SIMULATION OF A TACTILE TESTING SYSTEM OPERATION BASED ON THE TRANSIENT STRUCTURAL DYNAMIC ANALYSIS

GEAMĂNU I. Mihai-Bogdan
Faculty of Industrial Engineering and Robotics, Robotics specialization, 4th academic year,
email: bogdan.asx14@gmail.com

Scientific coordinator: Professor **Tiberiu DOBRESCU** PhD. Eng.
Scientific coordinator: Professor **Cristina PUPĂZĂ** PhD. Eng.

SUMMARY: The research consists in the simulation of a tactile testing system employed in the computer industry, comprising the end-effectors of the industrial robot, based on a Finite Element Analysis in a transient regime. Considering the detailed physical design of the end-effectors and intelligent computing devices, the components of the automated testing system are modeled in the CAD environment. By transferring the CAD database to the CAE preprocessing system, assigning the appropriate materials, and defining kinematic joints and constraints, simulation results can be extracted, and the behavior of the end-effectors and the components of the tested device can be assessed. The results can be further employed for the design optimization of the computing devices by replacing the materials from which some components are made, optimizing the shape of the keys and of the touch panel, but also optimizing the physical testing methods.

KEYWORDS: laptop, materials, tactile end-effector, dynamic behavior, optimization.

1. Introduction

Testing of the touch screens, keyboards, and touch panels of smart devices, such as mobile phones, tablets, and laptops, involves performing pre-set routines according to international standards, for instance: punctual precision testing, testing the input resolution, testing the replication of the trajectory established by the program, etc. These test procedures can be performed manually by humans, but the execution times of the tests is long, and the productivity is poor. Therefore, the tests are performed automatically with the help of industrial robots, which have end-effectors devoted to this task.

The overall size of the smart device test cell (Fig. 1) relies on the size of the device to be tested. The robot is then chosen in relationship with the workspace required to reach any point that is a target to be reached and tested. Generally, the robotic cell consists of the industrial robot, the tested device, the tactile end-effectors, end-effector storage system (if the robot is equipped with an automatic coupling-decoupling system), and the modular clamping system of the tested device, the computer through which the data acquisition is done, and the results of the tests are monitored.

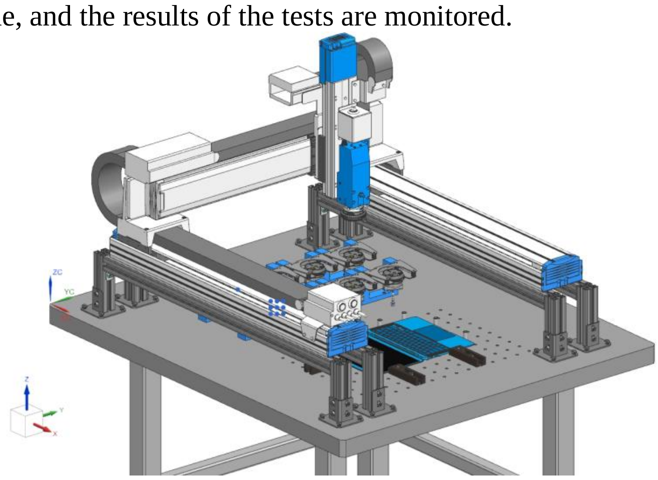


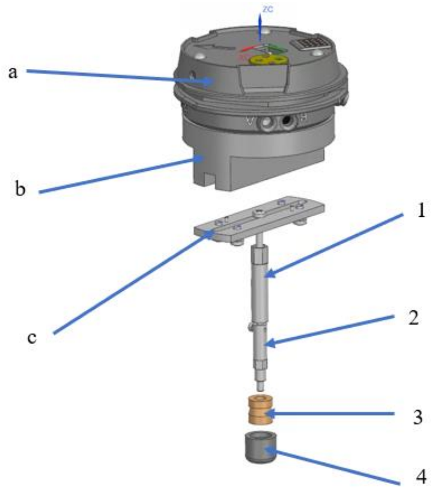
Fig. 1. Isometric view of the robotic cell virtual model

The robotic cell works in the following steps:

- The robot equips an end-effector corresponding to the test to be performed from the end-effector storage system;
- The robot moves to the point where it can start the testing procedure;
- The scheduled test is performed by pressing the tactile end-effector on the tested item (key, touch panel, touch screen);
- The robot moves to the safe position of the scheduled test;
- The robot moves to the point where it can store the tactile end-effector in the storage system;
- The end-effector is stored in the storage system, and the robot passes to the next programmed point.

