ASSOCIAÇÃO BRASILEIRA DE ENERGIA NUCLEAR – ABEN
2019 International Nuclear Atlantic Conference - INAC 2019
Santos, SP, Brazil, October 21-25, 2019

Agência Brasileira do ISBN
ISBN 978-85-99141-08-3

ASSESSMENT OF ANSYS LS-DYNA CAPABILITIES FOR ANALYSIS OF DROP TESTS OF NUCLEAR FUEL ELEMENT TRANSPORTATION CASKS

Daniel Silva de Mello Carvalho and Miguel Mattar Neto

Instituto de Pesquisas Energéticas e Nucleares, IPEN-CNEN/SP
Av. Professor Lineu Prestes 2242 – Cidade Universitária – CEP 05508-000
São Paulo – SP – Brasil
dansmcarvalho@gmail.com and mmattar@ipen.br

ABSTRACT

During the transportation of fuel elements, the cask has to provide shielding to protect workers, the public and the environment against the effects of radiation, to prevent an unwanted chain reaction, damage caused by heat and also to provide protection against dispersion of the contents. In order to standardize the design of fuel assembly transportation devices by numerical analysis, a set of dynamic analyzes was conducted to converge in a representative way the phenomena found in the drop tests used in the project qualification. Thus, this paper aims to present and discuss updated recommendations for contacts, material models and general configurations in three benchmarks. These benchmarks represent the phenomena found in numerical simulations of drop trials. Moreover, they are important to obtain an adequate correlation with the lowest possible use of computational resources. From the simulations, it was possible to observe the influence of an analysis carried out in plane strain and another one performed with the complete geometry modeled in scale 1:4 in relation to the computational cost and the precision of the results. A methodology was proposed to calibrate the stiffness and the damping control of the contacts and, mainly, their influence on the behavior of the structure.

1. INTRODUCTION

With the computational advancement, the codes became more robust at the same time as more accurate, allowing greater precision of the extracted results and therefore requiring the user-specific knowledge as well as understanding the problem's physics. The ANSYS Workbench LS-DYNA [1] is the result of the integration of the LS-DYNA solver developed by Livermore Software Technology Corporation, in [2], with the ANSYS Workbench interface, as well specific tools developed for this environment.

As predicted in [3], the computational improvement allows certain optimization of the modeling process and precision in the development of the mathematical model in observance of resource saving. It will, therefore, benefit both the Verification Authority and those proposing new designs. Keeping this process in mind, the following study describes suggestions for capturing the main phenomena encountered in the design of irradiated fuel element transport devices. Herein, an explicit dynamics analysis is used to determine the dynamic response of a structure due to stress wave propagation, impact or rapidly changing time-dependent loads.

From [3], it was proposed to reproduce the same study only with updated ANSYS Workbench LS-DYNA tools. The objective of this work is to compare the Benchmark previously performed to the current possibilities, describing the differences of the finite element modeling method and computational performance, exploring alternatives and proposing improvements in the input data. Divided in Benchmark 01, 02 and 03, respectively, Flat Side Impact of Concentric Cylinders, Corner Impact of Steel Cube and Axis Vertical Impact of Wooden Impact Limiter with Liner, each with its complexity, different phenomena are observed and so each of the following sessions will be separated accordingly.

2. BENCHMARK 1 – FLAT SIDE IMPACT OF CONCENTRIC CYLINDERS

For this benchmark was modeled as an external steel cylinder compressing a concentric inner cylinder of lead by impacting a flat surface after a drop test, in order to represent the cylindrical hull of a fuel element transport device. Were performed studies with the 2D order modeling, as suggested by [3], and 3D, thus establishing a comparison of the computational cost and the time of configuration of the analysis and its results.

2.1. Geometric Details e Materials Properties

Table 1 presents the material properties applied to the mathematical model as Figure 1 displays half cross-section dimensions.

Table 1: Material information of Benchmark 1.

Material	E [N/mm²]	ρ [kg/m³]	σ_y [N/m³]	σ_{hard} [N/m³]	ν	Behavior
Steel	2.10E+11	7850	2.00E+08	1.00E+09	0.30	Bilinear elastic-plastic material
Lead	1.70E+10	11340	1.90E+06	7.90E+08	0.45	

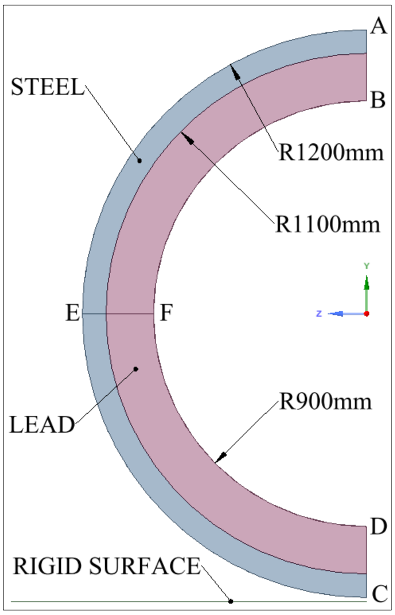


Figure 1: Schematic information of geometry.

2.2. Boundary Conditions

All contacts were defined with a friction coefficient of 0.2 (Coulomb Coefficient), being: Lead-Steel and Steel-Rigid Surface, as shown in Figure 2, Figure 3 and Figure 4. To find the plastic deformations identified in transportations casks drop tests, the free fall movement was representatively configured with the vertical final velocity, or impact velocity, of 30 m/s, applied at both cylinders. Then, the observation period of the structure behavior was estimated in 70 ms from the initial contact of the Steel Cylinder on the Rigid Surface.

2.3. Points of Interest and Outputs

The vertical displacements of the nodes at points A, B, C, and D, as Figure 1 shows, were defined as points of interest for evaluating the results, as well as the horizontal displacements of the nodes at points E and F, as shown in Figure 1.

