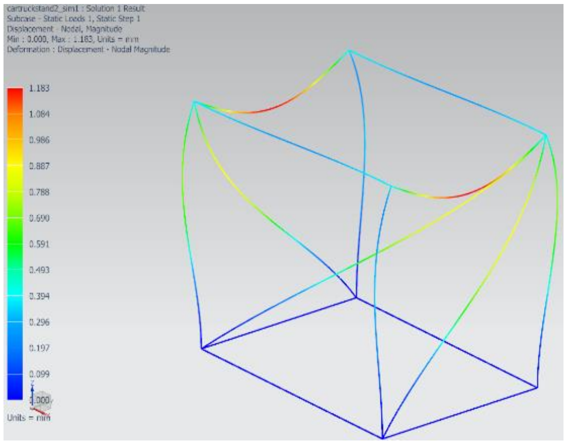


Figure 4: FE Model with Boundary Conditions

Keeping in mind that one of the objectives of this paper is first to validate the results of the earlier published work [8], FEA has been performed using 1D mesh in NX. Figure 4 shows the FE model of the stand with twenty (20) 1-D, type PBeam elements per each member. With 15 members and common nodes at each joint (connection), the total number of elements and nodes in the FE model are observed to be respectively, 300 and 293. All the bottom nodes of the stand are constrained in all directions (6 degrees of freedom per node), and vertical load applied to the top nodes on the two members. This is somewhat consistent with how the load is applied to the vertical frame members of the original stand shown in Figure 1. Also, this model is similar to the earlier work [8]. Since the stand is intended to be used for different types of wagons and an engine, the typical weight of a freight train engine is used in the design. The engine weight is around 40 Tons (80,000 lbf = 177,928.8 N), with the railcars weighing much less than the engine. During the maintenance, one end of the engine car is lifted and the other end stays on two side wheels on the railway track. It is therefore assumed that only half of the weight of around 88,964.4 N (20 Tons) is to be borne by the truck stand. This load is applied as vertical distributed load acting downwards on each of the two 5” SCH 40 pipes on the top two members of the stand (shown as members 1 in Figure 3). Self-weight of the stand as calculated by NX of 104.6 kg (1026.13 N) has also been included in the NX FE simulations. The center of gravity location and the other section properties of the stand can also be obtained from NX.

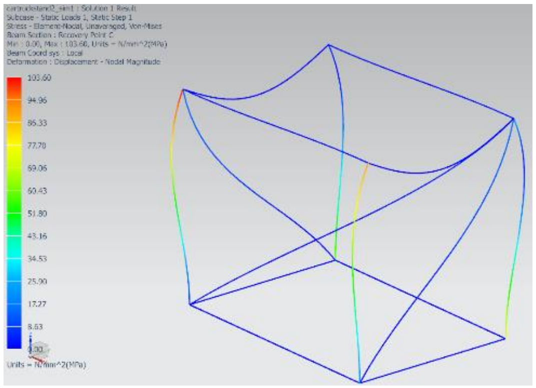
Based on the above FE model, solution from three iterations are presented in Figures 5 and 6. Figure 5 (together with Table 2) shows the magnified (exaggerated) deformation contour plot of the members of the stand solved by NX. As expected, the top two members (labeled as Member 1 in the Table) experience the maximum deformation of nearly 1.2 mm, which is quite acceptable, being very small. The results of previous work using AutoDesk [8] gave exactly the same result of 1.19 mm. Both NX and AutoDesk 1D simulations yielded similar results since the solution process for 3D space frames and the database used for the steel pipe schedule are almost identical.

Figure 6/Table 3 show the stress contour plot of the members of the stand solved by NX. As expected, the vertical four members (labeled as Member 3 in the Table) experience the maximum von-Mises stress of 103.6 MPa. For the material used, this gives a safety factor of 2.4. However, AutoDesk gave a stress value of 120.1 MPa [8], which is a difference of about 16.4 MPa. In FEA, the stresses are usually calculated based on the strains and the moduli values, which is where the difference can occur.