This research is focused on the operation of the tactile testing system, respectively the tactile end-effectors employed by the industrial robot, based on a dynamic Finite Element Method (FEM) analysis in the transient regime.

The original design of the end-effector (Fig. 2) is the first objective to create the computational model.



a – tool mounting flange;
b – customized adapter plate;
c – end-effector mounting plate using the customized adapter plate;
1 – end-effector housing;
2 – tactile finger;
3 – brass fingertip;
4 – rubber sheath around the brass fingertip to protect the pressed surfaces.

Fig. 2. CAD model of the end-effector employed in the computer testing system

Additionally, the connection between the upper housing and the tactile finger of the end-effector is made by means of a helical spring.

The objective of the simulation employing FEM is to analyze the behavior of the tactile system during the interaction with the tested device. The most important elements of the tactile end-effector design are *the helical spring* and *the rubber sheath* around the brass fingertip. The rest of the paper is divided in: a brief assessment of published studies on the topic in Chapter 2, a description of the computational model preparation stages included in Chapter 3, discussion of the simulation results in Chapter 4 - conclusion regarding the industrial use of the tactile testing system and future work.

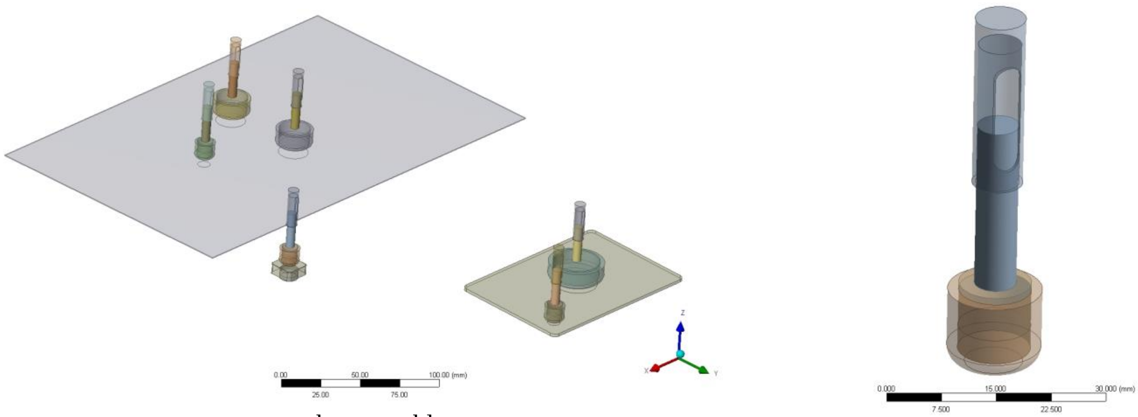
2. Related work

While the technology of using industrial robots with tactile end-effectors to test the functionality of touchpads, keyboards and touchscreens is still at the beginning, the research done regarding this new technology is still developing [1], [2], [3]. The studies on tactile end-effectors are focused on the end-effectors design, stiffness, and the manufacturing technology. None of the published works treats the dynamic response created by pressing the tactile end-effector employing an industrial robot on the tactile surfaces. The only available information about tactile surface testing automation and optimizing the test methods by analyzing mechanical information are the videos shared by some companies on YouTube, which can not be considered scientific published works.

3. Preparation of the computational model

The major difference between the CAD model of the robotic cell and the computational model in a CAE environment is that the FEM model encompasses fewer components - only those relevant to the simulation - and these parts have undergone geometrical simplifications. The reason for geometric defeaturing is that the discretization can much more correctly approximate simple geometries rather than detailed ones and we can avoid small distorted finite elements generation.

