ANALYSIS OF THE BEHAVIOUR OF MODULAR WOODEN PANELS DURING A DROP TEST USING EXPLICIT DYNAMICS

DATCU Tudor-Răzvan
Faculty: FIIR, Specialization: Robotics, Year of studies: IV, e-mail: datcutudor@yahoo.com

Scientific coordinator**: Prof. Dr. Eng. Cristina PUPĂZĂ**

SUMMARY: The research comprises a modular wooden frame composed of planks, modeled and analyzed in ANSYS. The structure was subjected to a drop test from the height of 2m and an inclination angle of 15^{0}. The simulation was performed employing the Finite Element Method in an Explicit dynamics approach, on a model with a highly controlled mesh and advanced computation settings.

KEYWORDS: modular frame, drop test, FEM, Explicit dynamics.

1. Introduction

The research is focused on the behavior of the prefabricated wooden frame manipulated by a robot, during a manufacturing process that takes place in a robotic cell.. It will be subjected to an accidental fall on the floor, from a height of 2m and an angle of 15^{0}.

Fig. **Eroare! În document nu există text cu stilul precizat.**. The wooden frame placed on the table surface

2. State of art

The FEM simulation is based on an Explicit dynamics solver that includes advanced capabilities which allow depicting the dynamic behavior of the assembly on a detailed model, and to captures the physics for almost any field [1].

Model with 1D, 2D or 3D structures can be studied, considering contacts with severe nonlinearities, residual shape changes (loss of structural stability), cracks and cumulative effects. This method is used for transient phenomena with a short duration of time and extreme non-linearities [2]. The simulation project is illustrated in Fig. 2.

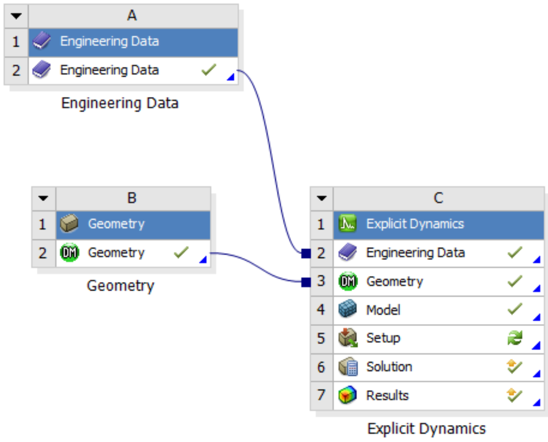


Fig. 2. Project structure in ANSYS

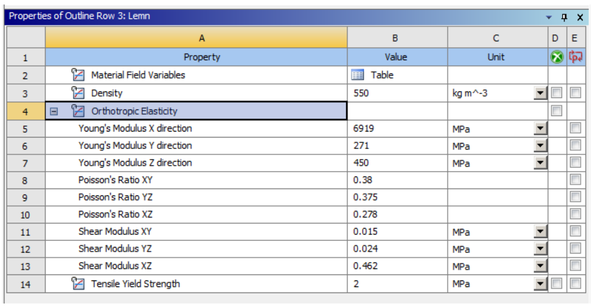
The material selected for the frame is pine wood. Because wood is a complex material with orthotropic properties based on its internal structure consisting of long and thin fibers arranged in concentric layers, it has different elastic properties on the 3 directions (longitudinal, radial, tangential). All the material properties were extracted from a recent scientific research article [3] in Table 1.

Table 1 [3]

Table 1. Off-diagonal terms and eigenvalues of the compliance matrix (2.3) for species of softwood

	data found in the literature				calculations in this paper			
	E_1	G_{12}	ν_{12}	ν_{21}	$-c_{12}$	$-c_{21}$	λ_1	λ_4
	E_2	G_{13}	ν_{13}	ν_{31}	$-c_{13}$	$-c_{31}$	λ_2	λ_5
	E_3	G_{23}	ν_{23}	ν_{32}	$-c_{23}$	$-c_{32}$	λ_3	λ_6
	[MPa]		[-]		[10^{-12}Pa^{-1}]		[10^{-6}Pa^{-1}]	
Pine [21]	6919	262	0.388	0.015	56	56	0.00014	0.00382
	271	354	0.375	0.024	54	54	0.00170	0.00423
	450	34	0.278	0.462	1027	1027	0.00282	0.02941

The allowable maximum stress σ_a was chosen as an average between several models of pine wood from the Matweb library (Fig. 3).