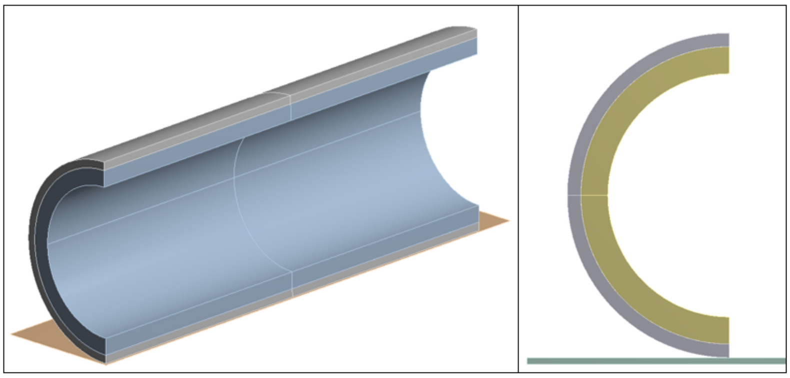


Figure 2: 3D (left) and 2D (right) geometric models.

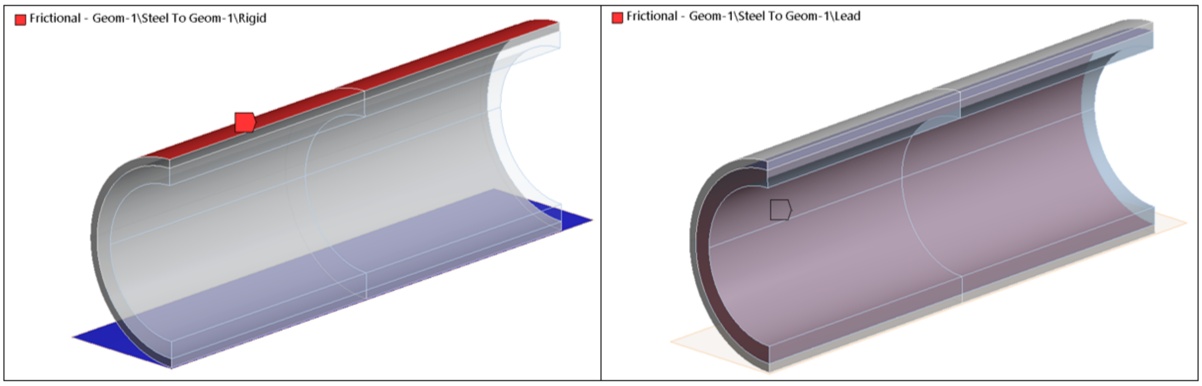


Figure 3: Contact view of the 3D model, Steel-Rigid Surface (left) and Lead-Steel (right).

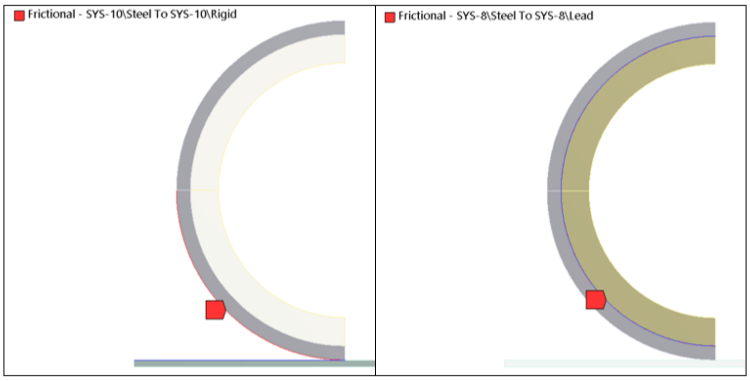


Figure 4: Contact view of the 2D model, Steel-Rigid Surface (left) and Lead-Steel (right).

2.4. Mathematical Models Comparison between the Reference [3] and Current Study

For the convergence study, three element sizes were applied in order to have in radial direction 6, 12 and 24 elements, as shown in Table 2. For the 2D environment only the most refined condition was applied, as seen in Table 2 continuation.

Table 2: Model discretization on the interest section.

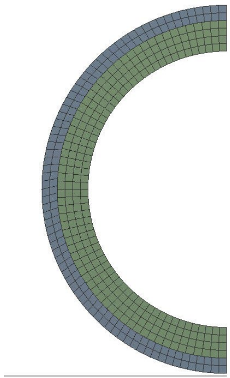
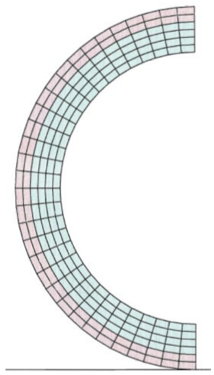
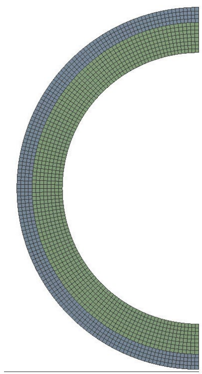
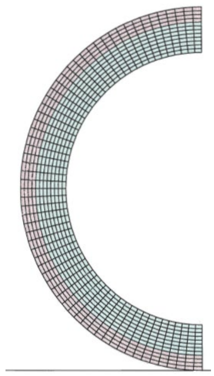
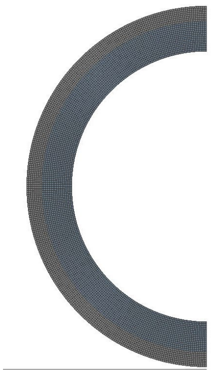
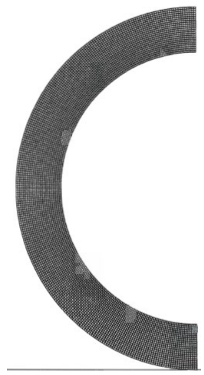
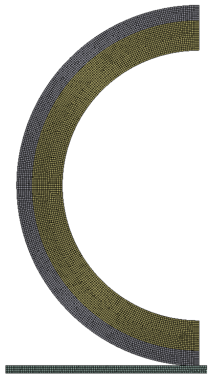
3D Model section	Reference
 402 Elements - 50mm	 192 Elements
 1592 Elements - 25mm	 768 Elements

Table 2: Model discretization on the interest section (continuation).

