THE LIMITING ASPECTS OF PRODUCT DESIGN SOFTWARE APPLIED TO THE VIRTUAL PROTOTYPING OF CAR BODY LIFTING END EFFECTORS

FLORENTIN MARIAN ILIE [1], TUDOR GEORGE ALEXANDRU [2]

Abstract: In the automotive industry, the transfer of car bodies among manufacturing systems is performed with the help of industrial robots equipped with lifting end effectors. Such End-of-Arm Tooling is subject to bending loads, demanding an adequate sizing process to be deployed to ensure imposed structural reliability criteria. This paper addresses limiting aspects of the process, with emphasize on computer aided engineering simulations performed as part of product design software suites. A new approach is proposed involving the use of simplified finite element modeling. This allows for the load distribution to be captured precisely with minimal computational expenses.

Keywords: car body lifting, computer aided engineering, virtual prototyping, structural sizing

1. INTRODUCTION

In automotive manufacturing, car body transfer involves lifting cars from one station to another by means of heavy payload tools. In latest assembly lines, the process is fully automated being performed by closed kinematic chain industrial robots equipped with specific end effectors. The frequent changes in manufacturing typologies impose reconfiguring the End-of-Arm tooling to accommodate new car bodies. To address such challenges, product design suites are deployed as comprehensive virtual prototyping technologies. Among the software involved, Computer Aided Engineering (CAE) brings together multiphysics simulations for capturing the behavior of products subject to in-service loading. This allows tooling reconfiguration to be validated from static and dynamic viewpoints, with a minimum resource usage. Even so, the peculiarities of the car lifting end-effectors structure bring

[1] *Ph.D. Eng student, University Politehnica of Bucharest*
[2] *Assistant Professor, Ph.D. Eng student, University Politehnica of Bucharest*

in to discussion finite element modeling issues. From this perspective, the accuracy of simulation results and the involved computational costs can be significantly influenced by adapting the wrong scenarios. This paper proposes a simplified finite element modeling approach that involves dividing the 3D structure in sub-elements that can be approximated by 1D beam or 2D shell assumptions. The linkage between the two different element types is ensured by means of rigid bars. Welded junctions are also taken into account by applying additional constraints to the respective degrees of freedom (DOF). A static analysis is performed to illustrate the given concepts.

2. STRUCTURAL ASPECTS OF CAR LIFTING END EFFECTORS

Car lifting end effectors are in general stiff assemblies comprising structural framing and adjustable fittings. Additional elements are added for attaching the effector to the output flange of the industrial robot. Considering the high mass of the lifted car bodies, multiple sizing criteria have to be verified to ensure structural reliability and safe operation of the assembly. From a static viewpoint, the high bending moments resulting from the lifting process leads to mechanical stress that must be checked against allowable limits. The cyclic loading induced by repeated operation can be a source of fatigue. Thus, the number of cycles until failure requires evaluation. Due to the robot motion and the coexistence of other forced vibration sources, dynamic criteria are verified, such as natural frequencies and harmonic response.

The geometric characteristics of a car lifting end effector are depicted in Figure 1. The notations performed carry the following significance:

- 1 – Structural framing made from high-strength steel alloys;
- 2 – Adjustable fittings;
- 3 – Mounting bracket;
-

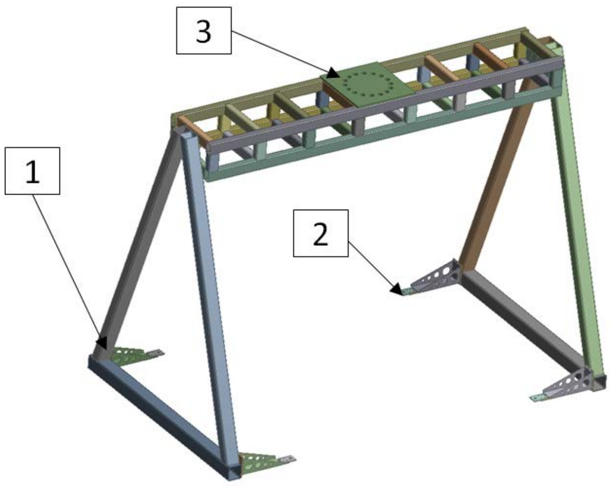

Fig.1 The geometric characteristics of a car-lifting end effector

3. CAE CONSIDERATIONS

CAE software has evolved in the past decades from a highly specialized tool to a Graphical User Friendly (GUI) interfaces that requires minimum engineering knowledge. Even so, structural analysis demands, most of the times, choosing the correct assumptions and simplifying hypothesis for evaluating desired criteria. For example, 3D geometries of prismatic bodies are, in the first stage, simplified by removing small chamfers and holes. The mesh that approximates the resulting model is refined only in areas of interest, such as steep corners or cut-outs.