Properties of Outline Row 3: Lemn

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	550	kg m^-3		
4	Orthotropic Elasticity				
5	Young's Modulus X direction	6919	MPa		
6	Young's Modulus Y direction	271	MPa		
7	Young's Modulus Z direction	450	MPa		
8	Poisson's Ratio XY	0.38			
9	Poisson's Ratio YZ	0.375			
10	Poisson's Ratio XZ	0.278			
11	Shear Modulus XY	0.015	MPa		
12	Shear Modulus YZ	0.024	MPa		
13	Shear Modulus XZ	0.462	MPa		
14	Tensile Yield Strength	2	MPa		

Fig. 3 Material properties for pine wood

The CAD model was prepared in the DesignModeler module, and simplified by considering the entire frame as a single part, disregarding the nail joints that are present in the real model. All the modeling commands in model preparation stages are summarized in Fig. 4.

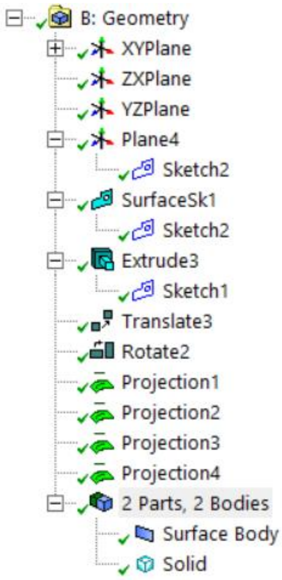
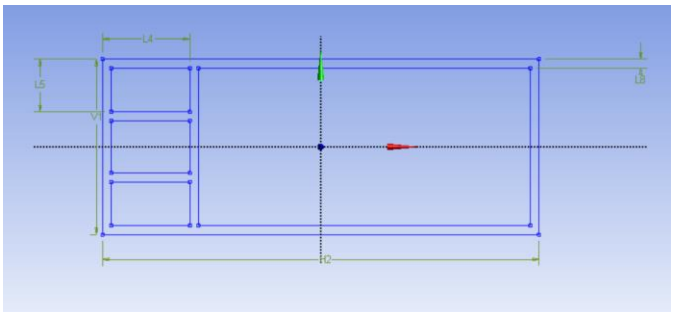


Fig. 4. Modeling stages in DesignModeler

The frame was designed based on the dimensions of the planks from the robotic cell, with the standard dimensions in the imperial system of 2x4 in. From the initial sketch, an extrusion was performed to materialize the 3D part. All the dimensions are summarized in Fig. 5.



Dimensions: 5	
H2	2500 mm
L4	500 mm
L5	300 mm
L8	50.8 mm
V1	1000 mm

Fig. 5 The 2D sketch that

The structure was placed at an initial height of 2m and tilted by 15º to represent an unfavorable position of the accidental fall (Fig. 6).

Details of Translate3	
Translate	Translate3
Preserve Bodies?	No
Bodies	1
Direction Definition	Selection
Direction Selection	Plane Normal
FD2, Distance	2000 mm

Details of Rotate2	
Rotate	Rotate2
Preserve Bodies?	No
Bodies	1
Axis Definition	Selection
Axis Selection	3D Edge
FD9, Angle	15 °

Fig. 1 Details of the initial conditions

All the edge points have been projected on the surfaces to enable the mesh generation. The maximum element size was set to 30 mm, to avoid the occurrence of the hourglass energy during the simulation (Fig. 7).