		2D Plane strain
		
6352 Elements - 12.5mm	6144 Elements	6876 Elements

2.5. Results and Reviews

For this case, two types of analysis were carried out: 2D plane strain as suggested in [3] and 3D with the consideration of long tube engineering with the use of two planes of symmetry, therefore 1/4 of 12-meter-long concentric tubes. The model of 6 radial elements presented insufficient results to describe the behavior of the structure since the models with greater discretization were more accurate and converged faster, as shown in Table 3.

Table 3: Models comparison of Benchmark 1.

Models	Nº Elements	Elapsed Time
3D - 6 elem.	52.248	1 m 5 s
3D - 12 elem.	385.981	11 m 43 s
3D - 24 elem.	3.052.860	177 m 27 s
2D - 24 elem.	6.876	53 s

A normal restriction to the plane of the study at the end of the tube was carried out to verify the influence of the long tube consideration. It was also observed the behavior of points in the plane at 3 meters from far end of the tubes to verify the assumption.

The condition of the different analyzes was essentially the same, but as the size of the element was reduced it was necessary to insert a keyword input to inform the Solver that it used more memory than the default. It was allowed that if necessary, could add small increments of mass when in some situation of instability in the element, that is, Automatic Mass Scaling was enabled, however, no mass was added. In order to approach the model from the previous study [3], Hourglass control settings were selected as Standard LS-DYNA [1] which even if unspecified, is assumed to have been used.

In Figure 5, the results of the reproduction of the 3D study on the section of interest, presented through the plastic strain, side by side to [3].

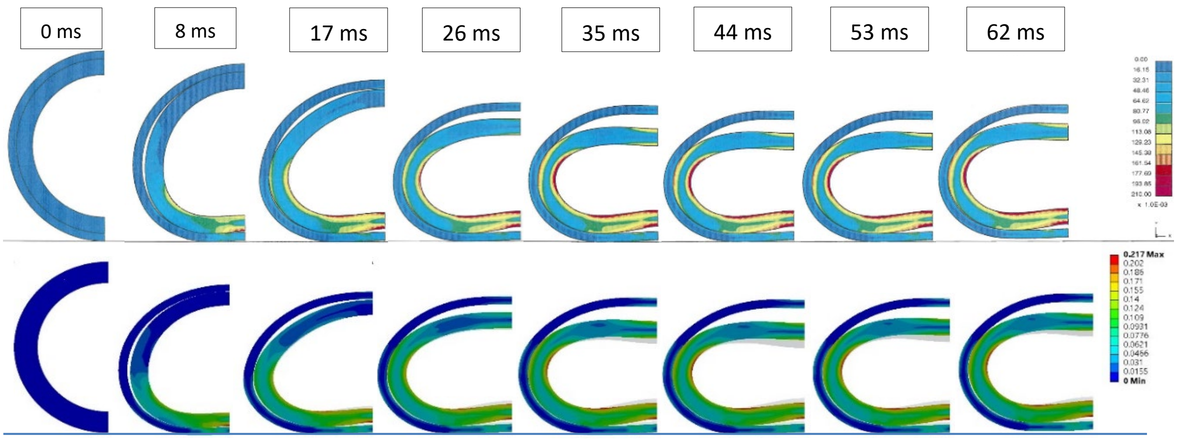
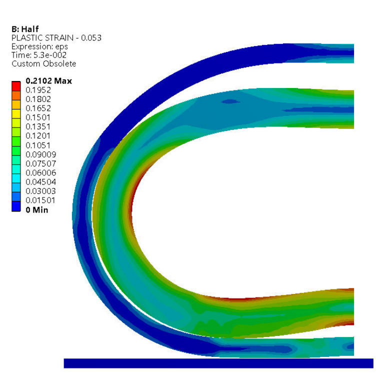
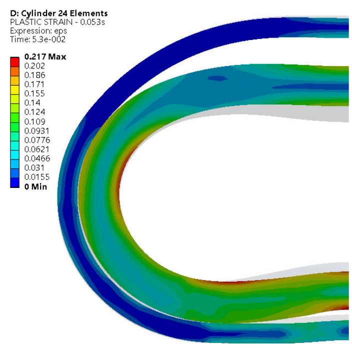
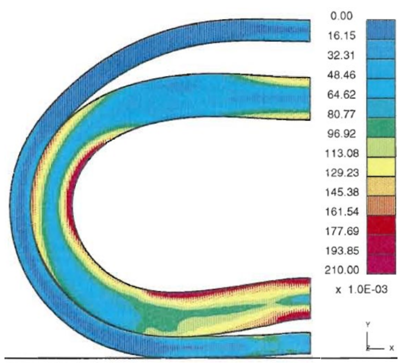
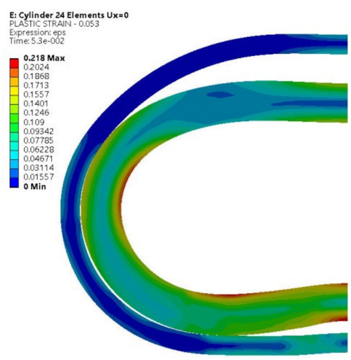


Figure 5: Plastic Strain results plot in time, [3] (up) and authors (bottom).