Member	Deformation by NX (mm)
1	1.18
2	0
3	0.71
4	0.88
5	0.88
6	0.38

Figure 5/Table 2 (right): Plot/values of the Deformation of the Stand Members



Member	Stress (MPa)
1	0.37
2	0
3	103.6
4	23.2
5	6.9
6	3.6

Figure 6/Table 3 (right): Plot/values of the Maximum von-Mises Stress in each member of the Stand (NX)

Further studies have been conducted using NX for impact and buckling analyses. 1D type PBeam elements have been used. For the impact analysis, simple hand calculations have been done to calculate the impact force as shown below. Note that these calculations are done using the US Customary Units. NX allows inputting mixed units. However, the final calculations are internally converted to the initially chosen Units, which in our case is the SI Units. Buckling analysis is presented later in this paper.

Impact force calculations:

$$a = \frac{\sqrt{2gh}}{t}$$

If the car is dropped from 6” (0.5 ft) above the stand:

$$a = \frac{\sqrt{2*32.2*0.5}}{0.08} = 64 \text{ ft/s^2} \approx 2 \text{ g; this estimates a time pulse of } 0.08\text{s}$$

Using $F = ma$

$$m = \frac{40 \text{ ton coach}}{2 \text{ (stand only supports one side)}} * \frac{1}{2 \text{ load bearing beams per stand}} * \frac{2000 \text{ lbf}}{1 \text{ ton}} * \frac{1 \text{ slug}}{32.2 \text{ lbf}} = 621 \text{ slug}$$

This gives:

$$F = 621 * 2g = 40000 \text{ lbf}$$

The impact force calculated above is twice the amount of force used for the earlier presented linear static analysis. Therefore, we can expect that the deformation and the stress will increase proportionately. The magnitude of the maximum deformation on the top two members from the NX simulation came out to be 2.366 mm and the maximum von-Mises stress of 207 MPa on the vertical members of the frame. Since the yield stress is 250 MPa, the stand is marginally safe under impact loads, which needs to be taken into consideration while carefully loading the railcar on to the stand. Linearity studies for both the deformation and the stress have been done by varying the load on the stand from 20,000 lbf (88964.4 N) to 80,000 lbf (355,857.6 N) in four steps with R^2 values of nearly 1.0. As mentioned before, NX accepts inputting mixed Units which are internally converted to proper Units.

Similar studies (of linear static analysis) have been performed using 6061-T6 aluminum alloy for all members made of similar aluminum schedule pipes available in the literature. NX calculated the self-weight of all aluminum stand of 36.2 kg, which is nearly 1/3rd of the A36 steel stand. The deformation and the von-Mises stress contour plots have also been obtained for each member of the aluminum stand using a similar FE model in NX except for the material change. The maximum deformation in the top two members came out to be 3.56 mm, while the maximum stress remained same as for steel at 104 MPa. This gives the minimum safety of around 2.4. Since the deformation is proportional to the modulus of elasticity, E, one can expect that the deformation of aluminum members should be nearly three times greater that of steel.

Few more simulations have been performed using the steel and aluminum material combinations (multi-materials, MML) using the 1D elements in a view to optimize the deflection and the weight of the stand. The purpose of the MML study is to initiate an understanding of how to perform optimization by trial and error method of the stand that satisfies the criterions of minimal deformation, minimal stress (below the yield point) and that the design that yields nearly the same safety factor in all members of the stand. The asymmetrical stand design presented in this paper is just one of these attempts that fulfils to a great extent the stated objectives.

Figure 7 shows the first iteration in which the multi-material combination (denoted as MML 1) is arbitrarily chosen for the stand. The members in green color are all A36 steel schedule pipes and those in blue are 6061 aluminum schedule pipes. This stand weighed 64.4 kg. Figure 8 shows the deflection and von-Mises stress contour plots for MML 1. A second iteration with multi-materials (MML 2) has also been carried out for a different arbitrarily chosen material makeup of the members. The self-weight of this stand is around 93.6 kg, which is slightly higher than the previous material combination. Many other member size and material combinations have been tried, similar to those reported in the earlier study [8]. Table 4 shows the summary of all the important results obtained from the 1D NX simulation.

