AC 2012-3018: DESIGN OPTIMIZATION OF A CAR-TRUCK STAND

Prof. Raghu Echempati, Kettering University

Raghu Echempati is in the Department of Mechanical Engineering at Kettering University. He has several years of teaching, research, and consulting experience. He is an active member of ASEE, ASME, and SAE. He has more than 100 technical paper publications in journals and conference proceedings of repute. He has chaired several sessions at national and international conferences and delivered numerous invited talks and keynote addresses. He has reviewed several textbooks, journal papers, and conference papers. He is an active member of many conference committees.

Mr. Vipul Laddha, Kettering University
Mr. Sualp Ozel P.E., Autodesk, Inc.

Sualp Ozel is a Professional Engineer who spends his time listening to simulation experts' product needs and designing solutions to advance the use of simulation. He holds a bachelor's degree in mechanical engineering from Virginia Polytechnic Institute and State University. Before joining Autodesk as a Product Manager, he worked for RJ Lee Group, Inc., and Algor, Inc.

Mr. James Herzing

DESIGN OPTIMIZATION OF A CAR-TRUCK STAND

Abstract

Car-truck stands are used during maintenance operations to support one end of a freight car or a commuter car used in railway industry. Usually, they consists of several pieces of steel tubes and other steel members welded together to form a rigid frame structure. From a safety perspective, the car-truck stands are 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 purpose of this paper is to conduct virtual experiments for the optimal design of a different car-truck stand structures using Autodesk Simulation program as a CAE tool. The idea is to include this work as a part of final project in a traditional finite element analysis (FEA) course taught at Kettering University. For the work reported here, the structural steel members are simplified by using standard pipe sections, which are then optimized for strength and weight reduction, as well as for buckling. It is hoped that through this study a clear understanding of assumptions made in the FEA course topic on frames is realized by the students. Initial assessment done indicates that students appreciated the use of a CAE tool for optimal design of frames and other structural members.

Introduction and Brief Literature:

There are several railcar industries throughout the world including trains in USA, Canada, Europe and other countries. Truck stands are used for both assembly and service operations of railcars. Although not a very critical component, due to the large volume of stands used by the rail industries, optimal design is justified to minimize the costs. The study and structural analysis of a car-truck stand was carried out as a part of FEA course studied at Kettering University. Several textbooks on FEA are available in the literature, for example Logan [1], and Carroll [2]. Earlier work done by Fox and Echempati [3] consisted of performing design of experiments (DOE) to study the effect of changing the geometry variables of the stand structural members on the overall strength of the stand. A conventional design of the car stand is shown in Figure 1, which is made of structural steel members. Several designs of this stand were virtually analyzed using trial and error approach in a view to obtain an optimum design. Unigraphics was used for solid modeling while SolidWorks software was used for analysis. It was observed that the software didn't have enough capabilities to do optimization of the structural members that yield a desired safety factor. Therefore, using trial and error method, finite element analysis was extensively used on one of the optimized designs to analyze the member stresses of the rigid frame. A safety factor of 2 based on strength has been assumed in the design. Buckling analysis using hand calculations was also carried out in that study. A real prototype of one of the optimized stand has been fabricated and stress components in the critical members were measured using strain-gage technique and the results compare with the virtual predictions. The results did not compare well as expected due to the welded connections on the real prototype being not fully modeled in the virtual studies. Also there could be issues with the strain-gage mountings and measurements that lead to the discrepancies in the results.

As compared to work reported in [3] in which only the variation in width/thickness of the sections used was carried by the trial error method, in this paper, a new design of the car stand is considered as shown in Figure 2, in which different cross sections and sizes were used and both stresses and deflections were obtained from the CAE analysis. As mentioned before, the design and FEA simulation of this car stand was done using **Autodesk® Simulation Mechanical**. This design was optimized in 2 stages. In stage 1 an exhaustive optimization analysis was performed and results were exported to Excel for graphing purposes. In stage 2 the design was subdivided in to unique parts in order to optimize each member based on weight and factor of safety.

Figure 1: Conventional design of a car-truck stand

As mentioned above and from an educational perspective, a different car stand has been considered and optimized. The structure is shown in Figure 2 and it consists of several members with different cross sections of the same shape.

Evaluation of car-truck stands using Autodesk® Simulation Mechanical

Design Assumptions:

Following the technical specifications for railway tracks [4], the following specifications for the car stand are used for the test stand.

a. Distance between rails is 1435 mm.
b. Distance between rail track and coach floor is 1300 mm.
c. Car stand is 1000 mm long.
d. Beam element type will be used for the analysis
e. In order to eliminate any confusion in the strong vs. weak axis, a symmetrical cross section will be used for the optimization
    a. AISC 2005: Pipe Schedule 40 (STD)
    b. AISC 2005: Pipe Schedule 80 (XS)
    c. AISC 2005: Pipe XXS
f. Weight of coach is 40 tons. So the structure has to withstand load of 20 tons. (Assuming other sets of wheel will support 20 tons).
g. Material used for Car Stand is Structural steel ASTM A36 steel.
h. Yield strength of structural steel (ASTM A36) is 250 MPa.