It is possible to establish an adequate verification from results found with the free-end 3D model and the results from [3] described with restriction in the Z plane. As Table 5 shows the results compared at the nodes of interest.

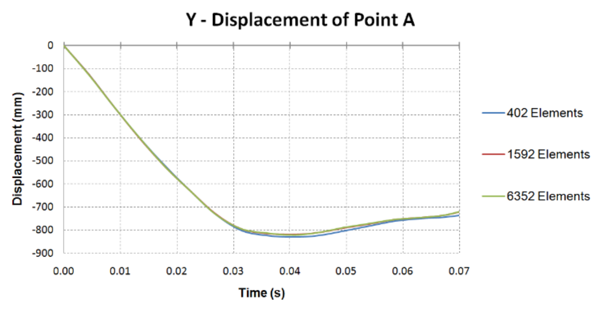
Following, in Table 4 and Table 6, are presented the results obtained with and without restriction at the end compared with the result of the 2D model. The instant of 53 ms was chosen because it represents the time the cylinders begin the return path, away from Rigid Surface.

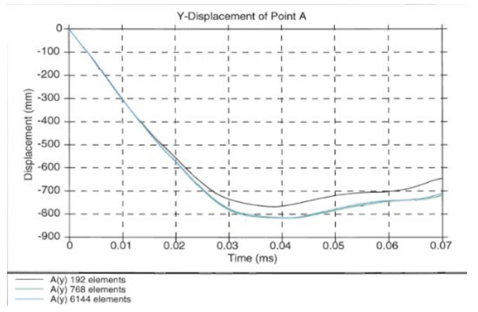
Table 4: Studies comparison.

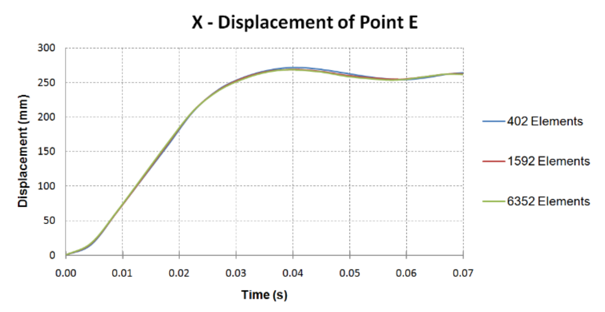
2D Plane Strain	3D *WITHOUT* restriction at the end
	
Reference	**3D *WITH* restriction at the end**
	

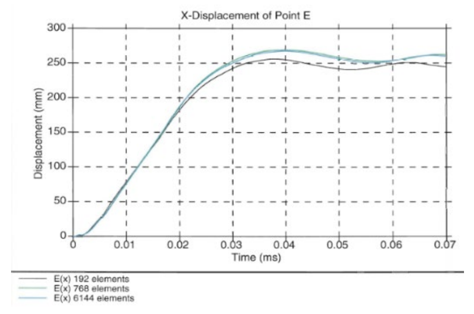
It can be observed that the three different analyzes presented the same result, and consequently, it was concluded that there is no need to perform the 3D modeling, and thus with the ANSYS Workbench LS-DYNA module it is possible to achieve the same precision in the map of plastic strain, which input card is in Output Controls in the Analysis Settings field, and plane strain, or 2D, is still a Beta option in the ANSYS Workbench LS-DYNA [1] environment.

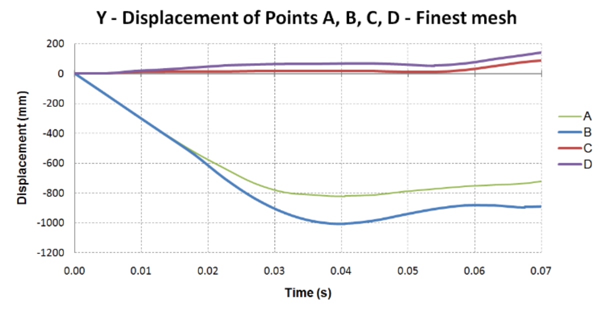
Table 5: Comparison of points A, B, C and D for 3D cases without restriction and [3].