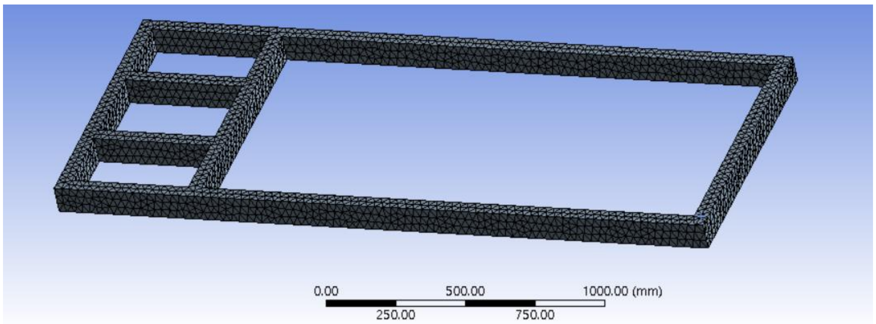


Fig. 7 The computational model of the wooden frame

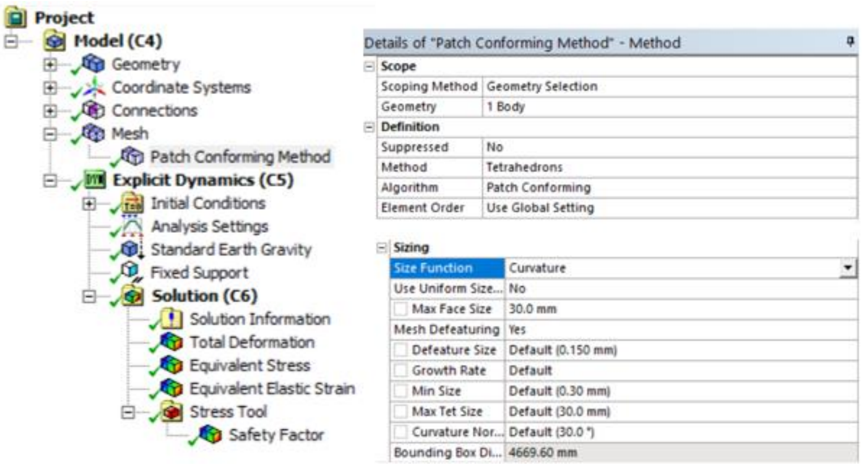


Fig. 2 Tree and discretization parameters

A standard earth gravity of 9.81m/s^2 was applied to the frame, and the floor was considered rigid. The total duration of the drop test was 1.5 seconds, and the tetrahedral elements integration was set at constant pressure (Fig. 9).

Solver Controls	
Solve Units	mm, mg, ms
Beam Solution Type	Bending
Beam Time Step Safety Factor	0.1
Hex Integration Type	1pt Gauss
Shell Sublayers	3
Shell Shear Correction Factor	0.8333
Shell BWC Warp Correction	Yes
Shell Thickness Update	Nodal
Tet Integration	Constant Pressure
Shell Inertia Update	Recompute
Density Update	Program Controlled

Fig. 3 Explicit dynamics settings

The most important results of the simulation are the equivalent stress according to the von Mises criterion, the equivalent elastic strain and the safety factor. The maximum displacements are not relevant because the frame is simulated during motion and the values correspond largely to the distance during in the fall [4] [5]. The maximum equivalent von Mises stress of 3.0337 MPa is significant and exceeds the yield strength of the pine wood at 2MPa (Fig. 10). The maximum values appear after the impact and start to decrease afterwards. The maximum value of the elastic strain is 0.11%, acting in an alternating pattern on the opposite side of the frame (Fig. 11).