The structural framing of the car lifting end-effector comprises long beams, with bending stress representing the main source of failure. All other components are thin-walled structures that are welded to the frame, allowing for tension and bending loads to be distributed from the adjustable fittings to the robot output flange. From this perspective, 1D elements provide best accuracy in CAE simulations when modeling beam structures. On the other hand, 2D shell elements achieve a good description of the load distribution, being appropriate for verifying junction safety criteria.

A study is performed to illustrate result accuracy in the calculation of the normal stress for a beam supported at one end and a central single circular hole in a finite-width plate by using different element types, mainly 1D - 3D for beams and 2D – 3D for plates. The simulations are performed by using ANSYS Mechanical APDL. Results achieved are compared with the ones calculated by well-established analytical models:

$$\sigma_{\max} = \frac{n}{I} M_{\max} \tag{1}$$

Where: Equation 1 calculates the normal stress in the beam $\sigma_{\max}$ (MPa)
$M_{\max}$ is the bending moment (Nm)
n is the distance from the neutral axis (mm)
I is the moment of inertia (mm^4)

$$\sigma_{\max} = \frac{F}{A} K_t \tag{2}$$

Where: Equation 2 calculates the normal stress around the hole $\sigma_{\max}$ (MPa)
F is the tension load applied (N)
A is the area normal to the applied load, excluding the hole (mm^2)
K_t is a stress amplification factor derived from experiments.

1D element types have the highest accuracy in estimating beam stresses; error achieved being around 0.11% than compared to the analytical model. On the other hand, 2D and 3D element types achieve comparable result accuracy, as well as stress distribution in the structure, for both 2D and 3D approaches (Figure 2). Even so, the 3D plate model demands a considerably higher number of nodes, resulting in increased computational demands. From these perspectives, it is imperative to choose the appropriate modeling approaches for structures comprising beams and shell, such that a subtle balance is maintained between the resources required and the quality of the results achieved.

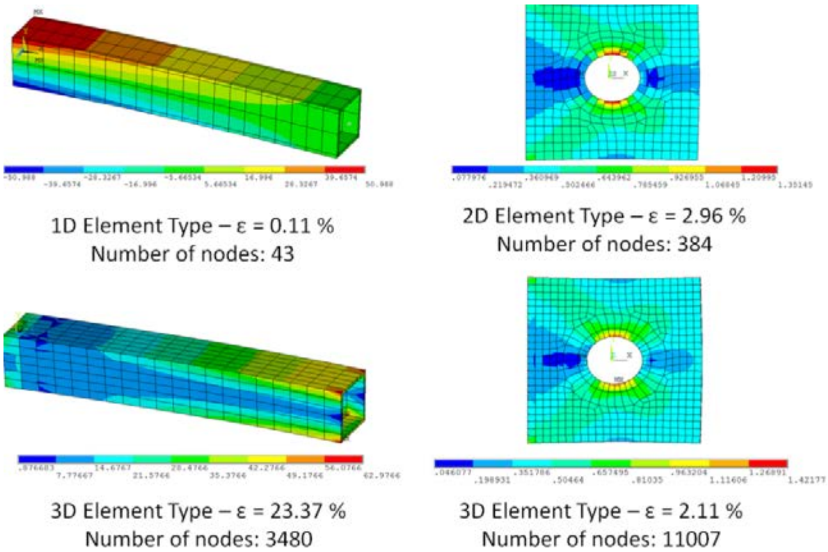

Fig.2 Comparison of errors between 1D, 2D and 3D modeling approaches

4. SIMPLIFIED FINITE ELEMENT MODELING FOR CAR LIFTING END-EFFECTOR STRUCTURES

A simplified finite element model is developed for the car lifting end-effectors, using the original 3D model as reference. The approach is deployed in three stages. In the first stage, only the major structural components are kept for the analysis. For example, the mounting bracket is removed, being replaced by a rigid support defined around the frame interaction area. A geometric analysis is performed to determine the cross-section types used for the beams, line bodies being generated from the centroid of them. This allows for 1D elements to be generated. Mid-surface extraction is deployed for the adjustable fittings, such that 3D bodies can be converted to 2D surfaces. The resulting entities are meshed with 2D elements by knowing the thickness offset property. The peculiarity of the methodology stands in connecting elements belonging to different modeling approaches. From this perspective, Figure 3 depicts how 1D beams are linked to 2D shells with the support of rigid bars. For this purpose, imprints are used to model the cross section interaction area. This allows for loads calculated in one node of the 1D element to be equally transferred to the shell surface. The DOF of the rigid bars are adjusted to take in to account welded areas. Another requirement is to keep the rigid bars parallel to the beam cross-section. A low level of distortion is recommended for the nodes corresponding to the 2D elements.

