International Journal of Mechanical and Industrial Technology **ISSN 2348-7593 (Online)**
Vol. 4, Issue 2, pp: (51-66), Month: October 2016 - March 2017, Available at: www.researchpublish.com

Correlation of 1D & 2D Finite Element Modeling Under Non-Linear Implicit Analysis

Madasamy Panneerselvam

Wayne State University, Detroit, MI, USA, Caresoft Global Inc., Burr Ridge, IL, USA

Abstract: **The Project involves nonlinear implicit analysis of the Roll over protective structure of a tractor using 1D Elements like BEAM, BAR, SPOTWELD instead of conventional 2D Shell elemental model. Normally, the CAD model is represented using 2D Elements in FEM, Due to that the pre-processing time and final computation is very high, 1D Element representation will be faster to build and simulation can be done lot quicker. The objective to run complicated nonlinear implicit analysis on vehicle structure using both 1D Model & traditional 2D model thereby capturing displacement, stress, strain, reaction force load-displacement plots, and energy-time plots and correlate the results. If the result correlation is good, then this new way of FE Modeling will help in reducing the lead time of these kind of analysis.**

The energy - time plots collected give us an idea about the amount of energy the structure could absorb during the roll over. Similarly the force displacement plots can give information about the load carrying capacity of the cab structure. The correlation between the simulation results of the 1D & 2D finite elemental model was compared, the results showed very good correlation between these 2 models.

Keywords: **Roll Over Protective Structure, Nonlinear Implicit Analysis, Hypermesh, and Abaqus.**

I. INTRODUCTION

ROPS, or Rollover Protective Structure, is a cab or frame that provides a safe environment for the tractor operator in the event of a rollover. The ROPS frame must pass a series of static and dynamic crush tests. These tests examine the ability of the ROPS to withstand various loads to see if the protective zone around the operator station remains intact in an overturn.

The ROPS must meet standards, such as those set forth by the American Society of Agricultural Engineers, which certify they provide adequate protection in a tractor upset. If the ROPS is certified, there will be a certification label on the unit. Standards.

Manufacturers have designed and tested ROPS to meet specific standards developed by the Society of Automotive Engineers (SAE), the American Society of Agricultural Engineers (ASAE)—now called the American Society of Agricultural and Biological Engineers (ASABE)—and other organizations. These standards indicate that a ROPS has passed specially designed crush, static, and dynamic tests that confirm its effectiveness.

ROPS must meet the following standards:

- SAE J1040
- OSHA 1928.51

II. SOFTWARE USED IN THIS PROJECT

- Abaqus 6.9 as the solver
- Hypermesh Version 14.0 as the preprocessor
- HyperView & Hypergraph as the post processor
- Windows operating system

III. THE ROPS REQUIREMENT

According to SAE J1040, three steps are needed for the ROPS of 8850 kg gross vehicle weight. The loads are applied according the sequence listed below.

Step 1 (Lateral load):

F = 6 * M =53100 N,

Step 2 (Vertical crushing load):

F = 19.61 * M = 173550 N

Step 3 (Longitudinal Load):

F = 4.8*M = 42480 N.

IV. PROCEDURE

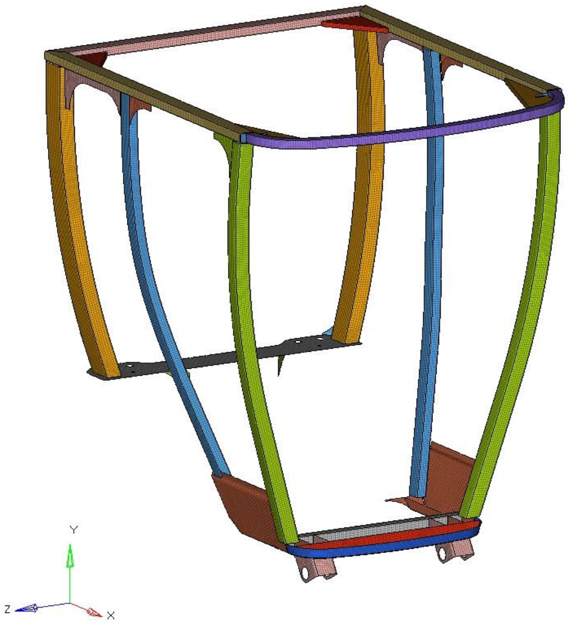


Fig 1. Finite Element Model of ROPS – 2D Shell

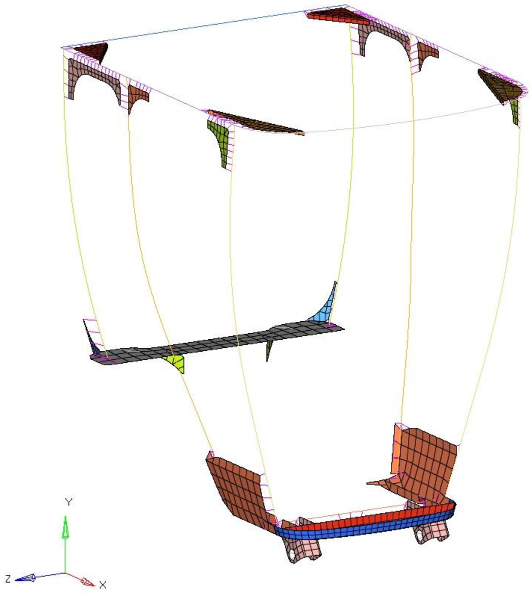


Fig 2. Finite Element Model of ROPS – 1D Beam, Bar & Spot – Weld Elements

V. ASSUMPTIONS

- The analysis conducted is a quasi-static analysis
- The material used for different parts are considered isotropic and homogeneous
- The geometry of the model and boundary conditions were already provided.

VI. ANALYSIS PROCEDURE

FE Model for both 1D & 2D was developed in Hypermesh. Abaqus/standard is used in the nonlinear calculation.

Step 1: Lateral Loading

The lateral cross member of the ROPS is displaced by the pusher in the lateral direction till the force applied on the structure is equivalent to 53100 N. Once this force is applied, the model is checked if Stress & Strain limit is within allowable limit. Also if there is intrusion inside deflection limiting volume.

Step 2: Vertical Loading

The lateral loading is released & the vertical cross members of the ROPS is displaced by the pusher in vertical downward direction till the force applied on the structure is equivalent to 173550 N. Once this force is applied, the model is checked if Stress & Strain limit is within allowable limit. Also if there is intrusion inside deflection limiting volume.

Step 3: Rear Longitudinal Loading

The vertical loading is released & rear cross member of the ROPS is displaced by the pusher in the lateral direction till the force applied on the structure is equivalent to 42480 N. Once this force is applied, the model is checked if Stress & Strain limit is within allowable limit. Also if there is intrusion inside deflection limiting volume.

A) Contact:

Many engineering problems involve contact between two or more components. In these problems a force normal to the contacting surfaces acts on the two bodies when they touch each other. The general aim of contact simulations is to identify the areas on the surfaces that are in contact and to calculate the contact pressures generated. In a finite element analysis contact conditions are a special class of discontinuous constraint, allowing forces to be transmitted from one part of the model to another. The constraint is discontinuous because it is applied only when the two surfaces are in contact. When the two surfaces separate, no constraint is applied. The analysis has to be able to detect when two surfaces are in contact and apply the contact constraints accordingly. Similarly, the analysis must be able to detect when two surfaces separate and remove the contact constraints.