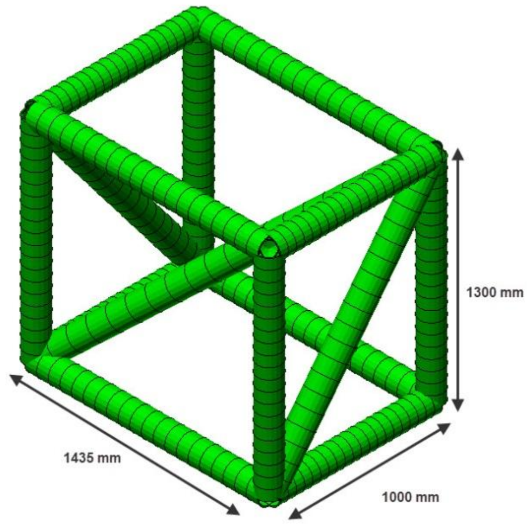

Figure 2: General Dimension of a car-truck stand

General dimensions of the new car stand are shown in Figure 2. The only parameter that can be changed is the cross section of the beam. Thirty seven iterations have been performed using Pipe cross section from the AISC database to determine the optimum cross section of this car stand. The major area of focus was keep stress and deflection of each member under the specified limits while minimizing the weight

Case 1:

Figure 3 below shows the design based on 1-D beam elements and loading of top two beams. Each beam member has been divided in to 20 subdivisions (elements) in order to obtain a more precise result. The weight of locomotive coach is assumed to be 40 tons. Hence this structure must support 20 tons. Loading has been done on projected beam length considering these loads. Also, 4 bottom corners of the car stand has been clamped, thereby, fixing all (6) degrees of freedom.

20 Tons-force = 177,928.8646 N
This load is applied to 2 beams
Each beam member receives 10 tons (88,964.4323 N)
Load per member is 88,964.4323 N/1000 mm = 88.9644323 N/mm

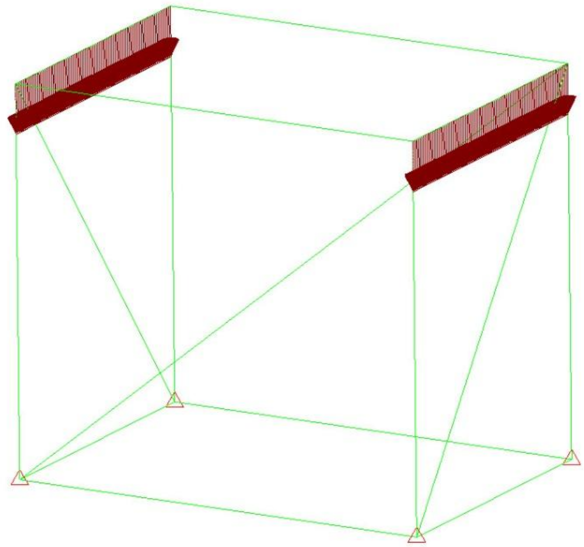

Figure 3: Loading and clamping of a car-truck stand

Based on the above set of load and boundary conditions, a structural optimization analysis was carried out using the following thirty seven cross sections (Table 1).

Table 1: List of thirty seven Pipe cross-sections used for initial optimization

Section Name	Schedule
Pipe 1/2	40,80
Pipe 3/4	40,80
Pipe 1	40,80
Pipe 1-1/4	40,80
Pipe 1-1/2	40,80
Pipe 2	40,80,XXS
Pipe 2-1/2	40,80,XXS
Pipe 3	40,80,XXS
Pipe 3-1/2	40,80
Pipe 4	40,80,XXS
Pipe 5	40,80,XXS
Pipe 6	40,80,XXS
Pipe 8	40,80,XXS
Pipe 10	40,80
Pipe 12	40,80