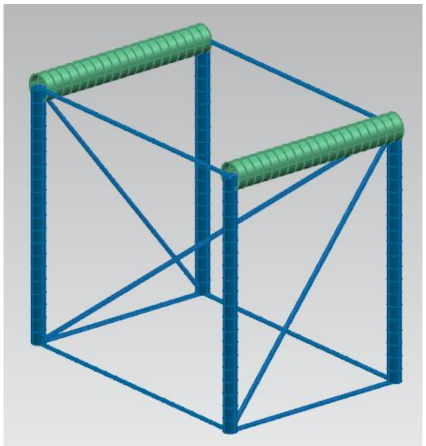


Figure 7: Iteration 1 with Multi-materials Using NX

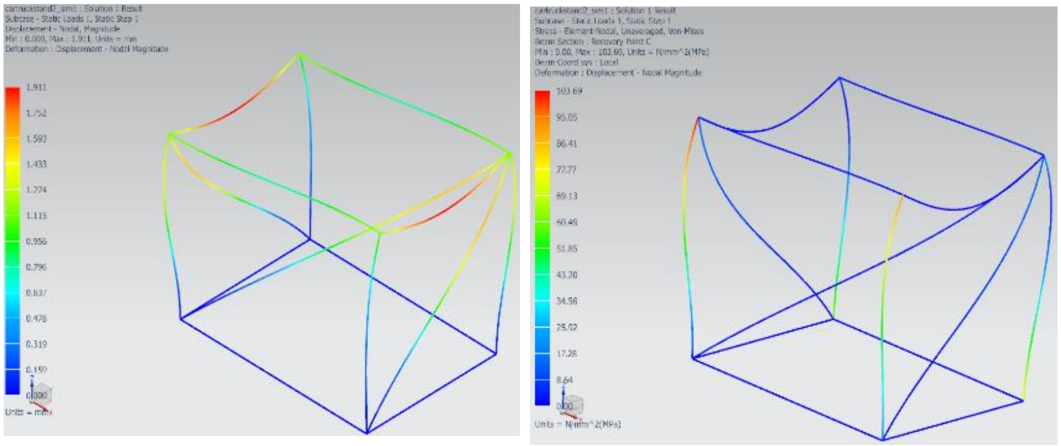


Figure 8: Maximum Deformation (1.91 mm) and Maximum Stress (14.1 MPa) from Iteration 1 with multi-materials (MML 1) using NX

Table 4: Comparison of Results of 1D Models Using NX (See Figure 3/Table 1)

Material	Mass (kg)	Max Deflection (mm)	Max von-Mises Stress (MPa)
All Steel	104.6	1.183	104
All Aluminum	36.2	3.56	104
MML 1	64.4	1.91	104
MML 2	93.6	1.17	104

It can be observed from the results presented in Table 4 that the magnitude of von-Mises stresses for each type of stand remain nearly the same due to the same geometry used for those members. The deflection values are the lowest in the MML 2 model, and it provides some weight savings in mass production, compared to the all-steel stand. Therefore, this design may be recommended for further studies in order to obtain nearly the same safety factor in each member of the stand. Obviously, this is not an easy task as size and even topology optimization of a manufacturable stand needs to be performed. Moreover, joining concerns need to be addressed due to the materials being dissimilar. As an example, literature shows that steel and aluminum can be joined using the following steps:

- Surfaces will be cleaned before joining
- Hot dip aluminize steel before welding
- The steel and aluminum will then be fusion welded together
- Structure will be pickled after joining
- The structure should be primed and coated with paint

With the advancements in multi-material joining processes using lasers, it may be possible to design and fabricate such stands. Until then, given the increased joining complexity and additional cost of buying aluminum when compared to steel, that the all-steel model may be the best. From a cost perspective, the current cost per foot of 1/2” SCH 40 pipe is $3.53 for 6061-T6, and $1.27 for A36, which further supports the use of all-steel frames. However, corrosion and other issues such as recyclability, etc., favors more the use of aluminum alloys for long-term usage. Next section shows the buckling studies.

Linear Buckling Analysis:

Buckling analysis of the steel frame has been carried out using NX and compared to those carried by AutoDesk [8]. In all, eight modes of buckling along with the respective magnitudes of the maximum deflection of the frame members. Figures 9 thru 12 show sample results of the buckling analysis. As can be seen in Figures 10 to 12, the diagonal members being longer, tends to buckle compared to other members of the stand. These results match very closely with the AutoDesk results [8]. Buckling of MML stands was not carried; however, all-aluminum stand shows similar modes of buckling.