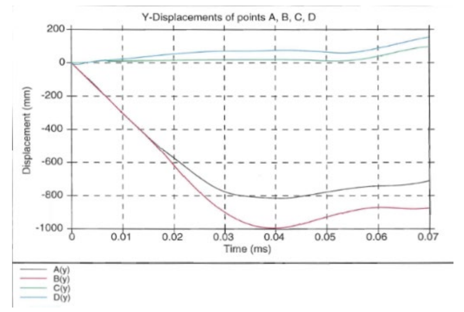


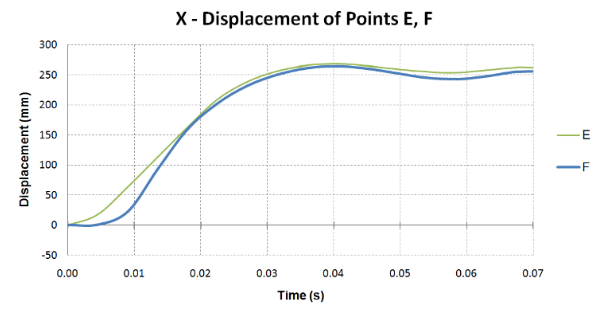












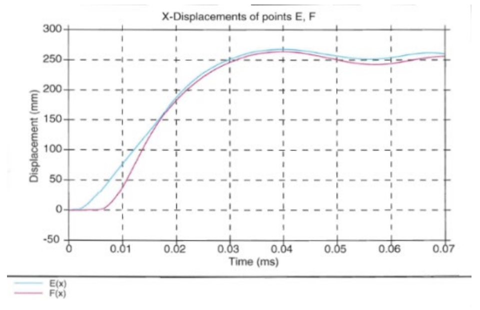
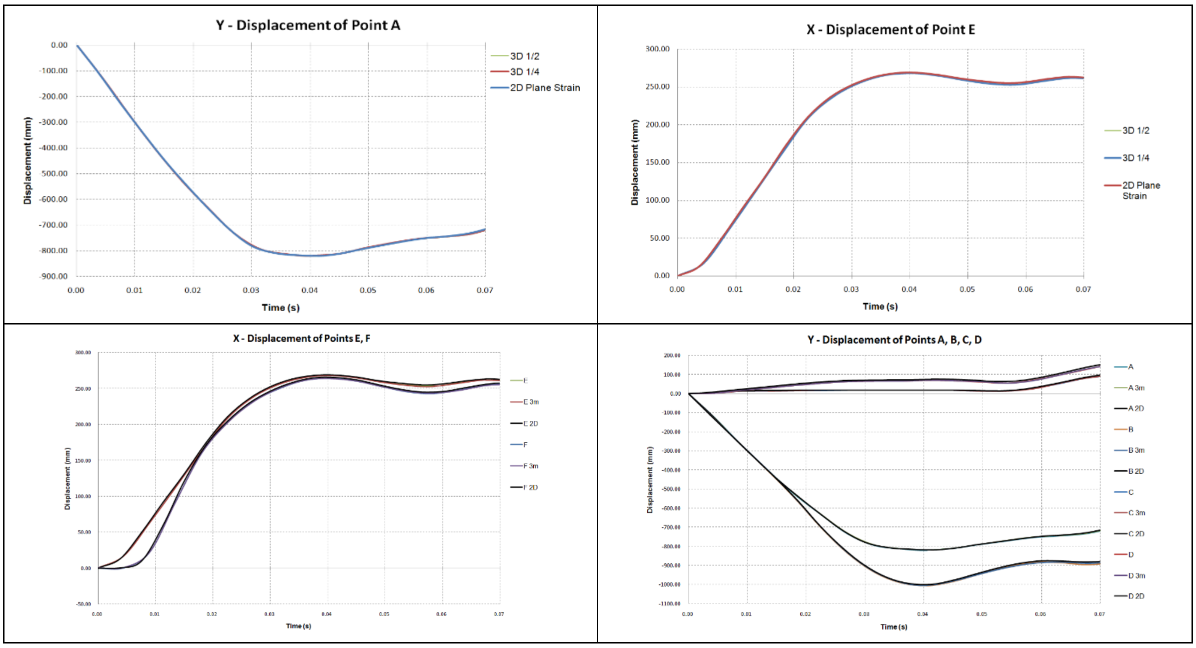


Table 6: Comparison between unrestricted 3D model points in two different sections and the 2D Plane Strain case.



3. BENCHMARK 2 – CORNER IMPACT OF STEEL CUBE

For this benchmark was modeled as a cube falling with one of its corners, or vertices, on a rigid flat surface, whose idealized density is corrected by three rigid plates fixed to the upper surfaces of the cuboid, in order to present the intense plastic displacement of a cubic geometry often found in impact absorbers in a fuel element transport device

3.1. Geometric Details e Materials Properties

Table 7 presents the material properties applied to the mathematical model in Figure 6 displays de geometrical model. The cube has an edge of 1 m and the rigid shells also with 1 m edge yet with 1 mm in thickness.

Table 7: Material information.

Material	E [N/mm²]	ρ [kg/m³]	σy [N/m³]	σhard [N/m³]	ν	Behavior
Steel Cube	2,10E+11	7850	2,00E+08	1,00E+09	0,30	Bilinear elastic-plastic material
Rigid Shells	-	1,67E+07	-	-	-	

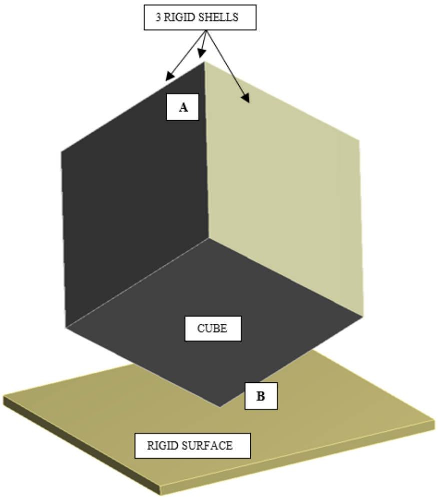


Figure 6: 3D geometrical model.

3.2. Boundary Conditions

The contacts of the Rigid Shells with the Cube have been defined as Bonded, so that they are completely connected to the top surfaces of the cube. The contact of the Cube-Rigid Surface was defined with friction coefficient of 0.2 (Coulomb Coefficient), as shown in Figure 7. To set the free-fall movement of 9 m height, the final velocity, or impact velocity, was determined at 13.3 m/s. Considering this velocity, the observation period of the structure behavior was estimated in 30 ms from the initial contact of the Cube on the Rigid Surface.