The two Contact Cards used in this model are:

B) Automatic Single Surface Contact:

Single surface contact is established when a surface of one body contacts itself or the surface of another body. In single surface contact, the LS-DYNA program automatically determines which surfaces within a model may come into contact. Therefore, single surface contact is the simplest type to define because no contact or target surface definitions are required. When it is defined, single surface contact allows all external surfaces within a model to come into contact. This option can be very powerful for self-contact or large deformation problems when general areas of contact are not known beforehand. Unlike implicit modeling, where over-defining contact will significantly increase computation time, using single surface contact in an explicit analysis will cause only minor increases in CPU time. Most impact and crash-dynamic applications will require single surface contact to be defined. Since automatic general (AG) contact is very robust and includes shell edge (SE) contact as well as improved beam contact, it is recommended as the first choice for self-contact and large deformation problems when the contact conditions are not easy to predict.

C) Automatic Node-to-Surface:

Node-to-surface contact is a contact type, which is, established when a contacting node penetrates a target surface. This type of contact is commonly used for general contact between two surfaces.

The flat or concave surface is the target. The convex surface is the contact surface.

The coarser mesh is the target surface. The finer mesh is the contact surface.

D) Automatic Contact:

Along with the general contact family, the automatic contact options are the most commonly used contact algorithms. The main difference between the automatic and general options is that the automatic contact algorithms automatically determine the contact surface orientation for shell elements. In automatic contact, checks are made for contact on both sides of shell elements.

***E) Defining contacts b/w the Pusher & ROPS Structure**:*

Contacts are defined between the pusher and the Cross member. Without contacts the pusher may not recognize the presence of the Cross member. Since the Cross member deforms penetration between its own surfaces can occur which is not desirable. To prevent these problems contacts have to be defined. Two types of contact surfaces are provided. One is “nodes to surface” and the other is “single surface” type of contacts. Here the pusher is considered as a rigid body and hence it is made the master surface. The Cross member is considered as a slave surface. The second type of contact is called single surface contact used for the Cross member surface. This type of contact prevents the surface of the Cross member to overlap and penetrate onto its own surface. Both the types of contacts are of automatic type. Use of the automatic card adjusts the normal of the contact surfaces to face in opposite direction to each other even if they are not.

F) Contact Control Cards:

The following contact control Cards were defined to stabilize the contact divergence

*CONTACT CONTROLS, MAXCHP=0.1, PERRMX=0.09 UERRMX=0.095

*CONTACT CONTROLS, AUTOMATIC TOLERANCES

*CONTACT CONTROLS, RESET

Database Option:

The following database cards were defined to obtain the output from the analysis. The output in ASCII format.

*NODE FILE, FREQUENCY, U – Nodal Displacement Output

*EL FILE, FREQUENCY, S – Stress Elemental Output

*EL FILE, FREQUENCY, E– Strain Elemental Output

*NODE PRINT, NSET, RF – Reaction Force Output Plot

*NODE PRINT, NSET, CF– Total Force Output Plot

Boundary conditions are defined such that the pinned connection on all 4 mounts (we allowed the rotation and fixed for translation degrees of freedom.

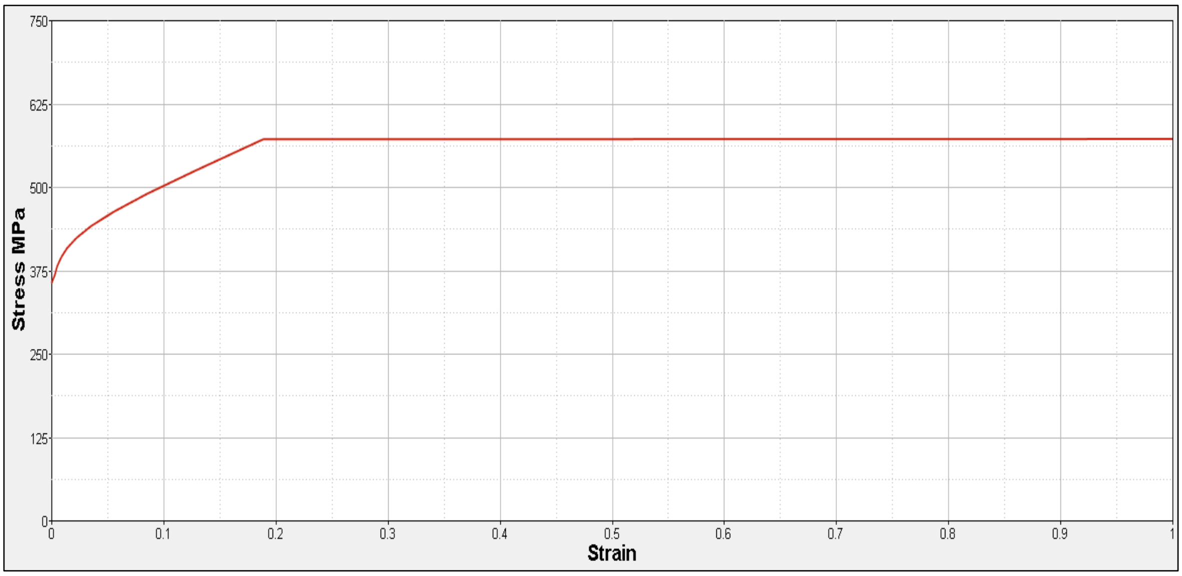
•Abaqus input deck is created and analysis is run

TABLE 1: COMPONENT DESCRIPTION

Name of the component	Gauge (mm)	Poisson’s ratio	Young’s modulus (MPa)	Yield strength (MPa)	Density Tonne/mm3
Front Pillar	5.0	0.3	210000	335	7.83e-9
Rear Pillar	4.0	0.3	210000	335	7.83e-9
Side Pillar	4.0	0.3	210000	335	7.83e-9
Longitudinal Crossbeam	4.0	0.3	210000	335	7.83e-9
Bracket	4.0	0.28	207000.0	210	7.83e-9
Gusset Plates	5.0	0.28	207000.0	210	7.83e-9
Bottom Support	8.0	0.28	207000.0	210	7.83e-9

TABLE 2: MATERIAL PLASTIC STRAIN GRPAHS FOR ABOVE COMPONENTS

Strain	0	0.002973	0.00507	0.013846	0.022316	0.035422	0.055364	0.08515	0.128696	0.188931	1
Stress MPa	355.6001	368.4637	381.067	408.2937	423.8022	441.5873	462.922	489.7338	524.9174	571.725	572