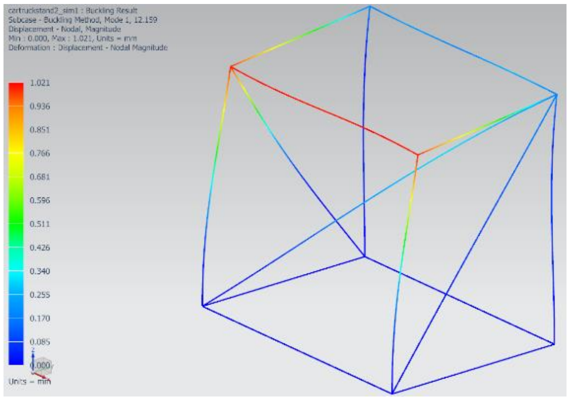


Figure 9: Mode 1 Deformation = 1.021 mm

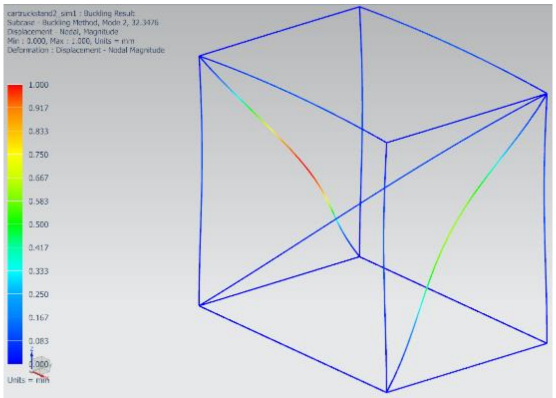


Figure 10: Mode 2 Deformation = 1.00 mm

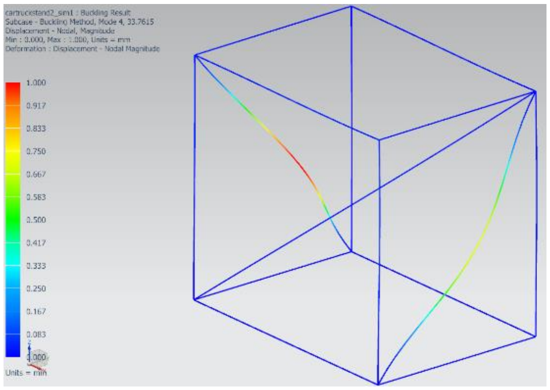


Figure 11: Mode 4 Deformation = 1.00 mm

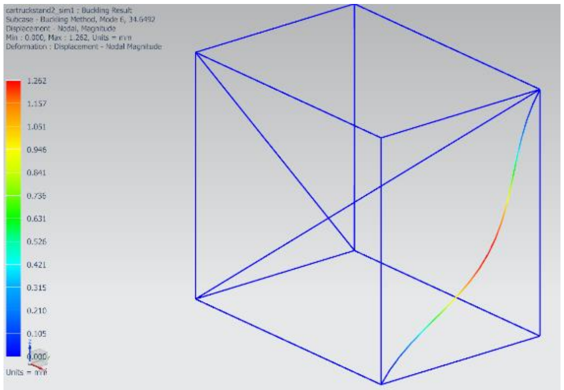


Figure 12: Mode 6 Deformation = 1.262 mm

3D Model and Analysis by SolidWorks [10]

The other group of two graduate students of the same class used SolidWorks in order to compare their 3D element mesh results with the 1D element solutions by AutoDesk and NX. Several 3D designs of the stands have been analyzed. Table 5 shows the overall results of six iterations of the stand with 3D mesh using SolidWorks. Last column lists the factors of safety (FOS) by both Solid Works, NX and AutoDesk 1D. Note that the AutoDesk 1D element results closely match with the NX 1D element results, as mentioned earlier. Therefore, this table does not list the 1D NX results. Also, note that the AutoDesk results for FOS are available for only two cases as shown. While the FOS calculated by SolidWorks and AutoDesk are very close for the 1.5” SCH 40 all-steel pipes, it vastly differed for the 3.5” SCH 40 all-steel pipe. One of the reasons may be the quality and size of mesh at the joints for the 3D model where three to five members are connected together using the Boolean operations. Identifying the general location on the model, especially at the joints where the maximum stresses are calculated and outputted is critical for the user to identify. This is certainly one of the learning outcomes of the course, in which interpretation of results is very important.