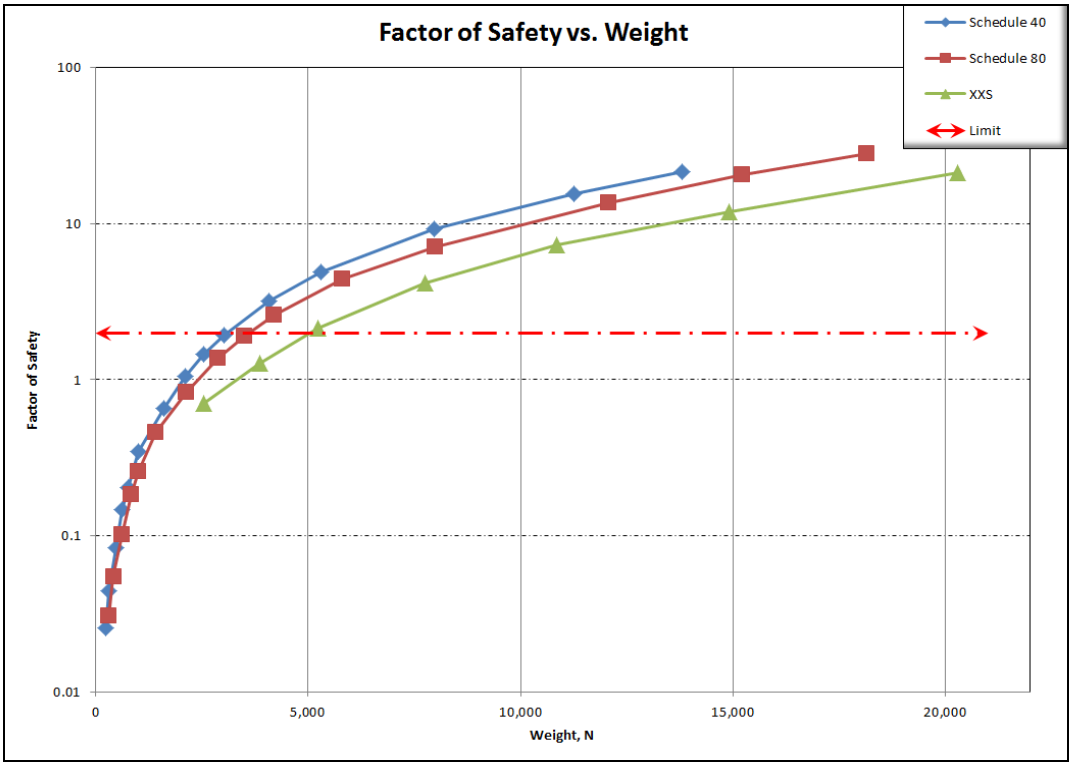

Figure 4: Log-graph of "Factor of Safety vs. Weight" for three different types of Pipe type beams.

Findings:

Table 2 below lists the 3 cross sections that gave a factor of safety of just above 2.

Table 2: Three cross sections with Factors of Safety slightly above 2.0

Section name	**Cross sectional Area, mm^2**	**Weight, N**	**Factor of Safety***
Pipe 5 Schedule 40	2600	4,089	3.20
Pipe 4 Schedule 80	2671	4,201	2.58
Pipe 3 XXS	3329	5,235	2.15

* The Factor of Safety shown in the table above refers to Yield Strength of 250 MPa divided by the worst stress in the beam.

Note that a larger cross sectional area does not necessarily mean a higher factor of safety. For a full list of all the results, refer to the attached **Appendix A – Table A1.**

The best choice based on
Figure 4 and Table 2 is "*Pipe 5 Schedule 40*". The displacement and stress results obtained using the software for this choice is shown in Figures 5 to 8. Figure 5 shows the deflection contours while Figure 6 shows the axial stress in members with their

maximum values. The worst stress for this cross section is 78.08 MPa and it is in the "Bending about Local 2 Axis" as shown in Figure 7.

Factor of Safety Calculation:

FOS = Yield Strength / Worst Stress (bending about local 2 axis)
= 250 MPa / 78.08 MPa
= 3.2

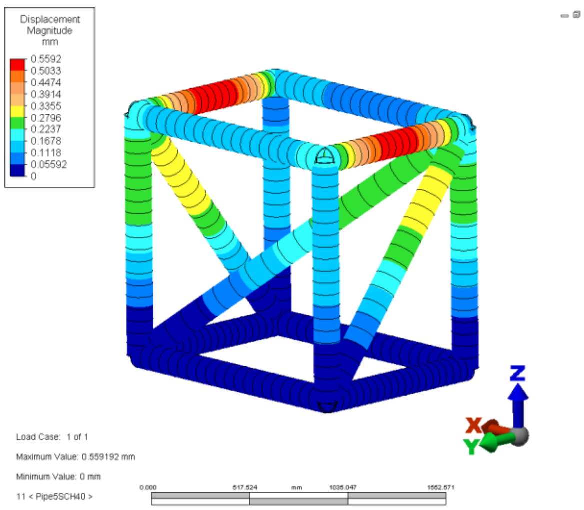

Figure 5: Displacement results for Pipe 5 Schedule 40 (0.56 mm max)