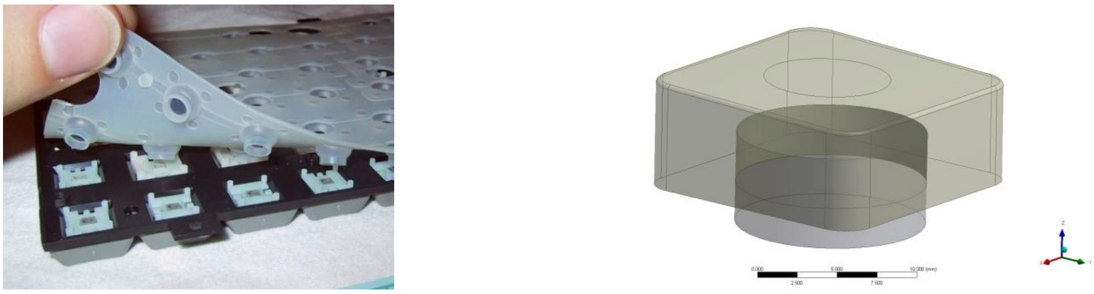
Figure 3 illustrates the simplified geometry of the computational model. It is evident that the housing of the computing device has been replaced with a simplified geometry that approximates the exact shape of the original housing, the keyboard keys that are not used in the simulation have been removed from the geometry, and for each tested item (touch panel, key, touch screen) a corresponding effector has been placed to perform the specific test during the simulation, as happens on the real robotic cell. The operation of each effector will be defined on different time steps in the transient analysis settings, to simulate the successive and individual compressions performed by the robot. Regarding the simplification of the end-effector geometry (Fig. 3-b), the fillet diameter of the lower finger was replaced by a simple cylinder with the same outer diameter, the key radius was deleted from the housing geometry, and the brass tip geometry was simplified and transformed into a cylinder



a - the assembly b - tactile end-effector geometry

Fig. 3. Simplified geometry of the computational model

A detail of the model is illustrated in Figure 4-b where a simple model of the membrane under the keyboard is required. The membrane makes the connection between the compressed key and the contact corresponding to the key on the motherboard. Since the scope is to press a single key, it is sufficient to model a cylinder similar in size to the membrane under a key and to create a cavity inside in the imported CAD model.



a - the membrane beneath the keyboard that transmits the impulse force of the key to the electric board

b - cavity model in the pressed key and the simplified membrane insert

In order to perform a reliable simulation, it is essential to properly define the materials of the components. As such, the materials employed will be the followings (Fig. 5): • aluminum alloy -

imported from the ANSYS Workbench materials library; • brass - defined; • glass - imported from the ANSYS Workbench materials library; • polycarbonate - defined; • rubber - defined.

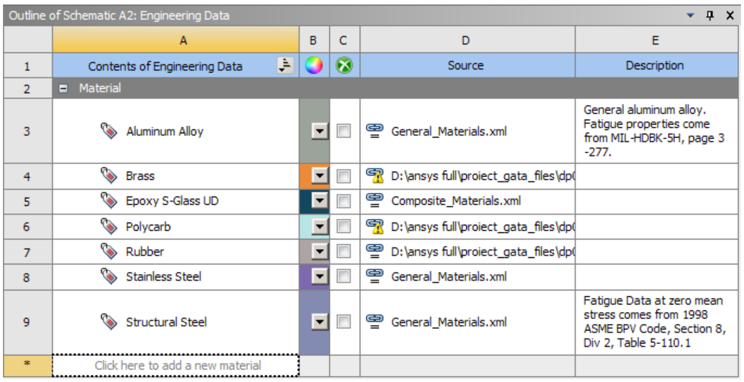


Fig. 5. Employed materials

For the newly defined materials the relevant properties are [4]: • density; • modulus of elasticity (Young's modulus); • Poisson's ratio; • transversal modulus of elasticity & the incompressibility parameter for rubber. The material properties defined for brass and polycarbonate are summarized in Table 1.

Table 1. Material properties for brass and polycarbonate

Material	Density g/cm^3	Isotropic elasticity	
		Young modulus MPa	Poisson's ratio
Brass	8.39	$1.17 \cdot 10^5$	0.34
Polycarbonate	1.2	2400	0.37

The rubber material properties are as follows: isotropic elasticity (Young's modulus = 60 MPa and Poisson's ratio = 0.475); tensile yield strength: 250 MPa; compressive yield strength: 250 MPa; tensile ultimate strength: 460 MPa;

For the rest of the materials employed in the project all the properties were imported from the ANSYS Workbench materials library. The transient structural analysis determines the dynamic response of the structure to the time-varying forces based on an implicit calculation scheme of the equations of motion [5]. [6]. The main results obtained from the transient analysis are: displacements, equivalent strains, equivalent stresses, force reactions, contact forces, etc. This analysis type can also be used to monitor the operation of the assembly throughout the kinematic cycle for a limited time, for which the functional simulation is performed.