Figure 13 shows the implications of joining 3 example members with different cross sections (on the left). In real life, there are different ways to join such members; for example, using gusset plates or simply by welding together, followed by other finishing operations. On the other hand, depending on how these are joined (assembled) in the CAD model (on the right), FEA requires the users to specify the friction coefficient at the intersecting surface to surface contact zones so that proper type of contact elements is established. The implications of the joining processes, particularly in 3D element modeling and the theory behind the meshing at the contact zones need to be explained and clarified to the students before they accept (or not) the default values set by the CAE tool. Literature shows that there are a few types of contact elements that the users can use while meshing assemblies with the same or different types of materials. Needless to emphasize that safety and ethical issues due to poor quality joining methods needs to elaborated and discussed with the students to understand their impact on the people and the society at large. When possible, an outside speaker from a nearby industry can be invited in-person or virtually to explain the practical considerations in joining members with the same or different geometric and material properties on the strength and integrity of such joints. Other issues such as corrosion and residual stresses at the joints can be explained and discussed. Take-home exercises can be assigned as a group or individually to read the available online resources and ask them to present their findings to the class.

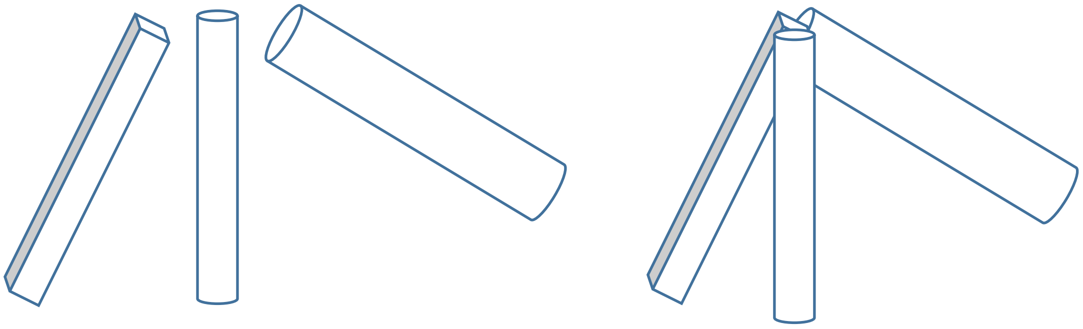


Figure 13: Implications of Joining and Meshing Members with Different Sizes, Cross Sections and Materials Using Boolean Operations

Table 5: Comparison of Results of 3D Element Models Using
SolidWorks with 1D AutoDesk FOS Results [10]

Material	**Max von-Mises Stress (MPa)**	**Deformation (mm)**	**Factor of Safety**	
			SolidWorks	**AutoDesk**
1.5 SCH 40 All Steel	1152.09	21.66	0.216	0.203
1.5 SCH 40 All Aluminum	1167.66	62.94	0.076	Not Attempted
1.5 SCH 40 MML	1209.03	23.18	0.207	Not Attempted
3.5 SCH 40 All Steel	79.36	0.80	3.183	1.45
3.5 SCH 40 All Aluminum	79.40	2.40	1.133	Not Attempted
3.5 SCH 40 MML	83.36	0.85	2.999	Not Attempted

Safety Concerns and Ethical Issues

By working on this project, the students clearly understood that design and analysis of real life railcar truck stands involves a clear understanding of the assumptions made in the design, modeling, and analysis stages. Understanding the role and use of design standards followed by the transportation industry, for example, APTA [16] is also very important. These standards are available for each component of a railcar and for the prime movers (engines or locomotives). Understanding the safety concerns and the lives of railroad workers and the passengers is very important while also optimizing the costs for the transportation industry. Understanding the limitations of the design and analysis, particularly in 3D modeling and FEA is paramount to meeting the ethical standards, for example, NSPE Code of Ethics [17]. These standards were reviewed and discussed using few examples during the class.

Learning Outcomes

As mentioned earlier, the final learning outcomes of assigning this project work to the class include understanding the connection between the knowledge gained in the prerequisite courses such as Statics, Mechanics of Materials, Design, CAD, Engineering Materials and basic FEA, and the present elective course on Lightweighting and Joining of Structures. Review of the different engineering assumptions made in both modeling and analysis of trusses, frame structures helped the students understand the results better. Programming skills in applying the math tools learned in math classes such as Maple, Excel, MATLAB, etc., needs to reviewed and practiced using the available worked examples in standard FEA books. Ethical and safety issues of designing load bearing structures have been discussed to understand their impact on people and the society at large. Importance of using the engineering standards for the material selection, design and analysis goes hand in hand with the safety and ethical issues. Time management skills and teamwork to both understand the process and the steps taken to work on the assigned projects and take-home work is realized by the students.