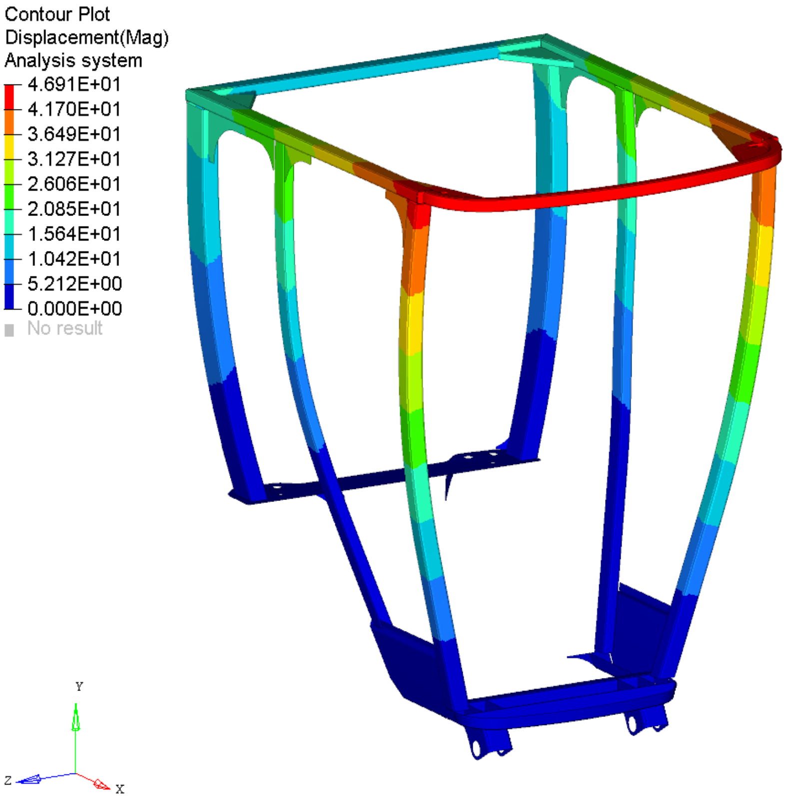


GRAPH 1. Stress Strain Curve – Plastic

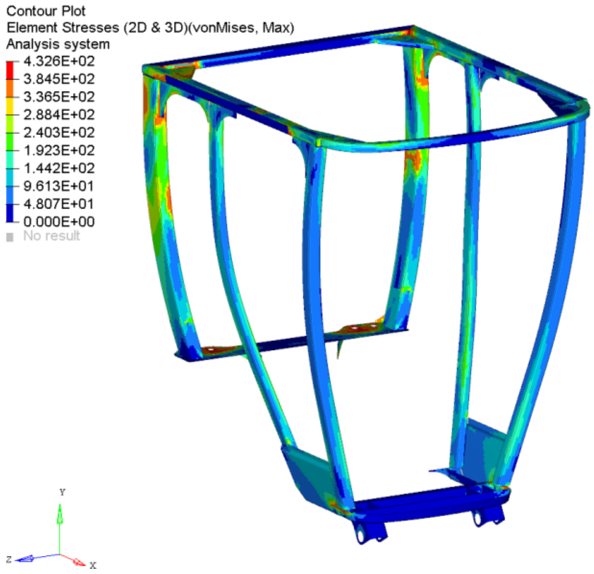
VII. RESULTS OF 2D FINITE ELEMENT MODEL

A) Lateral Loading Condition:

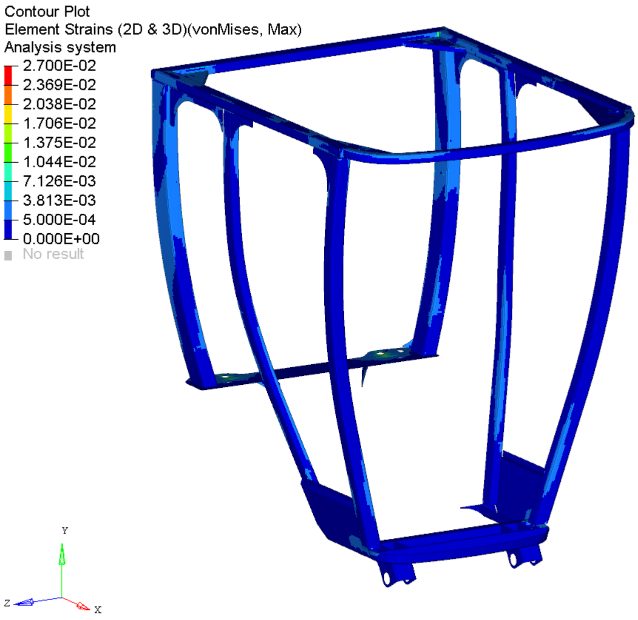
i) **Displacement Contour**



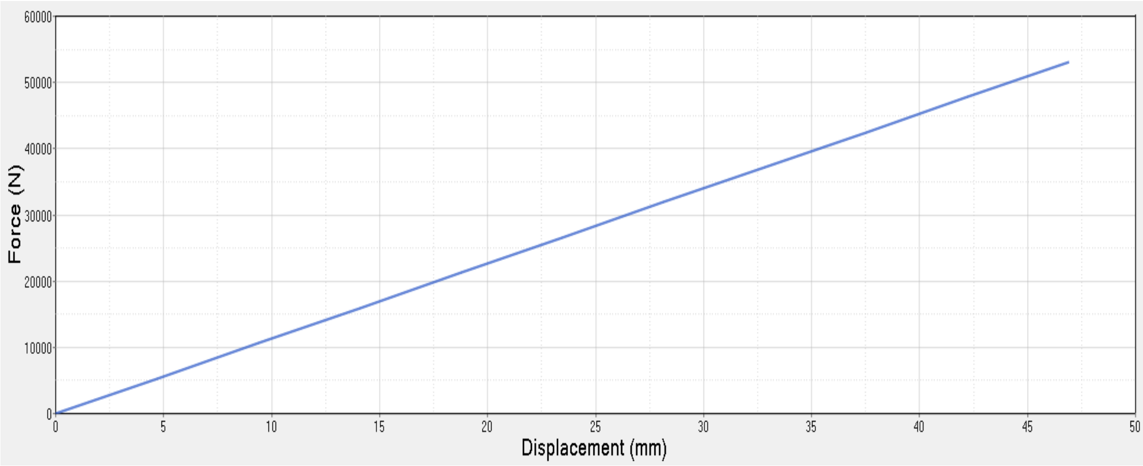
ii) **Stress Contour**



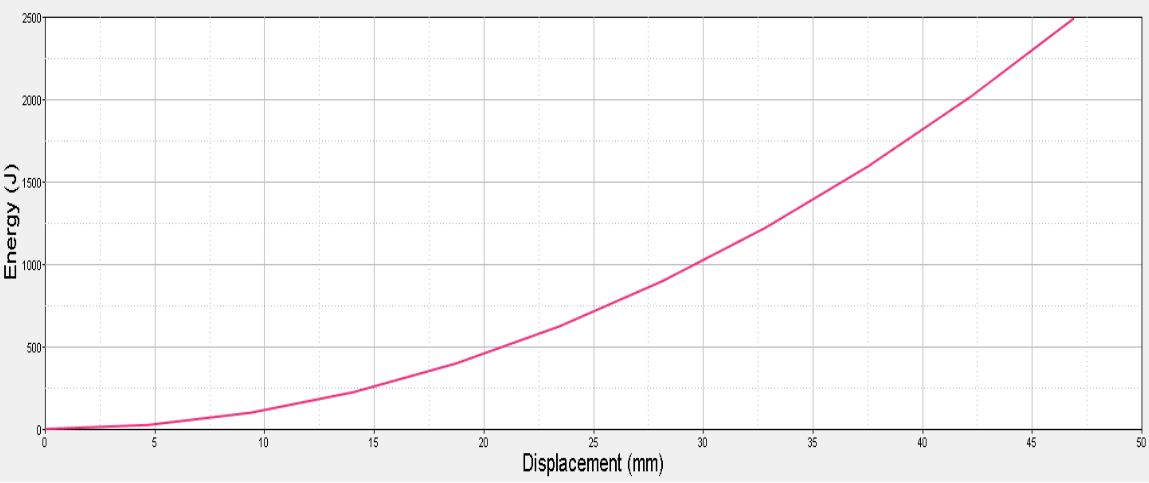
iii) **Strain Contour**



iv) **Lateral Force in N Vs Lateral Deflection in mm**

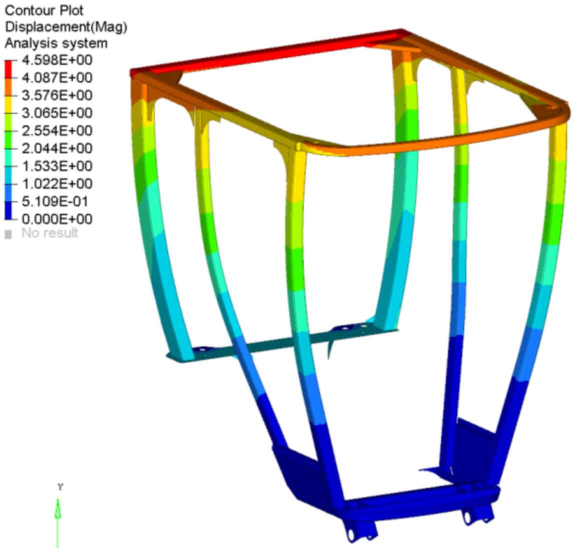


v) Energy in J Vs Lateral Deflection in mm

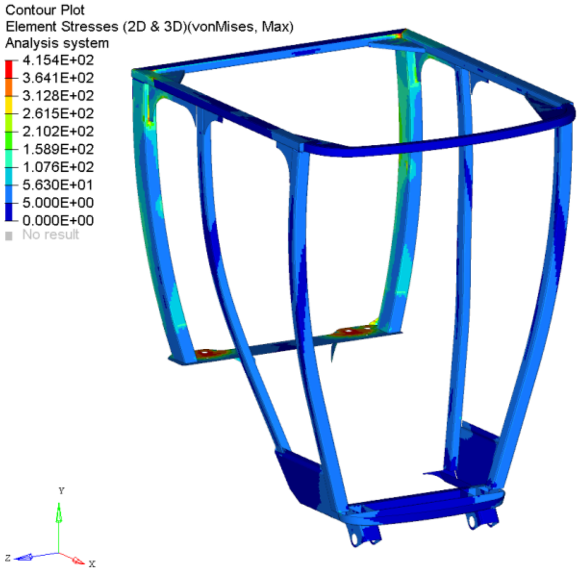


B) Vertical Crushing Loading Condition:

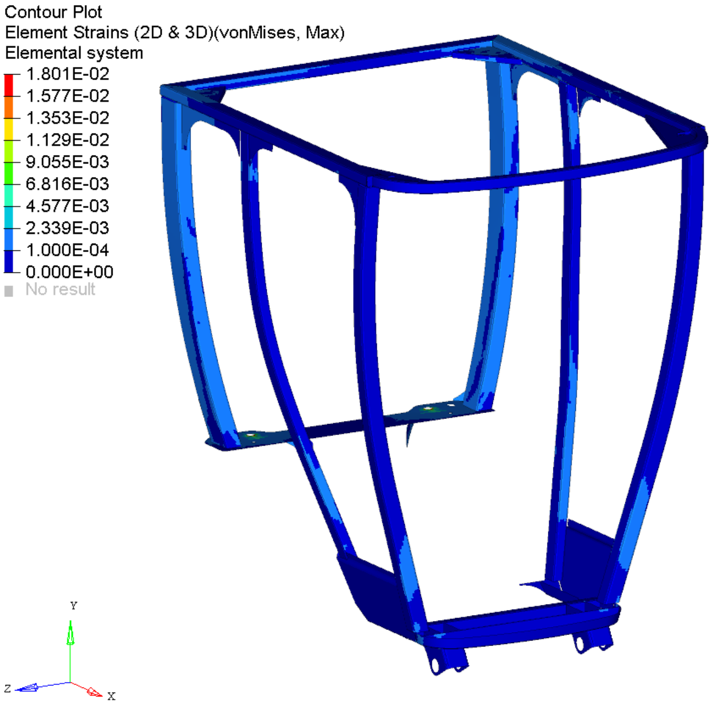
i) Displacement Contour



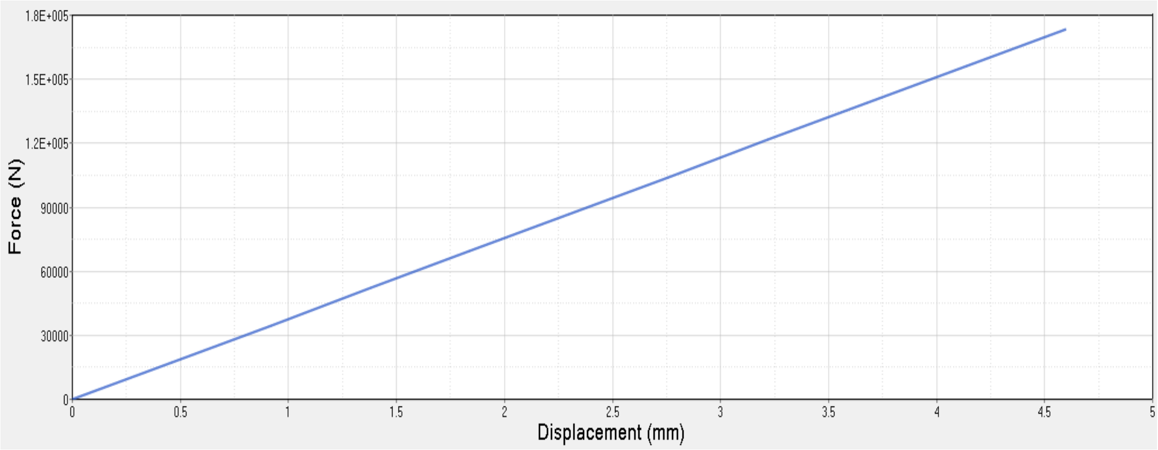
ii) Stress Contour



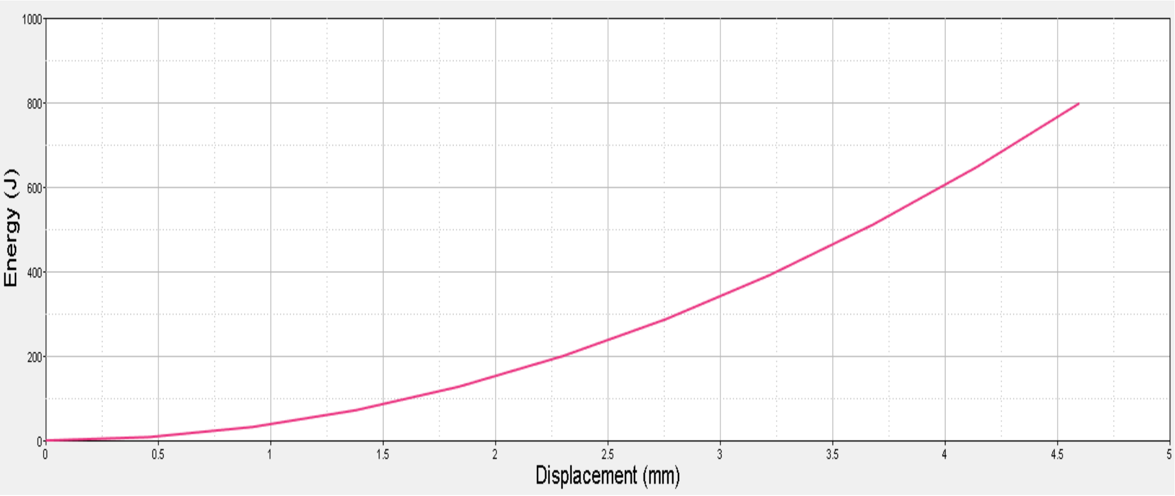
iii) Strain Contour



iv) Vertical l Force in N Vs Vertical Deflection in mm

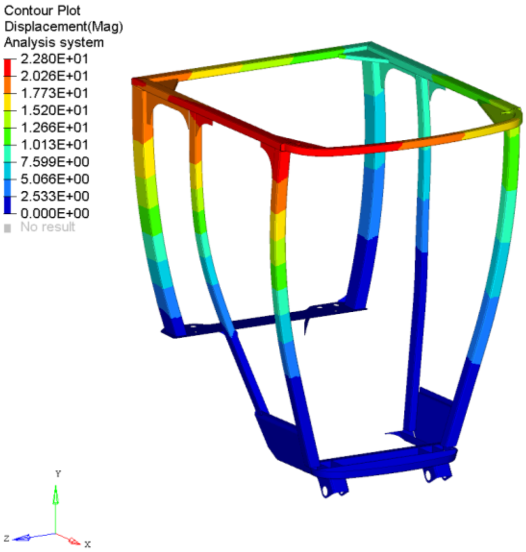


v) Energy in J Vs Vertical Deflection in mm

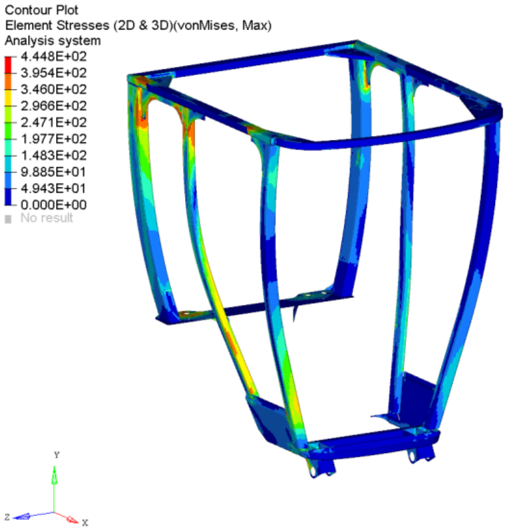


C) Rear Longitudinal Loading Condition:

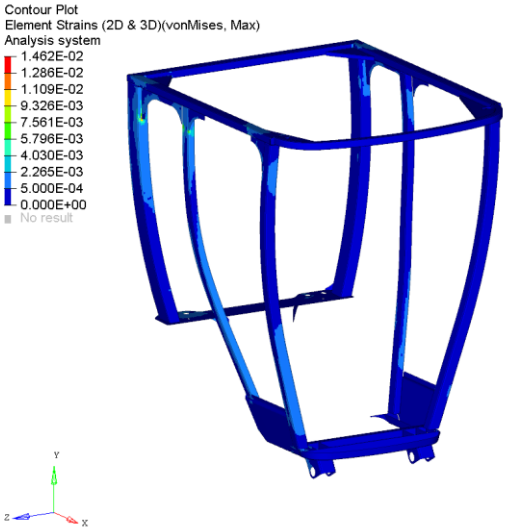
i) Displacement Contour



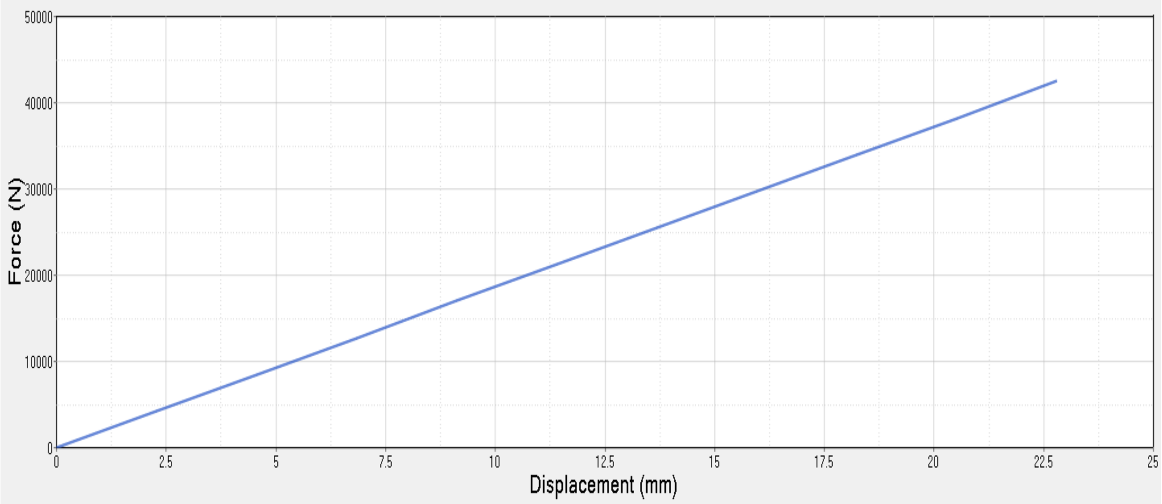
ii) Stress Contour



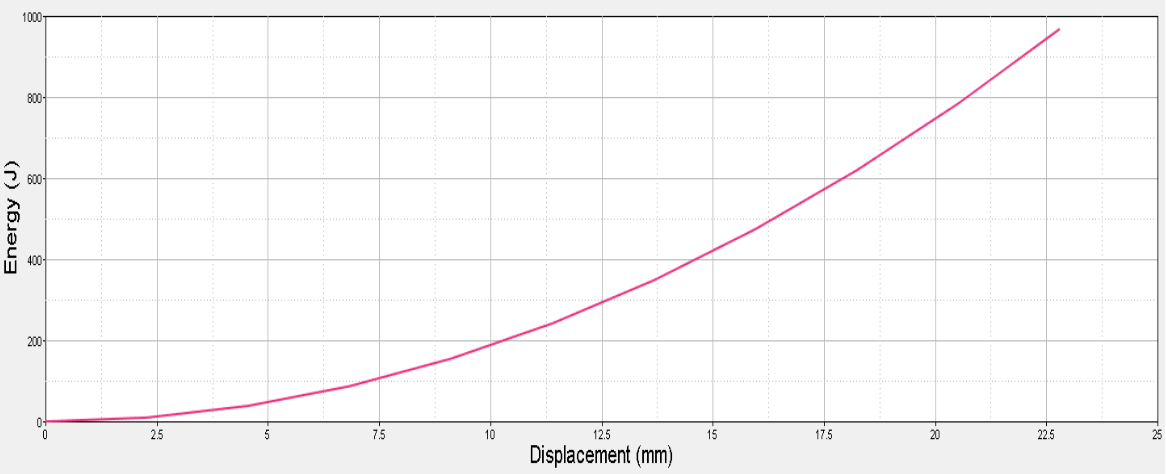
iii) Strain Contour



iv) Longitudinal Force in N Vs Longitudinal Deflection in mm



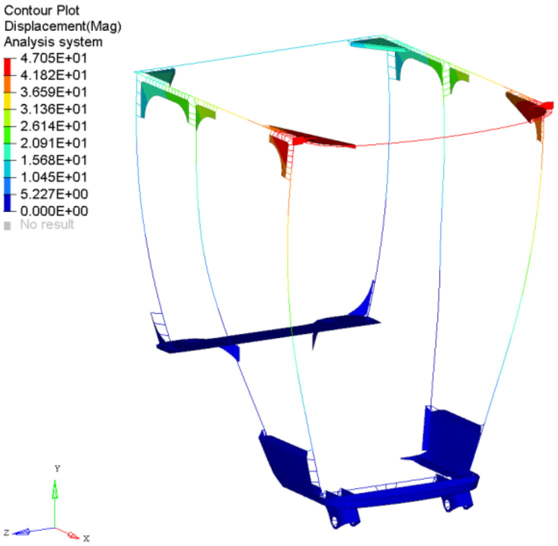
v) Energy in J Vs Longitudinal Deflection in mm