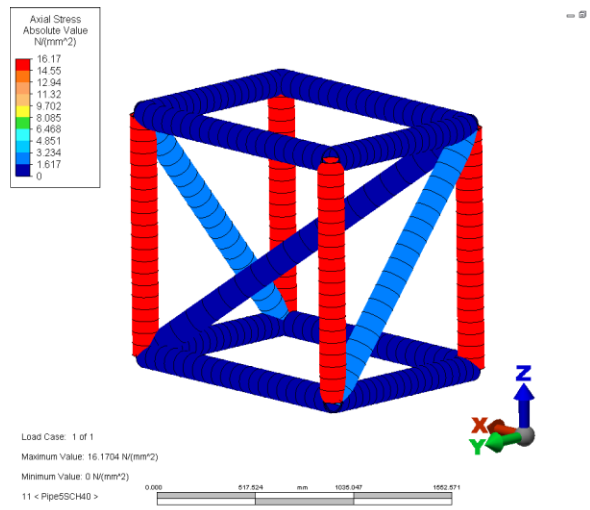

Figure 6: Axial Stress results for Pipe 5 Schedule 40 (16 MPa max)

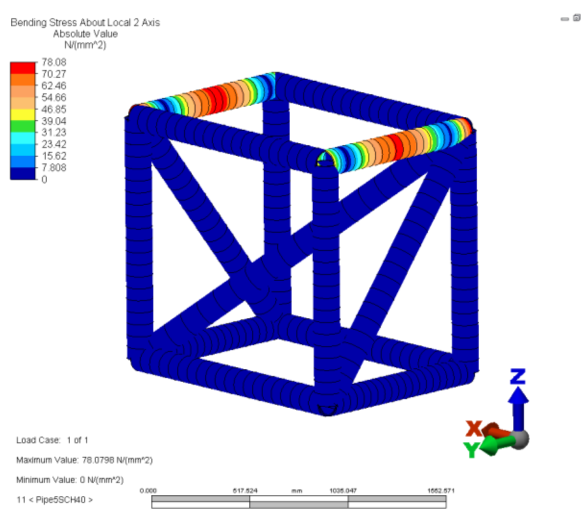

Figure 7: Bending (about local axis 2) Stress results for Pipe 5 Schedule 40 (78 MPa max)

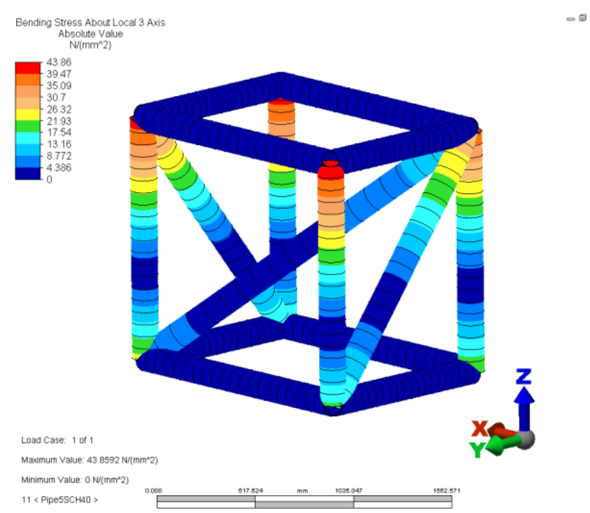

Figure 8: Bending (about local axis 3) Stress results for Pipe 5 Schedule 40 (44 MPa max)

Case 2:

As the FOS for several beam members is well above 2, a secondary analysis was performed in order to optimize the design in a view to minimize the weight while still maintaining a FOS of 2 or more in each member. In this analysis, we used the same set of assumptions as identified in case 1. The design was divided in to the 6 distinct regions as shown below.

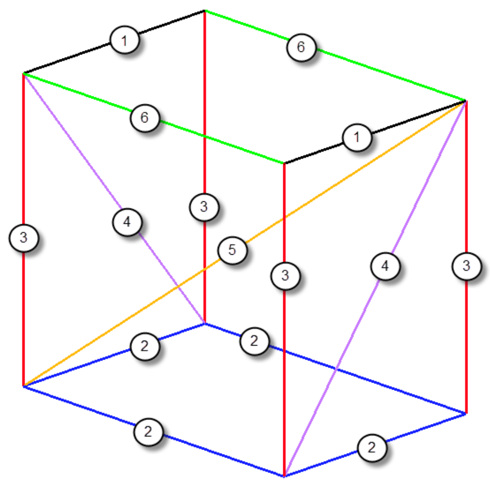

Figure 9: Section numbers to be used in Case 2

Using Autodesk Simulation Mechanical different cross section was designated to each member. Iterative analysis was performed in which smaller cross sections were used for members that did not carry the applied load directly. The final design, which resulted in a FOS greater than 2, is shown in the following table.

Table 3: Optimized Pipe Cross sections

Section number	Section name	Cross sectional area (mm^2)
1	Pipe 5 Schedule 40	2600
2	Pipe 1/2 Schedule 40	148
3	Pipe 2-1/2 Schedule 40	1025
4	Pipe 1/2 Schedule 40	148
5	Pipe 1/2 Schedule 40	148
6	Pipe 1/2 Schedule 40	148

The displacement and stress results obtained using the software for this cross section combination is shown in Figures 10 to 13, which show the magnitudes of maximum displacement and stresses.