The most important issues when running a transient analysis are: the appropriate choice of the material properties and material behavior laws, the accurate definition of the contact types between the components, and the right definition of the kinematic joints between the components. The kinematic joint motion between components can be programmed according to time duration of the simulation.

Because in the transient structural analysis the time response of the structure is nonlinear and the analysis involves large displacements of the parts in the computational model, the parameter of large deformations is considered as active.

In addition, nonlinearities occur during the simulation due to the interaction conditions and contact between parts by taking into account friction and contact stiffness, as well as the system damping. Therefore, the time step controls are established in the following way:

- duration of the time step: t = 1 s;
- initial time step: Δt = 0,05 s;
- minimum time step: Δt = 0,05 s;
- maximum time step: Δt = 0,1 s.

These values may vary depending on the studied phenomena, being higher in the case of smaller nonlinearities of the model and a very good mesh, or lower otherwise. In addition, during the simulation the contact between the components can be activated or deactivated using the “Contact step control” advanced functionality. The simulation starts with preliminary checks of the geometry integrity (the model topology) and mesh quality criteria.

The components for which the mesh is intended to be disregarded during simulation, such as laptop case, keyboard, etc., their behavior has been set to “Rigid”. The second step in performing the simulation is to define the interaction between the components, either contacts or kinematic joints. The links between the parts that have been defined are:

- Fixed joint between the tested device and the worktable;
- Fixed joint between the touch panel, screen, and keyboard membrane and the tested device;
- Bonded contact - securing the lower finger of the effector with the brass fingertip;
- Translational joint between the upper housing and the lower finger of the end-effector;
- Bonded contact between the brass fingertip and the rubber sheath;
- Frictional contact between the end-effectors and the tested device;
- Spring joint - the elastic connection between the upper housing and the lower finger using a helical spring.

Regarding the boundary conditions of the structural elements the following assumptions were done (Fig. 6):

- the touch panel has been realistically attached to the case, leaving the buttons free;
- the membrane of the key was fixed on the entire circumference of its base;
- the screen was fixed by recessing it using its side faces.

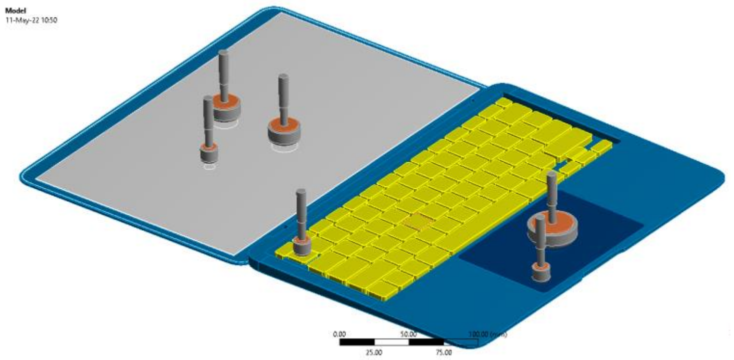


Fig. 6. Assembly in ANSYS Mechanical

To improve the default generated mesh to obtain accurate simulation results, the following mesh settings have been considered:

- the size of the mesh is different for each element, depending on its size and importance in the simulation;
- the mesh is mapped to the surfaces that allows this technique;
- the mesh for the upper housings, the lower fingers of the end-effectors, the brass fingertips, the rubber sheaths, the membrane of the pressed key and the housing of the pressed key is created with dominant hexahedral elements;
- the mesh in the contact area complies compatible mesh requirements between parts, such as: the mesh on the outer surface of the rubber sheath is projected to the touched surfaces.

Finally, using the global mesh quality criterion (Fig. 7) the mesh has reached an average value of 75.44%.