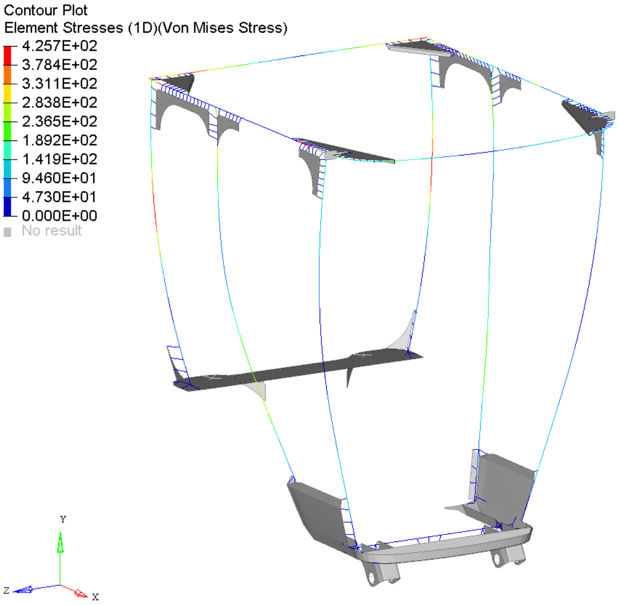
VIII. RESULTS OF 1D FINITE ELEMENT MODEL

A) Lateral Loading Condition:

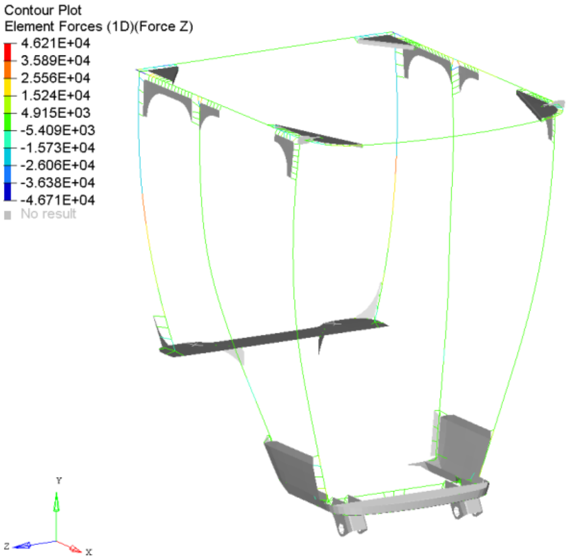
i) Displacement Contour



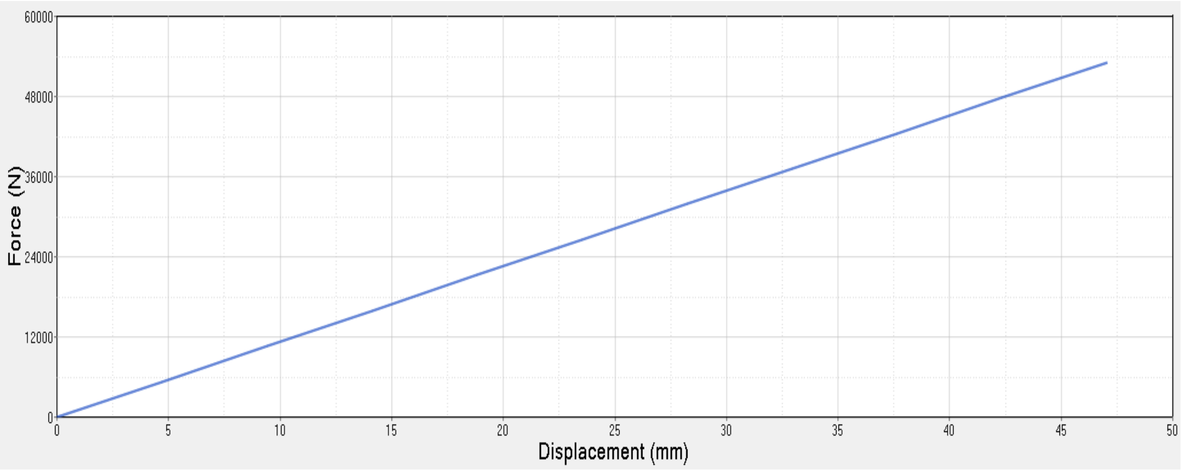
ii) Stress Contour



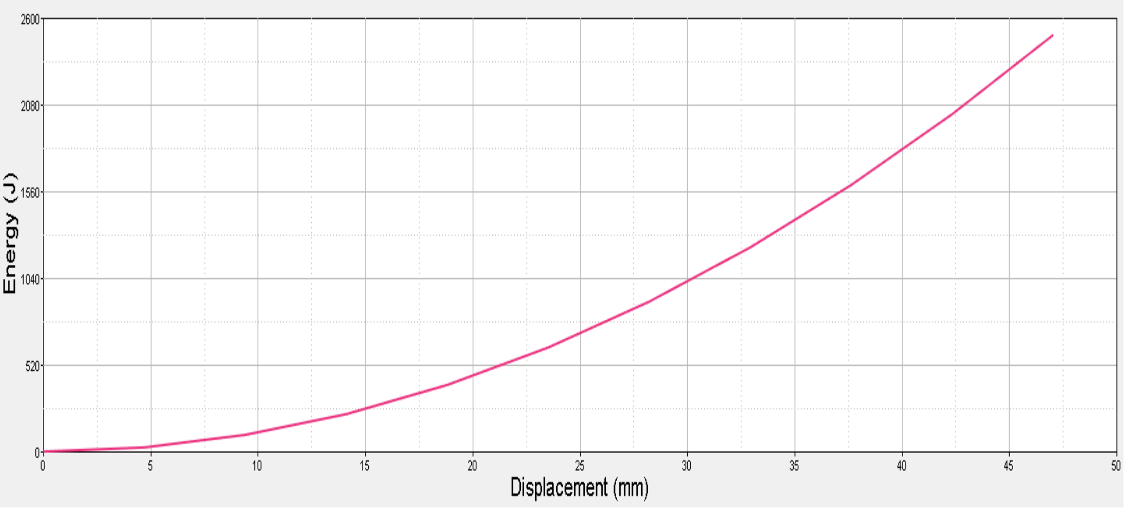
iii) Elemental Force Contour



iv) Lateral Force in N Vs Lateral Deflection in mm

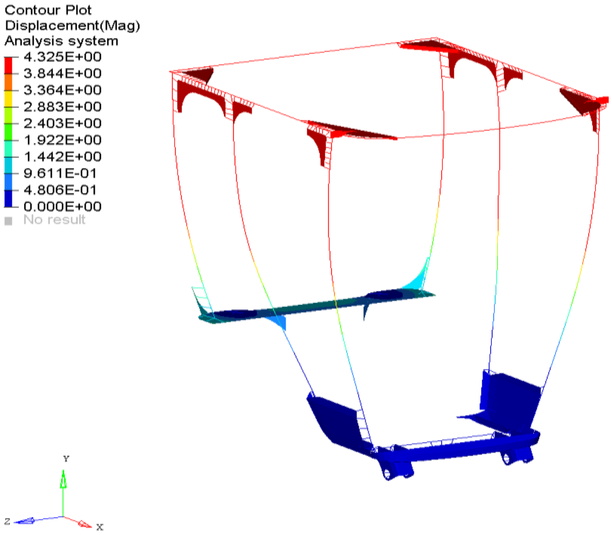


v) Energy in J Vs Lateral Deflection in mm

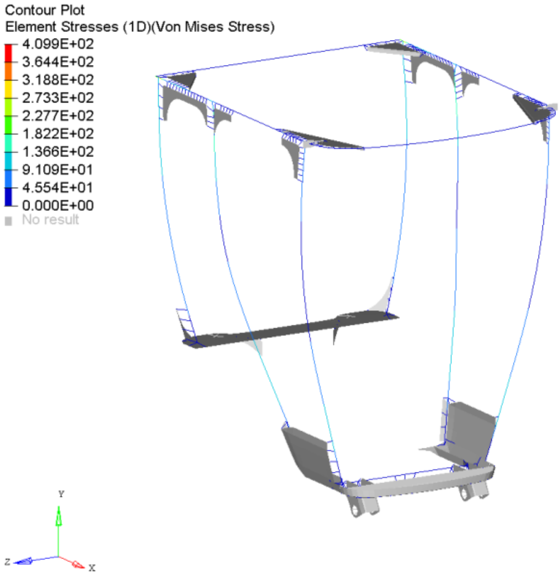


B) Vertical Crushing Loading Condition:

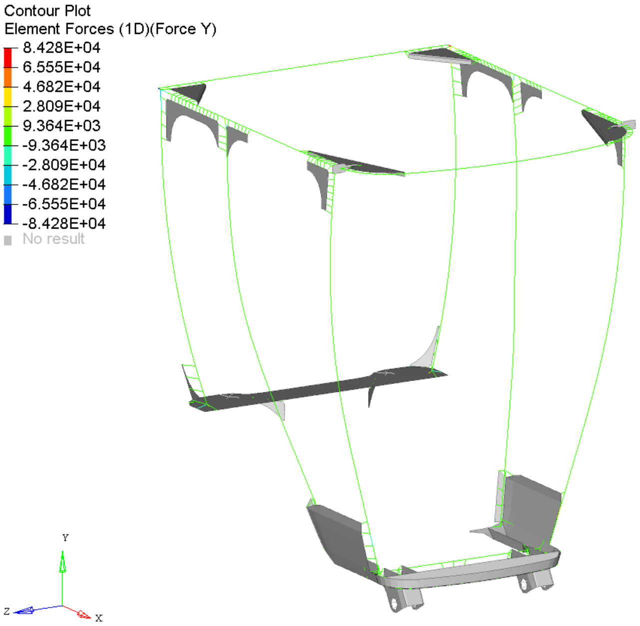
i) Displacement Contour



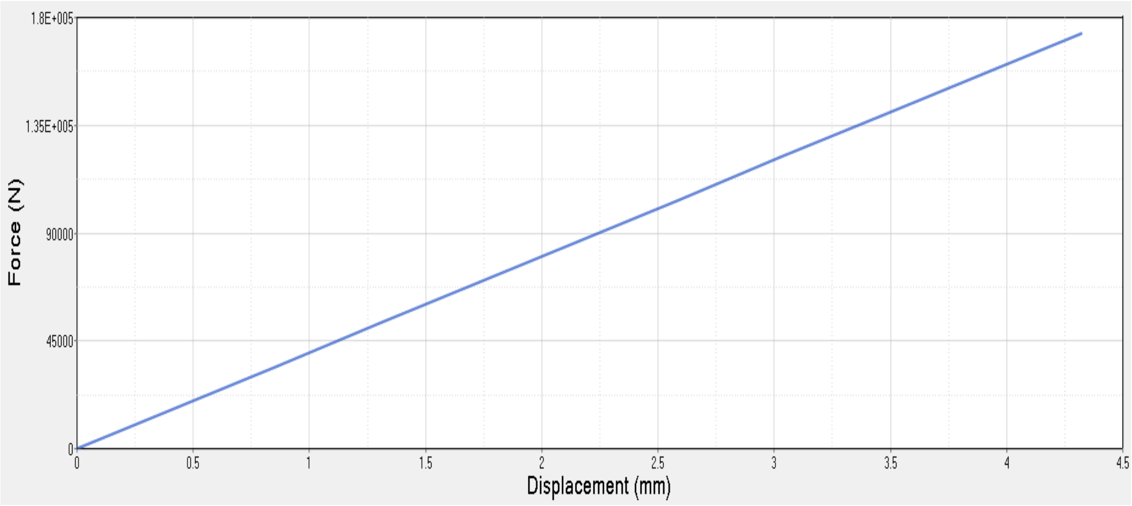
ii) Stress Contour



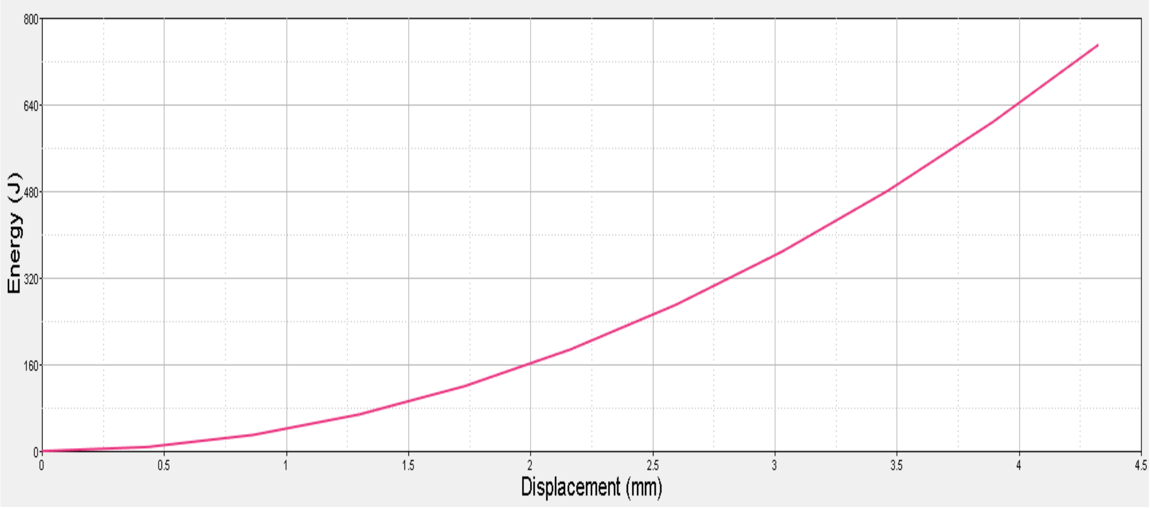
iii) Elemental Force Contour



iv) Vertical Force in N Vs Vertical Deflection in mm

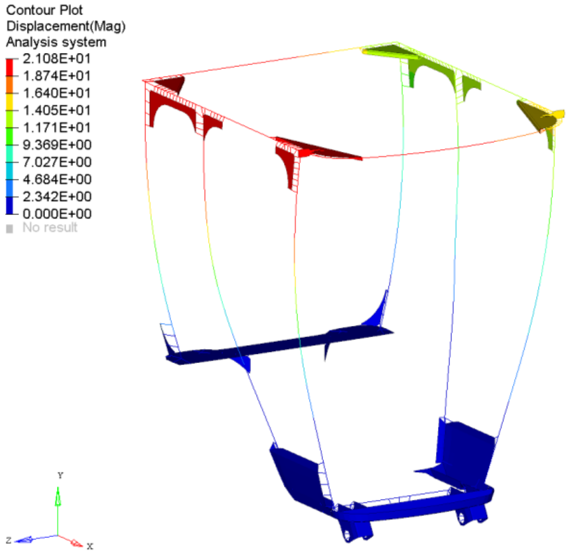


v) Energy in J Vs Vertical Deflection in mm

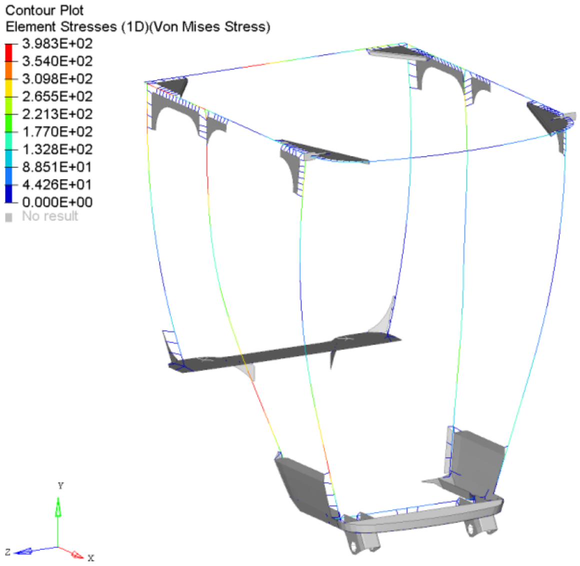


C) Rear Longitudinal Loading Condition:

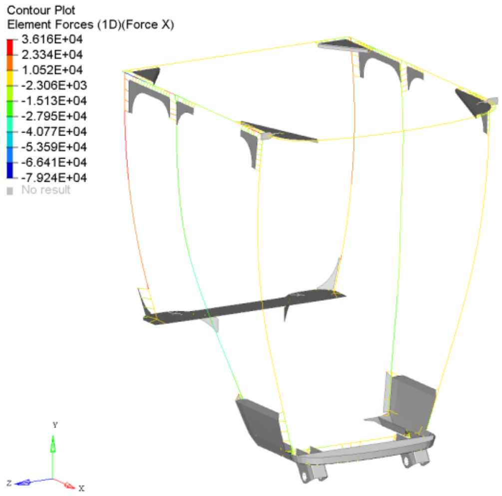
i) Displacement Contour



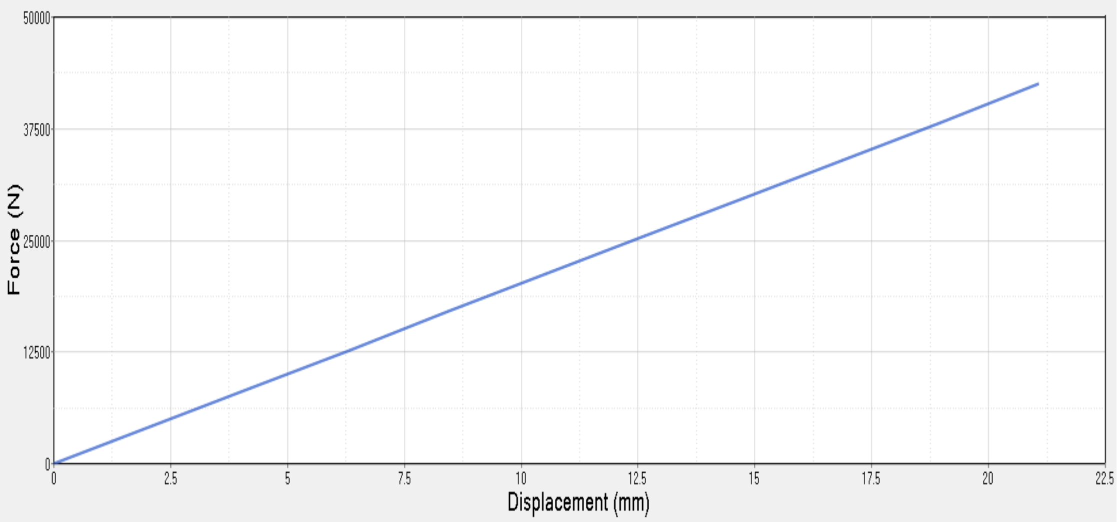
ii) Stress Contour



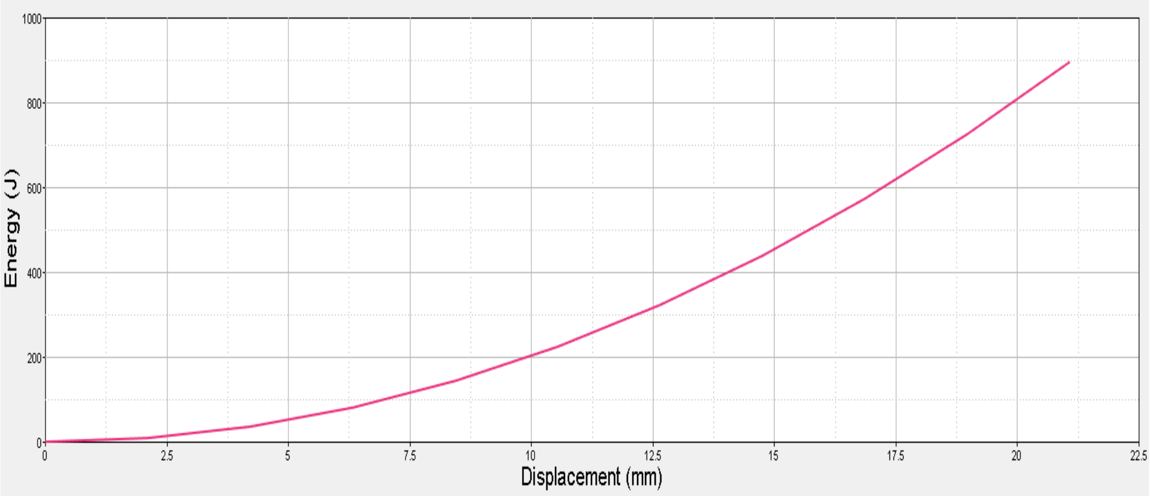
iii) Elemental Force Contour



iv) Longitudinal Force in N Vs Longitudinal Deflection in mm



v) Energy in J Vs Longitudinal Deflection in mm



IX. CONCLUSIONS

From the Displacement, Stress & Strain contour results and from the force Vs displacement plots & Energy Vs Displacement plots, We can clearly conclude that there is good correlation in results between 1D & 2D Finite Element Models. When comparing the Finite element modeling & analysis preprocessing, Solver & post processing time, there is 50% reduction in time. ROPS model has around 400,000 Elements & 1450909 Nodes, if the computation time is reduced to half then for BIW Model of an automotive or Chassis Frame Structure of a truck will have tremendous variation in the simulation & computation. Similar 1D Application can be applied to the 3D Elements, Casting components are hard to be captured exactly but if the flow path of the mesh is imaged internally then multiple 1D elements can been modeled to replicate the 3D mesh.

RESULT PARAMETERS	**Lateral**		**Vertical**		**Longitudinal**	
	2D Model	1D Model	2D Model	1D Model	2D Model	1D Model
Displacement in mm	46.91	47.05	4.598	4.325	22.8	21.08
Stress in MPa	432.6	425.7	415.4	409.9	444.5	398.3
Force Applied in N	53100	53090	173550	173550	42480	42395
Energy Absorbed in J	2512.5	2512.09	819	799	1059.5	989.5

REFERENCES

[1] Nitin S. Gokhale, Sanjay S.Deshpande, Sanjeev V. Bedekar "Practical Finite Element Analysis"

[2] Ahmed Elmarakbi “Advanced Composite Materials for Automotive Applications", Structural Integrity and Crashworthiness.

[3] Drabek, Pavel, Milota, Jaroslav "Methods of Nonlinear Analysis", Applications to Differential Equations.

[4] William F. Milliken, Douglass L.Milliken, Maurice Olley "Chassis Design" Principles and Analysis

[5] Shun-Hsyung Chang, Ivan Parinov, Vitaly Yu. Topolov, Advanced Materials: Physics, Mechanics and Applications

[6] K. K. Choi, Nam-Ho Kim, Structural Sensitivity Analysis and Optimization 2: Nonlinear Systems and Applications

[7] Ronald P. Nielsen, OSHA Regulations and Guidelines: A Guide for Health Care Providers / Edition 1

[8] https://www.osha.gov/pls/oshaweb/owadisp.show_document?p_table=STANDARDS&p_id=10957

[9] http://standards.sae.org/j397_200405/