Table 4 shows the comparison of FOS vs weight for cases 1 and 2. It is clear that with this second optimization we were able to cut down on the weight quite significantly.

Case 1 stand weighed 4089 N compared to 963 N for case 2. The difference between the 2 cases is 3126 N which is a 76% reduction in weight while keeping the FOS above 2.0

Factor of Safety Calculation:

FOS = Yield Strength / Worst Stress (bending about local 2 axis)
= 250 MPa / 120.1 MPa
= 2.08

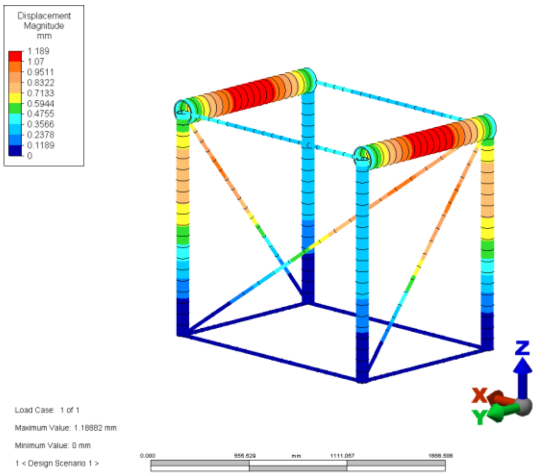

Figure 10: Displacement results for optimized pipe sections (1.2 mm max)

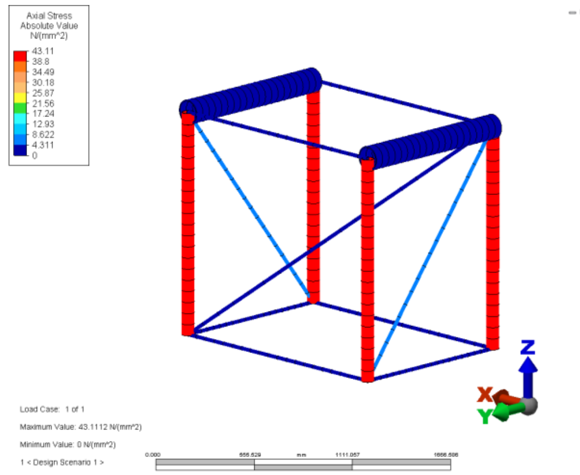

Figure 11: Axial Stress results for optimized pipe sections (43 MPa max)

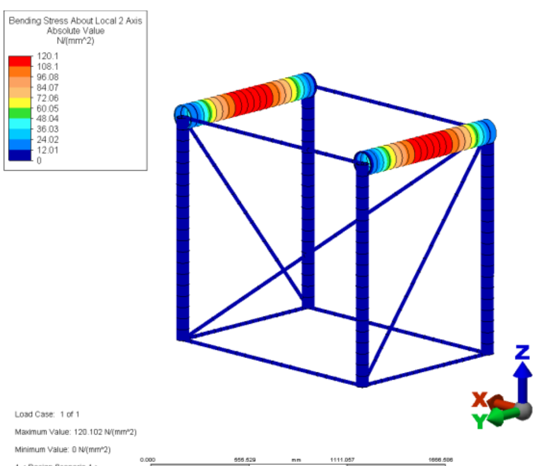

Figure 12: Bending (about axis 2) Stress results for optimized pipe sections (120 MPa max)

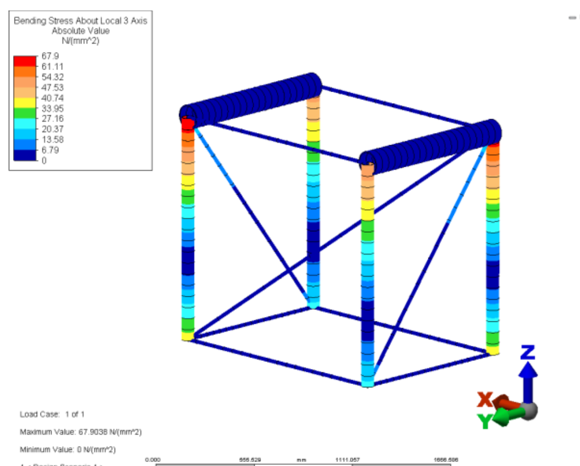

Figure 13: Bending (about local axis 3) Stress results for optimized pipe sections (70 MPa)

Table 4: Comparison of Factors of Safety versus weight

Case number	Section name	Weight, N	Factor of Safety
1	Pipe 5 Schedule 40	4,089	3.20
2	Mixed cross sections	963	2.08

Once the optimized design was obtained a "Critical buckling load" analysis was performed using Autodesk Simulation Mechanical for the first 5 modes of buckling. Critical buckling load analysis (also known as Eigenvalue buckling analysis) examines the geometric stability of models under primarily axial load. Buckling can be catastrophic if it occurs in the normal use of most products. Once the geometry starts to deform, it can no longer withstand even a fraction of the initially applied force.