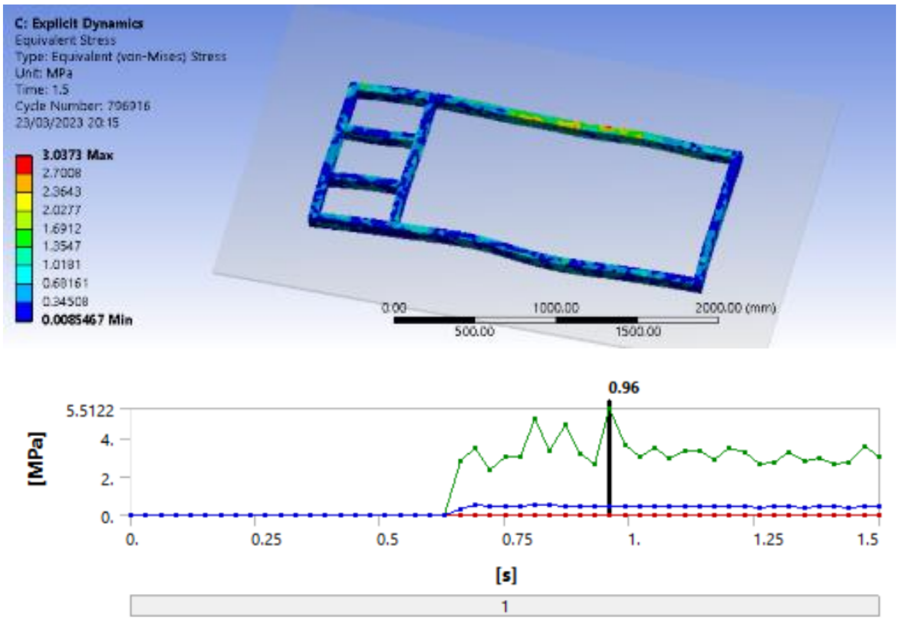


Fig. 10 Maximum equivalent von-Mises stress

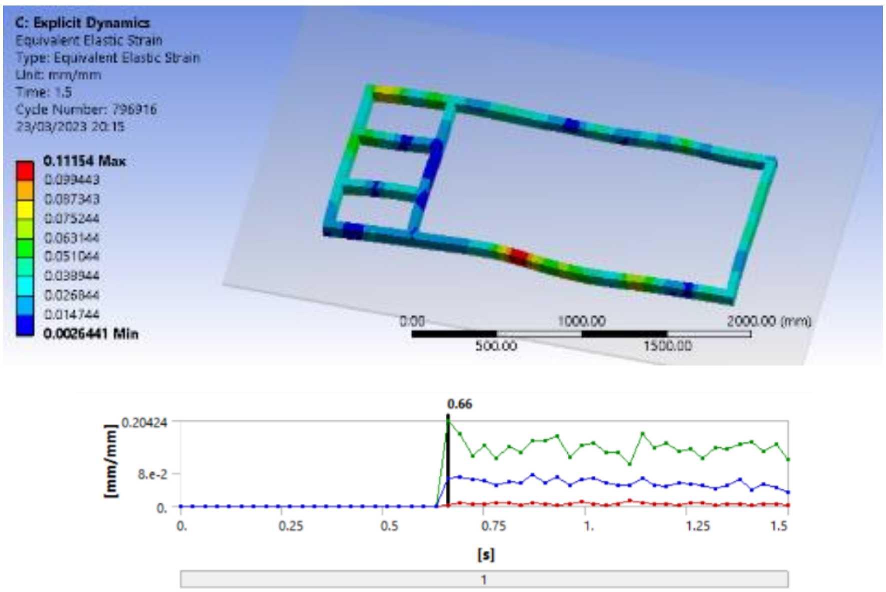


Fig. 11 Equivalent elastic strain

Throughout the simulation the safety factor is below the unit at several points of the model (Fig. 12). Thus, it may be assumed that the material does not withstand the impact loads, and this could lead to cracks occurring in the most peak points. To model the crack and breaking phenomena, additional material properties have to be introduced, most of the information can only be obtained experimentally.

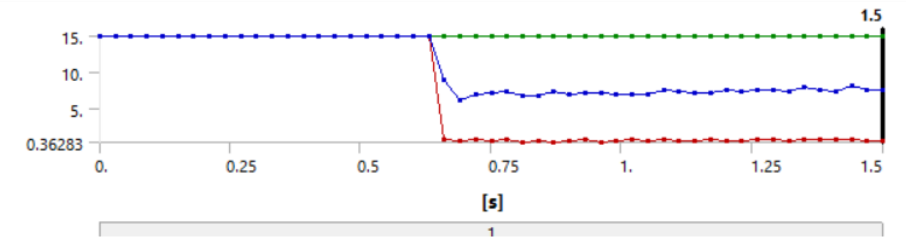


Fig. 12 Safety factor

3. Conclusion

The work has highlighted the pattern of deformation for the wooden frame during a fall and an accidentally impact from the height of 2m at a 15° angle. For the studied robotic cell, the trajectory of the robots handling these frames must be reconsidered, to avoid trajectories that are too high. From this perspective the design of the modular effectors could be improved to ensure sufficient grip of the frames, even in the event of a failure of the pneumatic system.

The novelty of the approach is based on the material model of the pine wood (wood, with orthotropic properties), the analysis be means of explicit dynamics, as well as entire robotic cell for assembling modular wooden panels.

Future work may focus on a different mesh pattern in respect to the hourglass energy to avoid wrong results. The simulation could also be performed in several what if scenarios, considering different heights and angles, to determine the point from which the wooden frame suffer permanent damage during the impact.

4. References

[1]. https://tensor.ro/ansys/ansys-structural-mechanics. Accessed at May the 2nd, 2023
[2]. Butnariu S, Analiza cu elemente finite în ingineria mecanică. Aplicaţii practice în ANSYS, editura Universitatii Transilvania din Braşov, ISBN: 978-606-19-0474-7, decembrie 2014
[3]. Obara, P. "Verification of orthotropic model of wood." Archives of Civil Engineering 64.3 (2018)
[4]. Pupăză, C. - Modelare CAD-FEM. Editura POLITEHNICA PRESS, 978-606-515-519-0, Bucureşti, 2013
[5]. http://www.ansys.com. – Demo, Tutorials, News, Ansys Advantage, ANSYS Focus, Accessed at April the 12th, 2023