Conclusions

This paper presented the design and analysis of a simplified railcar truck stand used in the maintenance operations of commuter and freight trains. Use of various CAE tools to perform several design iterations of the truck stand elevated the understanding of the assumptions made in the analysis and the associated limitations of the work. Various material makeups for the stand are considered in a view to obtain an optimum design that is both lighter, safer and somewhat cost effective. The results of the structural analysis including buckling analysis of the 1D and 3D finite element methods performed by AutoDesk, NX and SolidWorks are compared, and the differences between these observed. While the results of 1D element analysis are similar between the three CAE tools used, difference in 3D modeling was observed due to the method of solver settings in SolidWorks and the meshing details at the joints. More work is required in terms of geometry and topology optimization of a simple frame such as the one considered in this paper in order to satisfy several functional and design constraints. Math tools such as MATLAB can be used to develop a code using 1D elements, and to perform similar studies. This helps in understanding the finite element assembly and solution processes better than just using a CAE tool. Students learned the use of 1D analysis in NX for structural and buckling studies.

Students documented their qualitative learning outcomes from taking this elective course as follows:

- “I’ve learned the 1-D analysis from scratch”
- “I’ve also gotten results in respect to buckling and impact analysis”
- “I experimented using different materials”
- “I learned about joining mixed material joining”
- “It can be extremely difficult to optimize a structure”
- “After completing this project, we learnt new features in solid works like weldments, applying different materials for different members of the car truck stand”
- “We learnt how to use multi-materials in order to improve overall strength, reduce weight and increase efficiency of the structure”
- “We were able to perform different analyses like static, buckling and Impact analysis on the car truck stand”

References

1. T. Burns, *Applied Statics and Strength of Materials*, 2nd Edition, Delmar (Cengage Learning), ISBN-13: 978-1435413313, 2022.
2. F. Beer, E. Johnston, J. DeWolf, and D. Mazurek, *Mechanics of Materials*, 8th Edition, McGraw-Hill, ISBN-13: 978-1260113273, 2019.
3. S. S. Rao, *The Finite Element Method in Engineering*, 6th Edition, Butterworth-Heinmann, ISBN-13: 978-0128117682, 2017.
4. D. L. Logan, *A First Course in the Finite Element Method*, 6th Edition, Cengage Learning, ISBN-13: 978-1305635111, 2017.
5. H. H. Lee, *Finite Element Simulations with ANSYS Workbench 2021*, SDC Publications, August 9, 2021.
6. R. Anderl and P. Binde, *Simulations with NX / Simcenter 3D 2E: Kinematics, FEA, CFD, EM and Data Management*, 2nd Edition, ISBN-13: 978-1569907122, 2018.
7. P. Kurowski, *Engineering Analysis with SOLIDWORKS Simulation 2021*, SDC Publications, March 18, 2021.
8. R. Echempati, V. Laddha, S. Ozel and J. Herzing, "Design Optimization of a Car-Truck Stand" Presented at *ASEE 2012 Conference*.
9. B. Shrivastava and R. Echempati, "Optimum Design of railcar truck stand", presented virtually at *MS&T 2021 Conference* held in Columbus, OH, October 2021.
10. B. Shrivastava, A. S. Gone and K. P. Medishetty, *Unpublished Final Project Reports* of the Elective Course on Lightweighting and Joining Technologies of Body in White (BIW) Structures, Kettering University, Summer 2021.
11. A. Kumar, "Finite Element Analysis (FEA) of 2D and 3D Truss Structure" (https://www.mathworks.com/MATLABcentral/fileexchange/68337-finite-element-analysis-fea-of-2d-and-3d-truss-structure), *MATLAB Central File Exchange*. Retrieved February 2, 2022.
12. A. Ventura, "Structural analysis of 3D frames" (https://www.mathworks.com/MATLABcentral/fileexchange/49559-structural-analysis-of-3d-frames), *MATLAB Central File Exchange*. Retrieved February 2, 2022.
13. J. Clark, Unpublished *MECH-519 Final Project Report*, Kettering University, 2010.
14. Steel-Pipe-Data: http://extolohio.com/wp-content/uploads/sites/2/Steel-Pipe-Data-Pipe-OD.pdf.
15. OptiStruct: https://www.altair.com/optistruct/
16. American Passenger Transport Association (APTA): https://www.apta.com/research-technical-resources/standards/passenger-rail-equipment-safety-standards/
17. NSPE Code of Ethics.