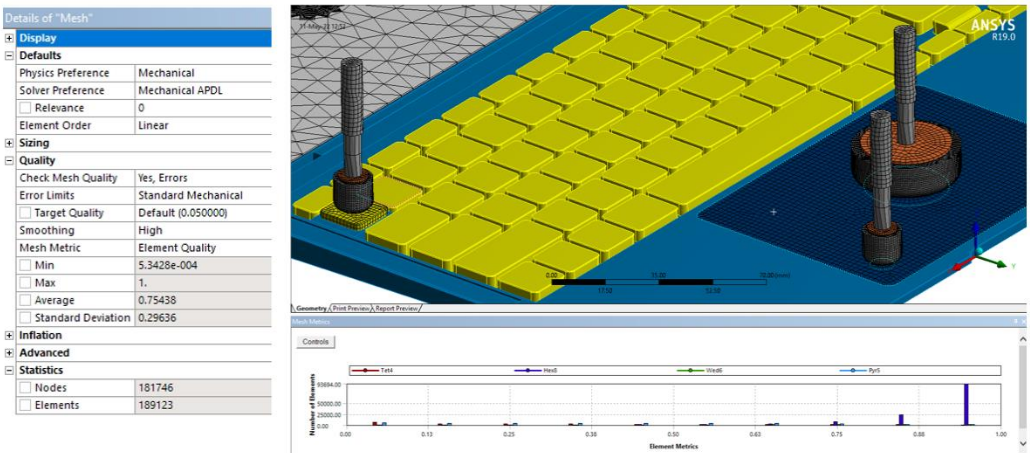


Fig. 7. Mesh and mesh quality and its graphical representation

Loads and constraints. In order to define each load pair and contact action as a function of time, it is compulsory to measure the initial distance between the parts that will come in contact and the movement of the mobile element (end-effector) with the added pressured distance. This distance was measured using the "Distance finder" command in the ANSYS Workbench Design Modeler. Remote displacements were defined to simulate the negative Z-axis movement of the robot and control the contact activity between each end-effector and the laptop.

The simulation solution was performed on a computer with the following specifications: CPU: Intel Core i5-3570K @4.0-4.4 GHz; RAM: 32 GB DDR3; SSD: 981 GB. The total elapsed time was 8 hours and 30 minutes. The results were obtained after 474 iterations (Fig. 8), after 10-time steps and a single bisection occurred during the simulation. Each time step was calculated from 0.05 to 0.05 seconds, according to the initial settings of the analysis. The bisection occurred when the 30 mm end-effector is separated from the touch panel and can be neglected due to the good overall global response of the assembly.

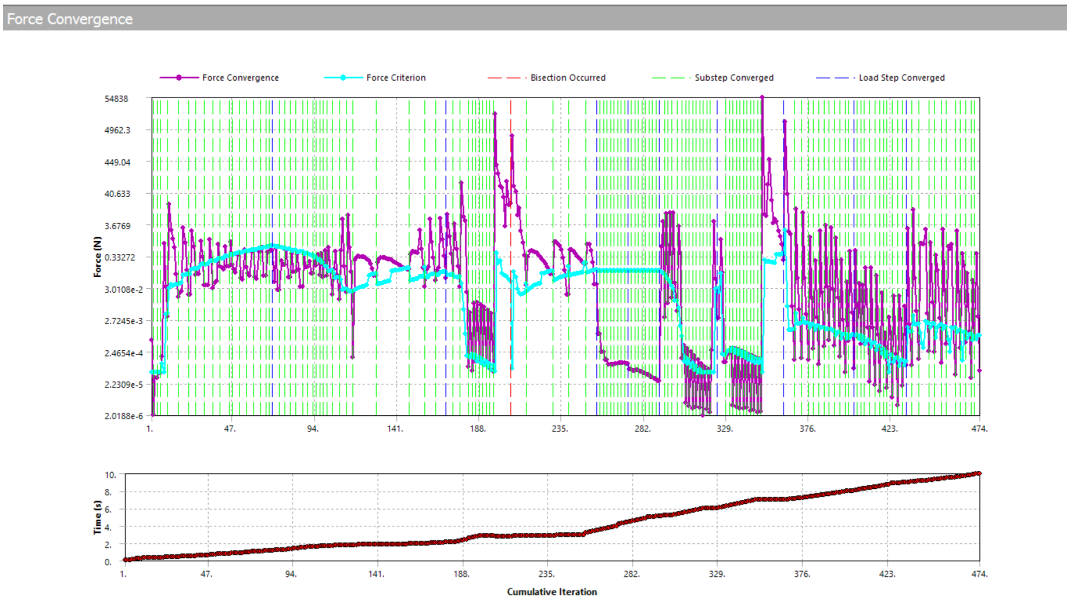


Fig. 8. Graph representing the evolution of the forces over time (force convergence)

The results were customized as follows: for the rubber sheaths of the end-effectors and the membrane of the key the equivalent elastic strains was processed (Fig. 9 and Fig. 12); - for the touch panel and the touch screen both the equivalent stress and the directional displacements on the Z-axis (Fig. 10, Fig. 11, and Fig. 13 to Fig. 16).