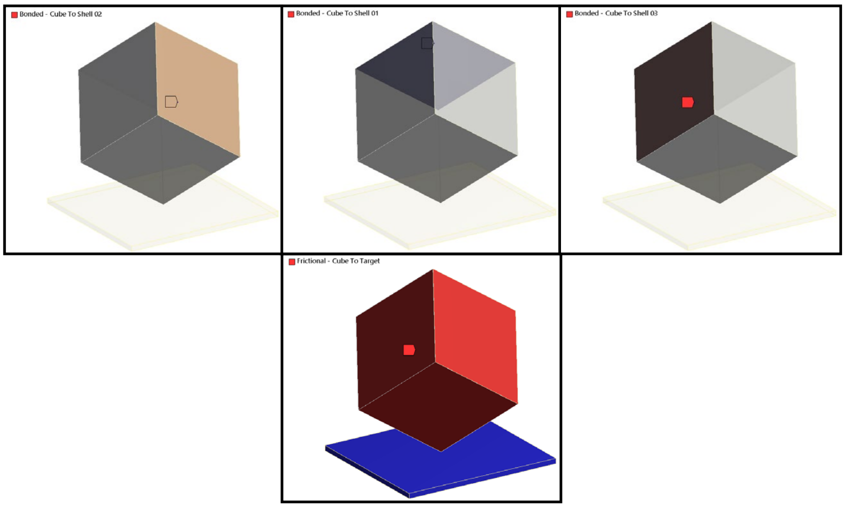


Figure 7: Rigid shells contacts(up) and cube-rigid surface contact.

3.3. Points of Interest and Outputs

The vertical displacements of the nodes A and B were defined as points of interest for the analysis of the results in time, as point A is the node of the top and point B the first node to impact the Rigid Surface at the bottom of the cube.

3.4. Mathematical Model

The element sizes used in the convergence study was: 50 mm (a), 25 mm (b) and 12.5 mm (c), as shown in Figure 8. The most refined mesh was modeled with the use of symmetry.

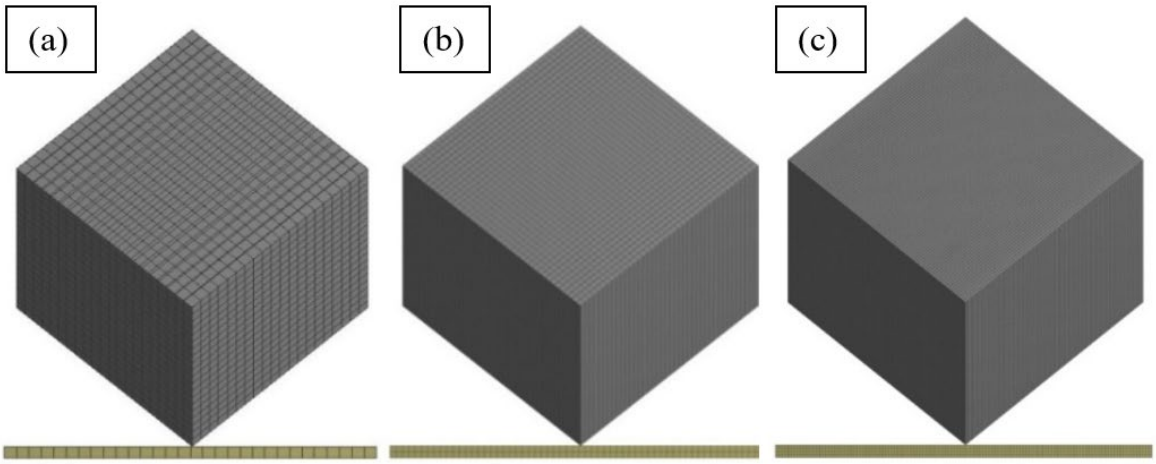


Figure 8: 50 mm (a); 25 mm (b); 12.5 mm (c).

3.5. Results and Reviews

In this case, a 3D analysis conducted in 2 steps: the first being to calibrate contacts for the expected phenomena and the second to discretize the mathematical model satisfactorily. Both the correlated steps, because it is possible to identify the influence on the behavior of the contacts according to size of the finite element, as well as the effect of the contact in the behavior of the structure and its compression. With isotropic materials of bilinear hardening, it was chosen to characterize the Rigid Shells component with shell element and both the Cube and the Rigid Target were modeled with solid elements of eight nodes. In parallel, a mathematical model was developed using symmetry that also allows a study of reduction in the time of computational use, as shown in Table 8.

Table 8: Models comparison of Benchmark 2.

Models	Nº Elements	Elapsed Time
50mm	10.500	23 s
25mm	77.600	4 m 43 s
12.5mm	595.200	68 m 10 s
w/ symmetry	298.967	64 m 55s

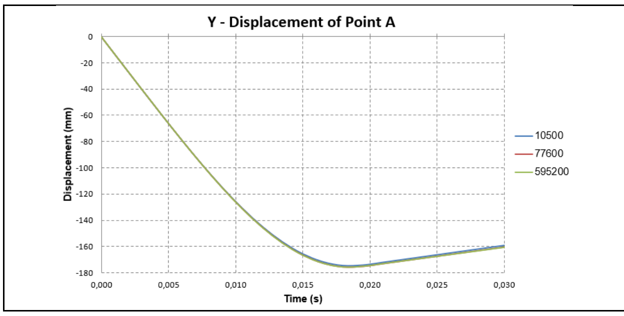
For most impact cases, the default ANSYS Workbench LS-DYNA contact configurations achieve good results, however due to the concentration of loads in a few nodes, more specifically, intense compression in a few elements, it was necessary to adjust some parameters. One of the most important parameters to be determined is the Time Step Safety Factor, being the default 0.9, but due to the size of the elements and the speed of the impact, smaller values were studied taking into account the computational time. In the same way, Automatic Mass Scaling was enabled to improve the resistance of the elements to this compression, but no local mass was added. It was also selected as Hourglass control the Standard LS-DYNA due to the lack of information about this in [3], there are other options that can influence an altered dynamic behavior, but proportionately Hourglass Energy was considered minimal and thus, against the limit of 10% given as good practice not requiring change in the control formulation.