The load multiplier necessary to the model to buckle as well as the mode shape of buckling (displacements are exaggerated to better show the direction of buckling) is shown in in Figures 14 to 18.

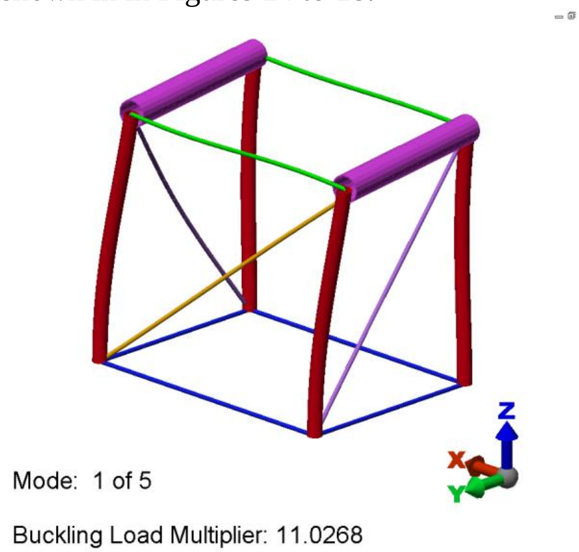

Figure 14: Critical Buckling Load Analysis – Mode 1

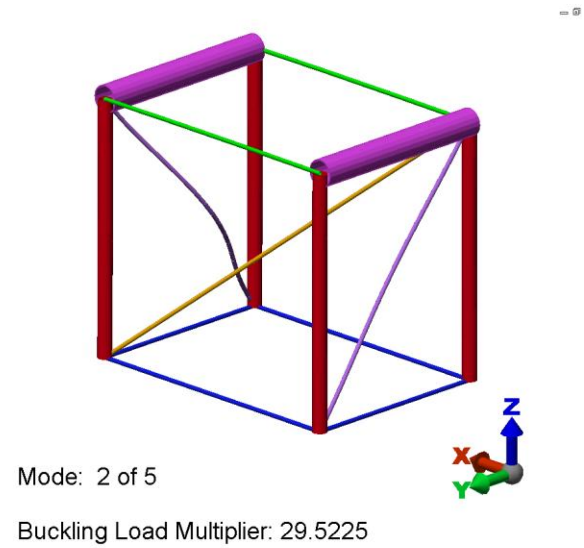

Figure 15: Critical Buckling Load Analysis – Mode 2

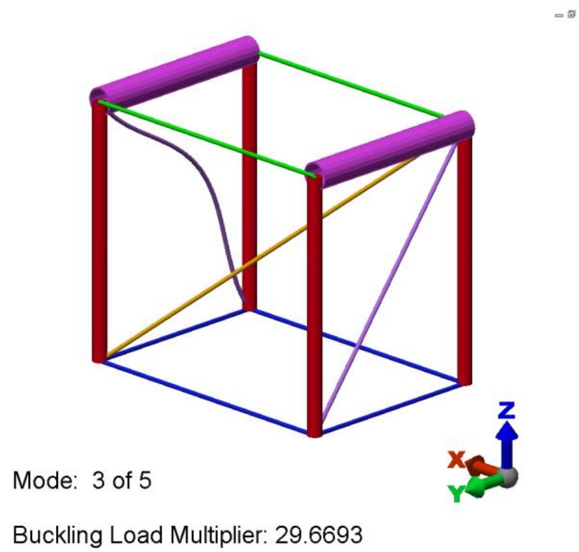

Figure 16: Critical Buckling Load Analysis – Mode 3

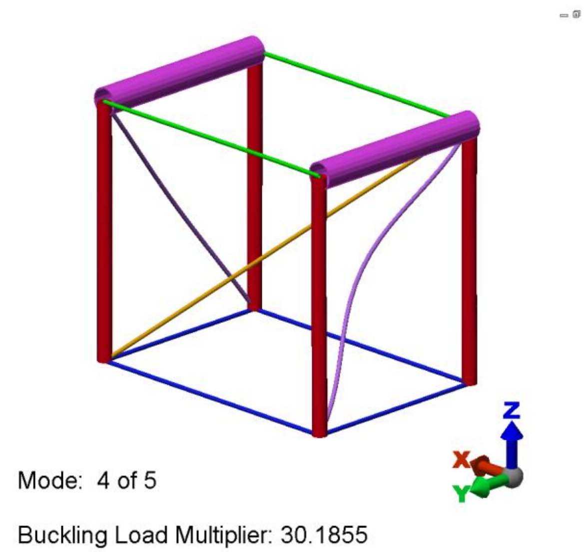

Figure 17: Critical Buckling Load Analysis – Mode 4

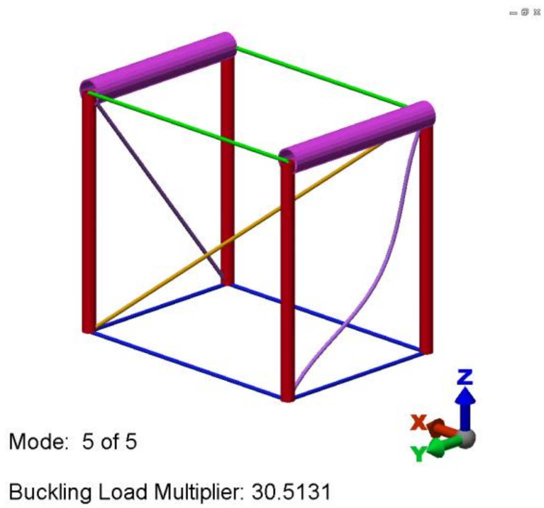

Figure 18: Critical Buckling Load Analysis – Mode 5

Mode 1 is the shown in the above figures has the smallest Critical buckling load multiplier. The value of 11 shown in the analysis results means this structure can withstand up to 11 times the applied load before it goes in to a buckling scenario assuming it can withstand the loads without yielding. However it was already established that this structure would yield if we were to double the applied load as shown in **Table 4**.