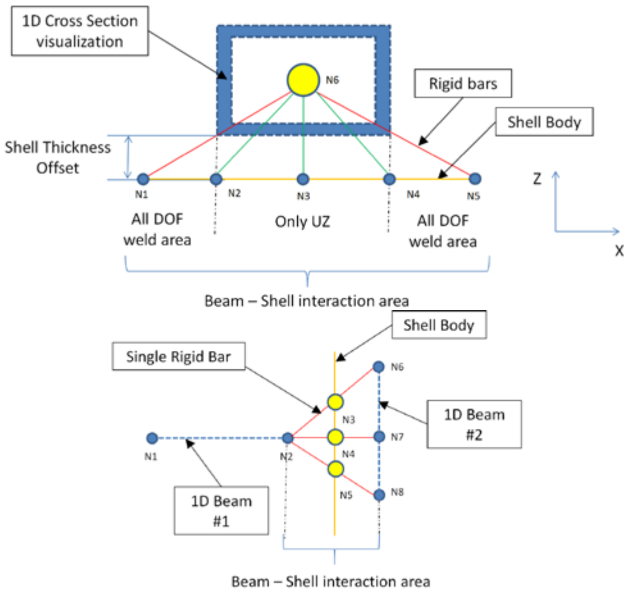

Fig.3 Definition of rigid bars

One limiting aspect in the approach is the existence of two beams that share a common interaction area. In this case, one single rigid bar is used to link both nodes corresponding to the 1D elements, the independent node belonging to the shell body. The methodology can successfully be implemented for all types of analysis demanded by the car lifting end-effector sizing processes. Even so, fast and accurate solutions can be achieved only by applying simplifying modeling and simulation assumptions. For example, symmetry can be used to calculate ¼ of the model when performing static analysis. On the other hand, 0D point mass elements can be deployed in modal and / or frequency response analysis, to take in to account the influence of the lifted object on the dynamic behavior of the assembly.

5. CASE STUDY

A case study is performed regarding the static analysis of a car-lifting end-effector by using ANSYS Mechanical APDL. Two planes of symmetry are defined, only ¼ of the model being analyzed. The total number of elements in the model is reduced to 13169. The results achieved are the main static sizing criterion: the normal, principal and equivalent stress. On the other hand, the total deformations are evaluated for assessing hoisting point deviations. Nodal extractions are performed such that the magnitude of tension and shear loads can be determined for verifying fastener and weld integrity. Based on the illustrated fringe plot, a clear distribution of the Von Misses stress can be observed (Figure 4).

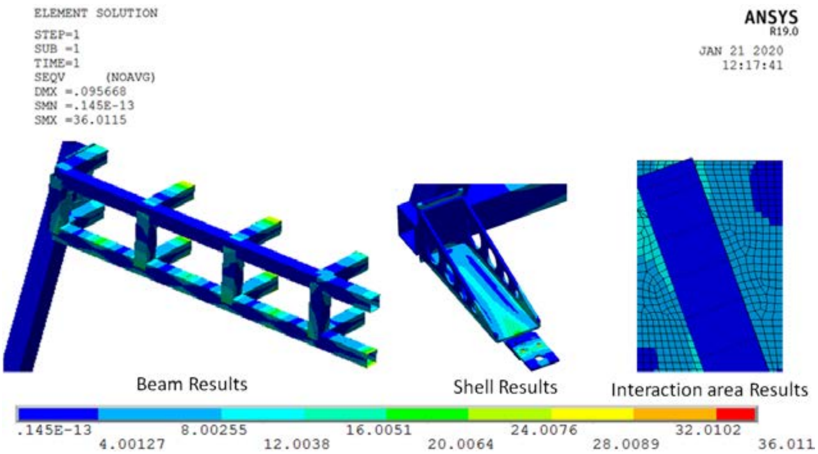

Fig.4 Details of the Von-Misses stress distribution for beam, shell and interaction area results

6. CONCLUSIONS

The present paper addressed the limiting aspects of product design software from the perspective of CAE tools deployed for car-lifting end effectors simulations. The proposed approach combined 1D and 2D element types for approximating the complex 3D geometry with maximized computational efficiency and result accuracy. Rigid bars were used to connect beams and shells. A case study was performed to illustrate the given concepts.

REFERENCES

[1]. Chen, X., Liu, Y. *Finite element modeling and simulation with ANSYS Workbench*. CRC press, 2014

[2]. Ili, S., Albers, A., Miller, S. *Open innovation in the automotive industry*. R&d Management, Vol.40, Iss.3

[3]. Li, Y. P. *Study on assembly task planning of multi-material car Body-in-White based on complexity analysis.* Mechanics and mechanical engineering: proceedings of the 2015 International Conference (MME2015)

[4]. Oñate, E. *Structural analysis with the finite element method. Linear statics: volume 2: beams, plates and shells.* Springer Science & Business Media., 2013

[5]. Popescu, F. D., Radu, S. M., Kotwica, K., Andraş, A., Brînaş, I. K. Dinescu S. *Vibration analysis of a bucket wheel excavator boom using rayleigh's damping model.* New Trends in Production Engineering, Vol. 2, Issue 1, pp. 233-241, (2019).

[6]. Pilkey, W. D. *Formulas for stress, strain, and structural matrices (Vol. 5).* New York: Wiley, 1994

[7]. Ugural, A. C. *Stresses in beams, plates, and shells.* CRC press, 2009