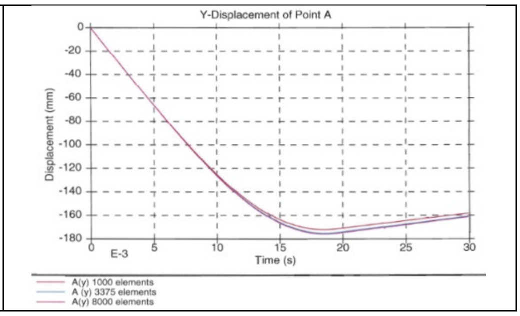
As far as the computational cost is concerned, the default memory allocation was not enough and so the limit was changed for all the analyzes according to the need of each mathematical model, besides requesting the use of the maximum possible of available cores.

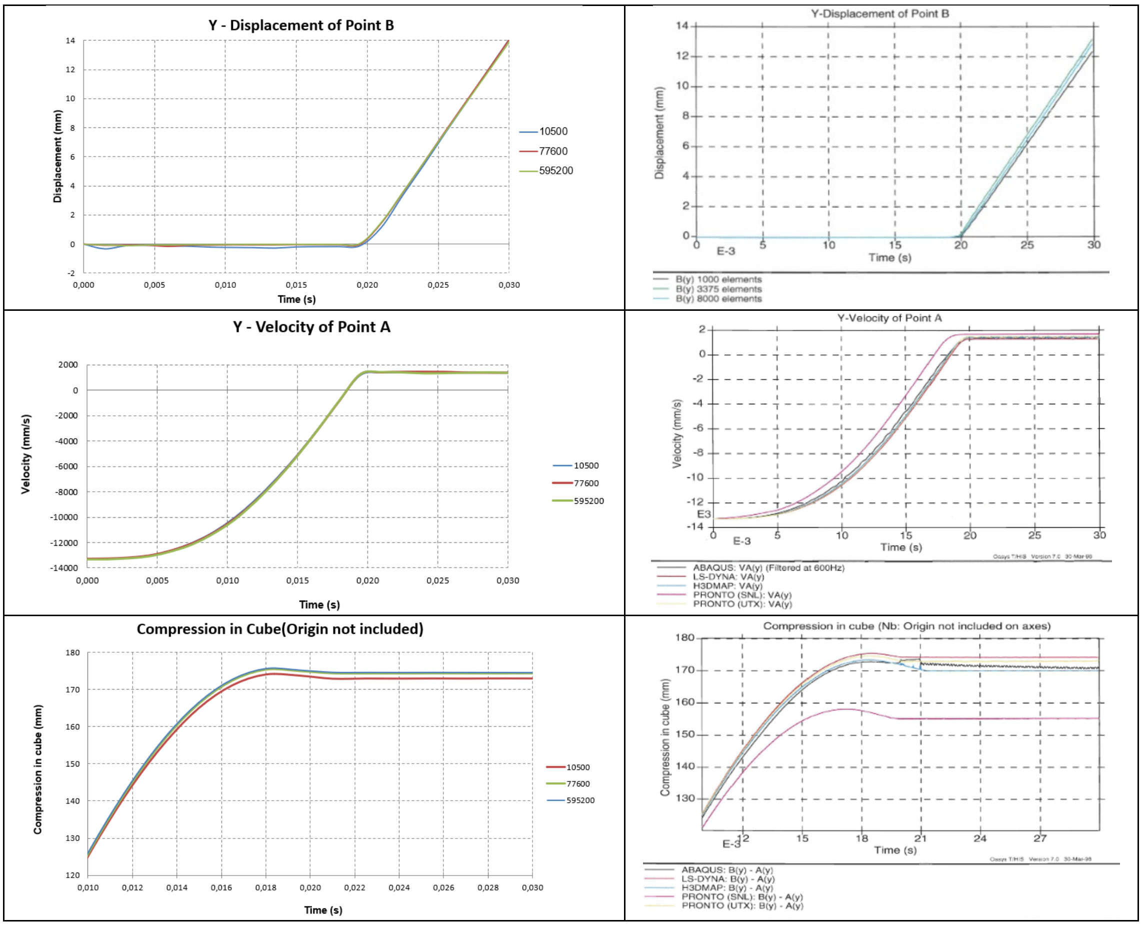
At this rate, for the contacts it is expected that Point B, or first node to contact the Rigid Target, has zero displacement in Y direction, that is, that there is no penetration between the elements in contact, at least until the Cube starts the return path. With this guideline, and with the less refined model, 50 mm elements, the Contact Properties card was used to calibrate the contact in order to fit the interaction between the bodies more precisely and for this, some of the parameters were Viscous Damping Coefficient (VDC), Slave Penalty Scale Factor (SPSF), Master Penalty Scale Factor (MPSF), Optional Solid Element Thickness (OSET), Soft Constraint Scale Factor (SCSF) and Depth. It is understood, respectively, as the damping (VDC) and stiffness (SPSF and MPSF) of the contact from the moment the nodes interact with each Time Step, once the increment has been defined it was possible to find the behavior described [3]. The OSET, SCSF and Depth parameters were defined through submodel studies and, with the aid of recommendations for special cases, studies were also carried out in the final model, whereas in order to calibrate these values it was more straightforward than for stiffness or damping that required a progression of more stages. Stiffness and damping needed to be calibrated in order to reduce penetration, achieve good stability. For this, the stiffness suffered increments of 0.5 while damping received increments of 5 units.

With the penetration module for the default settings, and then in the calibration process, the value of 1 mm for the OSET showed the best stability, and the same applies for SCSF at 0.15. In the case of the Depth parameter, the lowest possible value was selected so the code recognizes the contact between the surfaces, given the distance modeled in the CAD process.

Table 9: Results comparison current and reference.