Based on these analyses, it is safe to assume that this design is structurally safe for the operating conditions described above and it is optimized for weight.

Case 3:

There are several other designs considered for the car stand including solid rectangle and T-sections. The one shown in Figure **19** uses I-section with one of its axis oriented in different directions. Based on the same set of load and boundary conditions used for the pipe section structural analysis was carried out using a different CAE tool (I-DEAS). Stress and displacement for various members was plotted using 1D beam elements. Sample plot for stress generated is shown in Figure 20.

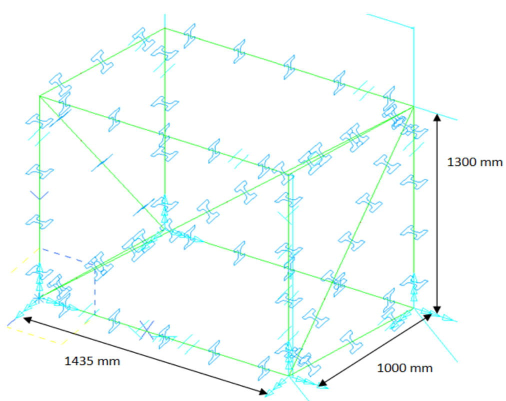

Figure 19: Car-truck stand using standard I-beams

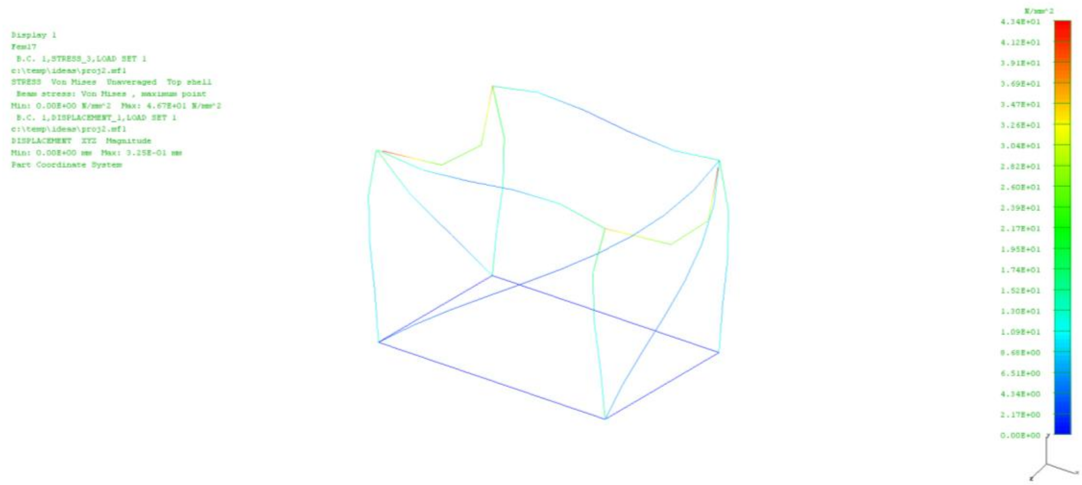

Figure 20: Stress generated (46.7 MPa maximum)

Findings:

Maximum stress generated is 46.7 MPa.
Maximum displacement is 0.325 mm.