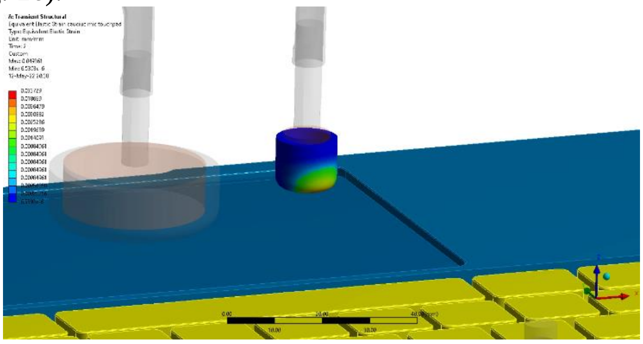


Fig. 9. The maximum elastic strain of the 10 mm rubber sheath when pressing the touch panel is 3.7%

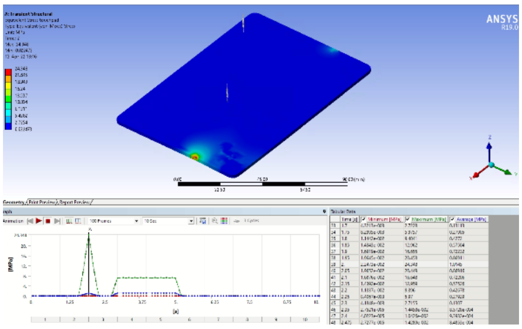


Fig. 10. The maximum equivalent von Mises stresses of the touch panel is 24.348 MPa

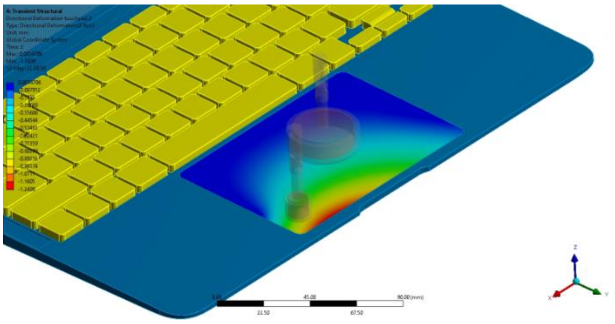


Fig. 11. The maximum deformation on the Z-axis of the touch panel is -1.2499 mm

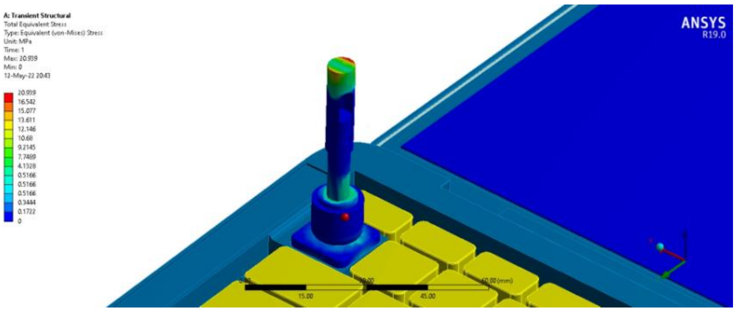


Fig. 12. The maximum equivalent stresses in the pressed key appear in the fixing area of the end-effector and are equal to 20,939 MPa

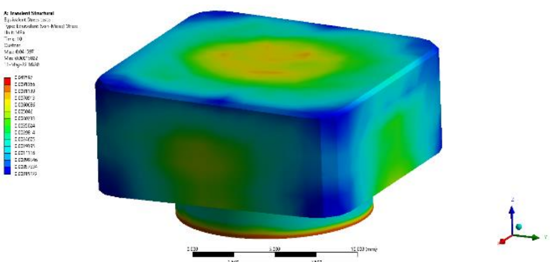


Fig. 13. Equivalent stresses occur in both the pressed key and the membrane; the maximum stress is equal to 0.04 MPa