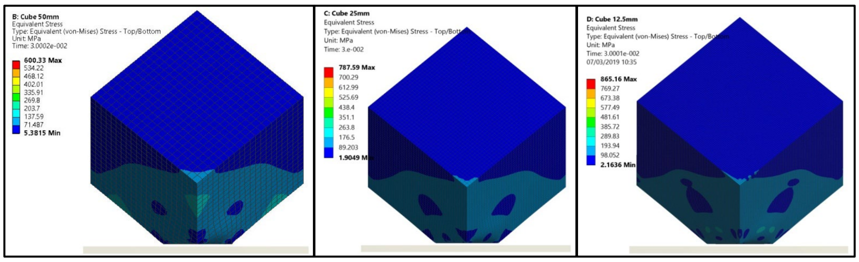
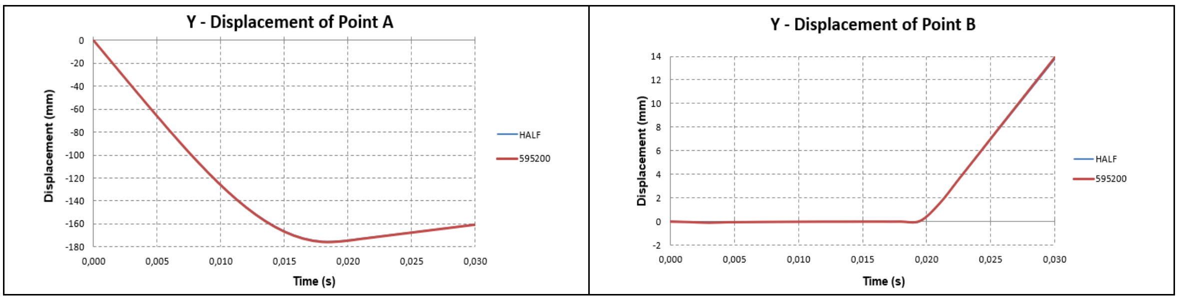
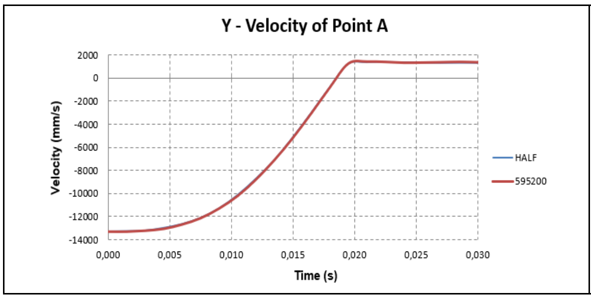


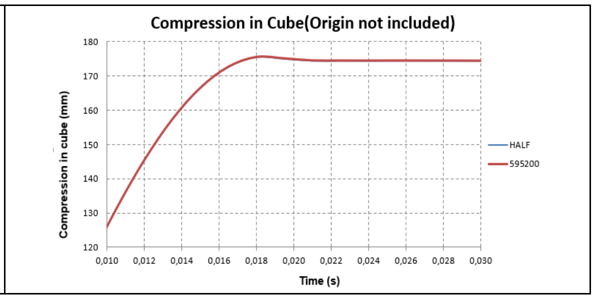
Figure 9: Deformed state displaying Von Mises tension map.

As demonstrate in Table 9 and Figure 9, it was possible to reproduce the study from reference [3], but this geometry allows at least one plane of symmetry, so for the most refined case a study was done for a half model, then the results are presented in Table 10.

Table 10: Results comparison half and full.

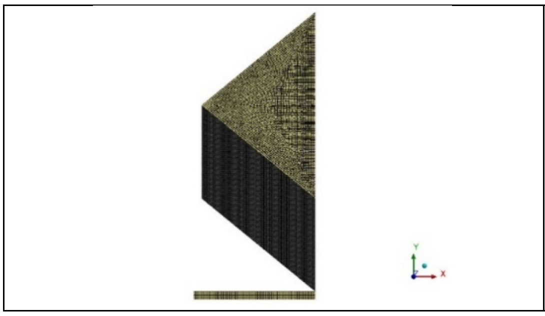


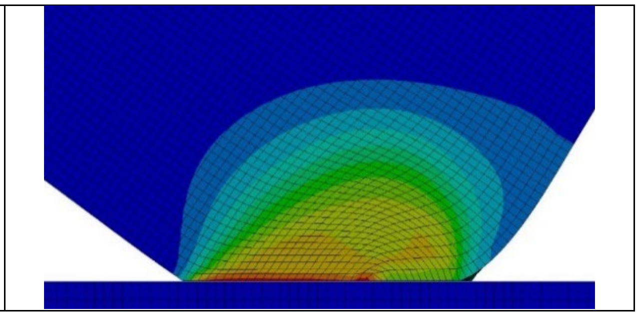




Next, the mathematical model with the application of the concept of symmetry and the plot of plastic strain, in Table 11.

Table 11: Half model appearance (left) and plastic strain plot (right).





3. CONCLUSIONS

In summary, it was possible to satisfactorily reproduce [3] with the available updated tools and explore different approaches. However, the third step, the Benchmark 3, which describes the compression of a piece of wood encapsulated with steel sheets, was shown to be complex when taking into account studies involving wood properties that succeeded [3]. Therefore, this is a case that will be studied with more details in a future work.

ACKNOWLEDGMENTS

The authors are grateful to IPEN for providing all the necessary structure for the conclusion of this work.

REFERENCES

1. ANSYS® Workbench LS-DYNA, Release 2019 R2, Help System, ANSYS, Inc.
2. Livermore Software Technology Corporation, LS-DYNA Keyword User's Manual, Volumes 1 and 2, Version 10580, 2018.
3. Ove Arup and Partners International (OAPIL), Gesellschaft für Nuklear-Behälter mbH (GNB), "Evaluation of Codes for Analyzing the Drop Test Performance of Radioactive Material Transport Containers", European Commission DG 17, United Kingdom, Germany, March 1998, Report Ref 53276/02.