Factor of Safety Calculation:

Yield strength of Structural steel ASTM A36 steel is 250 MPa.
Maximum stress generated is 46.7 MPa.
Thus, the factor of safety FOS=250/46.7=5.35 is higher than our targeted value of 2.

Conclusions and recommendations for further studies:

In this paper, analysis and optimal design of a frame is carried out using a CAE tool. One of the applications of this frame is a commercial car truck stand using by the railway car companies. Optimization of members is carried using a pipe section with different standard sizes. It is concluded for this particular frame that if we are to limit our design to a single pipe cross section, the best cross section to select is Pipe 5 Schedule 40. Using this cross section, the design will be safe under the applied loading for the permissible stress and deformation limits. Buckling analysis was also carried out using the CAE tool.

There are several student learning outcomes that can be documented including the following:

- Modeling real structures using 1D finite elements has several limitations.
- Modeling welding at the joints using 1D analysis is not possible.
- Optimization of structural members is possible using a CAE tool.

- Optimization of real car truck stands using 3D geometry (such as the one shown in Figure 1) using a CAE tool is also possible but will require a lot of time for setting up and to post process the results.
- Buckling analyses can easily be carried out using a CAE tool and is necessary to understand all forms of possible failure in a given design.

Based on the above student learning outcomes, several recommendations can be made:

- Perform finite element modeling calculations (for example using stiffness method) by a math tool (such as MatLab or Maple) to compare the results from CAE results.
- Explore other designs for the test stand to determine of other optimum designs can be found.
- Study the effect of inclined loads in 3D to simulate combined axial, bending and torsion loads, in addition to buckling.
- Explore opportunities to carry out design of experiments (DOE).
- Model the frame as a truss (2-force members) using 1D elements to compare the results of beam versus truss elements and with those from the math tool.
- Perform 3D analysis of the optimized structure and compare the results with 1D element analysis results.
- Explore possibilities to use symmetry in modeling.
- Explore the possibility to perform structural optimization of the actual stand.
- Use other CAD and CAE programs and compare the results between the different programs.

Acknowledgements:

The authors would like to acknowledge the support provided by Autodesk Company in donating the software suite to Kettering University.

References:

1. John Clark and Raghu Echempati, “Design and Analysis of a Car-Truck Stand (Paper No. IMECE2011-63902)”, Proceedings of ASME IMECE 2011. Conference held in Denver, CO, November 2011.
2. Logan, D. L., “A First Course in the Finite Element Analysis”, 5th edition, Cengage Learning, 2011.
3. Carroll, W. F., “A Primer for Finite Elements in Elastic Structures”, John Wiley & Sons, 1999.
4. Technical Data for Railway Track Gauges. http://www.rrtools.com/Gauges/ThirdRail.asp

Appendix – A

Table A1: Factor of Safety for all thirty seven Pipe cross sections analyzed

Cross section	Schedule	Weight, N	FOS
Pipe 1/2	40	233.36	0.0257004
Pipe 3/4	40	314.53	0.0444069
Pipe 1	40	466.72	0.0832739
Pipe 1-1/4	40	629.06	0.146381
Pipe 1-1/2	40	760.95	0.203368
Pipe 2	40	1,014.60	0.345901
Pipe 2-1/2	40	1,613.20	0.658078
Pipe 3	40	2,110.40	1.05309
Pipe 3-1/2	40	2,546.70	1.44893
Pipe 4	40	3,013.40	1.92605
Pipe 5	40	4,088.90	3.20185
Pipe 6	40	5,296.20	4.86644
Pipe 8	40	7,964.60	9.19531
Pipe 10	40	11,262.00	15.5503
Pipe 12	STD	13,799.00	21.4977
Pipe 1/2	80	304.38	0.0303869
Pipe 3/4	80	415.99	0.0541442
Pipe 1	80	608.76	0.101804
Pipe 1-1/4	80	842.12	0.183373
Pipe 1-1/2	80	1,014.60	0.258089
Pipe 2	80	1,410.30	0.456244
Pipe 2-1/2	80	2,140.80	0.828195
Pipe 3	80	2,871.30	1.36477
Pipe 3-1/2	80	3,490.20	1.90663
Pipe 4	80	4,200.50	2.57851
Pipe 5	80	5,803.50	4.38244
Pipe 6	80	7,995.10	7.08687
Pipe 8	80	12,074.00	13.4854
Pipe 10	80	15,219.00	20.5397
Pipe 12	XS	18,161.00	28.1654
Pipe 2	XXS	2,546.70	0.70263
Pipe 2-1/2	XXS	3,865.60	1.26843
Pipe 3	XXS	5,235.40	2.14827
Pipe 4	XXS	7,751.60	4.17751
Pipe 5	XXS	10,856.00	7.28763
Pipe 6	XXS	14,915.00	11.8219
Pipe 8	XXS	20,292.00	21.0752