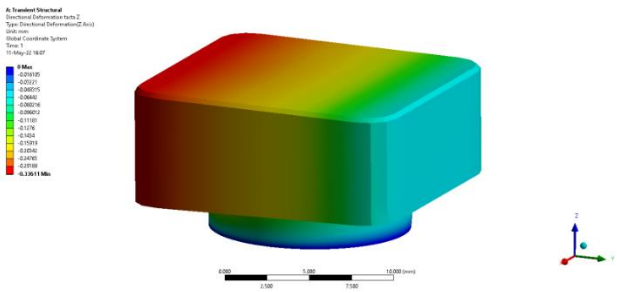


Fig. 14. Maximum directional deformation on the Z-axis of the pressed key occurs when it’s pressed, and the deformation is equal to -0.33 mm

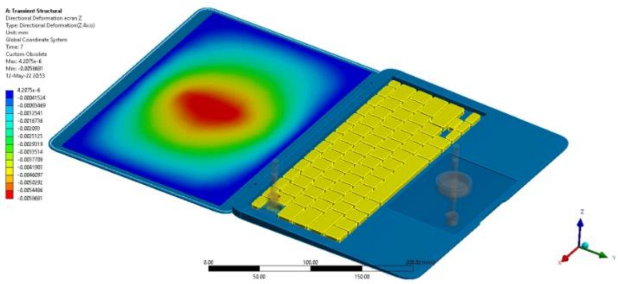


Fig. 15. Maximum deformation on the Z-axis of the screen occurs when it's pressed, and the deformation is equal to -0.006 mm

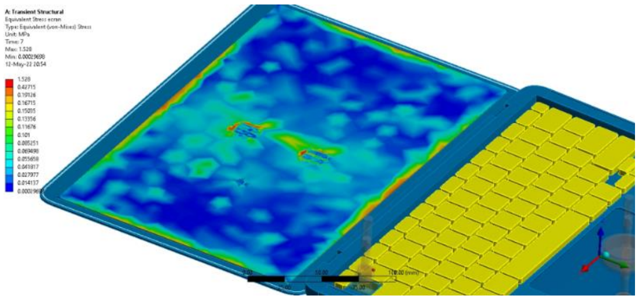
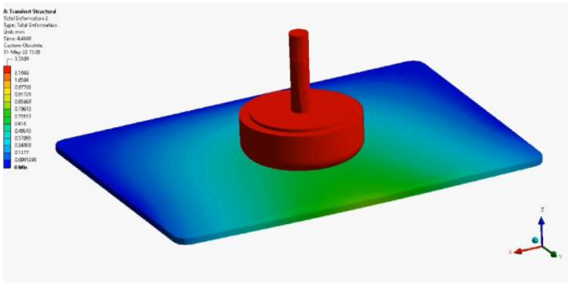


Fig. 16. Maximum stresses occur in the screen when pressed and are equal to 1.53 MPa

4. Conclusions

The transient structural analysis shows that the behavior of the elastic elements can be accurately analyzed and investigated over time and the simulated response of the system can be synchronized with the forces monitored on the real test bench.

a- real life touch panel confidence test [7]



b- simulated touch panel confidence test along with its results

Fig. 17. The touch panel confidence test – detecting involuntary human touches

The purpose of this research is to verify the laptop design when operating at extreme conditions. The possibility of material replacement for different components can be taken into account, but the results proved that for the analyzed system this was not needed.

FEM demonstrated its capabilities for creating virtual conditions for testing a laptop's components in order to check if it meets the hardware manufacturers, and the customer's requirements.

5. Bibliography

[1]. Țîmpea Ș., Cosma C., Șoșdean D. (2019), "Touch Screen Tester Device End-Effector", volume 56 (issue 2), pages 444-448, Materiale Plastice magazine.

[2]. *** Tactile Automation Official Youtube Channel, updated on May 12nd 2022

[3]. *** Tactile Automation – Spring Finger Datasheet & Fingertip Datasheet, updated on May 12nd 2022

[4]. *** AZO Materials – datasheets for rubber, brass, and polycarbonate, article numbers 6380, 920, 1934, updated on May 12nd 2022

[5]. *** Wikipedia – Augmented Lagrange Method

[6]. *** ANSYS Theory Manual, V19.0

[7]. *** www.upload.wikimedia.org/wikipedia/commons/6/6e/Membrane_keyboard.jpg