Proceedings in Manufacturing Systems
Proceedings in Manufacturing Systems, Volume 7, Issue 3, 2012 ISSN 2067-9238

DROP TEST OF ELECTRONIC DEVICE BY VIRTUAL PROTOTYPE SIMULATION

Georgi TODOROV[1], Borislav ROMANOV[2], Konstantin KAMBEROV[3,*], Svetoslav STOEV[4]

[1] Prof. PhD Eng., Dean, Faculty of Industrial Technologies, Technical University, Sofia, Bulgaria
[2] Eng., Research Engineer, Laboratory "CAD/CAM/CAE in Industry", Faculty of Industrial Technologies, Technical University, Sofia, Bulgaria
[3] PhD Eng., Research Engineer, Laboratory "CAD/CAM/CAE in Industry", Faculty of Industrial Technologies, Technical University, Sofia, Bulgaria
[4] Eng., Research Engineer, Laboratory "CAD/CAM/CAE in Industry", Faculty of Industrial Technologies, Technical University, Sofia, Bulgaria

Abstract: *Contemporary products are subjected to increased requirements for reliability and safety that reflect as new standards and norms. The development of numerical techniques based on virtual prototyping technology facilitates their application in a wide variety of new designed products. Implementing virtual prototyping in product development process results in reduced time/money spent for this stage as well as increased knowledge about certain failure mechanism. So called "drop test" became nearly a "must" step in development of handheld equipment, which could be very time-consuming task because of its nature. This study aims to present a simplified approach for fast design verification at its early stage – before final verification of the product and its documentation. It uses a quasi-static analysis approach to solve a typical dynamic task, instead of usual explicit dynamic procedure, which is related to time consuming nonlinear analysis. This two-step approach involves initial analysis to calculate dynamic factors for a final steady-state static analysis that results in sufficient like accuracy data. Two major advantages of this approach are the decreased overall simulations time – reaches more than 10 times reduction in some cases – and avoid of nonlinearity that could be a certain error source. Certain example is shown, based on a research study of electronic device, subject of impact loads during drop test.*

Key words: *simulation, drop test, virtual prototype, electronic device, FEA.*

1. INTRODUCTION

Electronic devices have been developed in a wide range in the past years, especially portable, hand held products. They are easy to handle and use but has a risk of being dropped more often. Thus, the most common failures are due to drop impact. Contemporary products are required to withstand such type of loadings at levels, described in international and local standards. Thus, drop impact becomes a must evaluation for reaching tradable item design [4]. Usual stage in the Product Development Process (PDP) involves experimental testing, which is high cost and time consuming process. Design details cannot be considered based on physical testing as it is difficult to measure responses at any locations, especially for small regions.

Virtual Prototyping (VP) is a technology that offers possibilities for higher level of exploration of physics-of failure. Simulations of the failure mechanism, based on VP, are a powerful tool for design improvement. Such a simulation has of course the advantage that the cost for a "numerical test" is significantly lower than for an actual drop test, but the numerical simulation gives also a better understanding of the underlying physics and allows the user to check rapidly the influence of specific parameters. The numerical simulations are also very useful in design since they allow the influence of modifications and different parameters to be assessed, directly, without spending time on prototype manufacturing.

The analysis of a drop test is a highly nonlinear and dynamic event and it is difficult to prove the accuracy and completeness of the results. The drop test is very complex to analyse and computationally extremely costly since it involves large deformations, dynamics and material failure. The tools for impact analysis have developed significantly and numerical simulations are now performed before performing required experiments [2, 5].

Usually, these numerical simulations are based on explicit nonlinear simulation techniques. These types of analyses could be very time expensive and require powerful computing resources. Additionally, because of its nonlinear nature, performed explicit analysis could give results with certain inaccuracy. The detailed output data is not necessary usually just for the performed design check. Preliminary simulations (at conceptual model stage of the PDP), or even some final design simulations, could require fast and cost effective solution rather than detailed, exact one. Possible way to solve out this problem is to apply steady-state static analysis which uses implicit analysis technique that will use fully linear computation of the examined mechanics.

This approach involves quasi-dynamic physics, where force/deflection loads correspond to applied kinematic energy of the moment of impact of the explored body to

---

[*] Technical University – Sofia, 8 "Kl. Ohridski" Blvd.,
Sofia, BULGARIA,
Tel.: +359 2 965 25 74;
Fax: +359 2 965 25 74.
E-mail addresses: *gdt@tu-sofia.bg* (G.Todorov),
bromanov@tu-sofia.bg (B. Romanov),
kkamberov@3clab.com (K. Kamberov),
sstoev@3clab.com (Sv. Stoev)

the ground. The approach is presented in detail in the current study, based on an example of impact analysis of electronic device [1 and 3].

2. SIMULATION APPROACH

The approach is based on the assumption that the complete kinetic energy is transferred in potential energy during the impact. It will be demonstrated based on sample problem of free falling of an electronic device (which has proper mass and rigidity distributions in the examined geometry volume) from 1 meter distance. The kinetic energy of a free falling body could be expressed as dependent on mass and height as is stated below:

$$E_k = \frac{m \cdot V^2}{2} = m \cdot g \cdot d \,, \qquad (1)$$

where:

$m = 34.1 \cdot 10^{-3}$kg – the mass of the falling object (the electronic device for current study);

$g = 9.8056$ m/s^2 – gravity;

$d = 1$ m – travel distance, 1 meter for the current research.

Thus, the kinetic energy value will be:

$$E_k = 0.334\ J\ . \qquad (2)$$

The potential energy could be calculated as follows:

$$E_p = R \cdot \Delta, \qquad (3)$$

where:

R – reaction force;

Δ – deformation in reaction force direction.

Generally, the directional structural stiffness is constant and thus:

$$E_p = R^2 / c, \qquad (4)$$

where:

$c = R_{initial} / \Delta_{initial}$ is the constant directional stiffness of the examined structure, determined by an initial analysis.

On the other hand, both kinematic and potential energy are equal in the moment of impact and so the needed acceleration, to be applied over examined structure, could be calculated as:

$$\frac{R^2}{c} = m \cdot g \cdot d \,, \qquad (5)$$

where:

$R = m \cdot a$ (a is the applied final acceleration), or:

$$a = \sqrt{\frac{c \cdot g \cdot d}{m}}\ . \qquad (6)$$

So, performed assumptions require initial analysis to be run in each of the main 3 directions, with applied sample acceleration, to find stiffness c of the structure. Next step is to calculate final required acceleration to be applied over the examined structure and to obtain final results data. Thus, overall number of performed analyses is six.

3. FE SIMULATION MODEL

An existing design of POM housing with PCB mounted on it is examined, according to the generated in the design process documentation and 3D models (see Fig. 1). The main goal of this study is to identify the force-deflection behaviour of mounting pins. Two of all four pins are thermally treated during PCB mounting and are the main connecting links to the housing that secure the PCB. The complete product is to be tested under impact loads.

The existing geometry models are directly used for FE mesh input. Received geometry models are imported without modifications in the finite element modeller and a mesh structure is built, based on them. Solid FE model is created, using type of element with 20 nodes (middle node on side edge) that allow more detailed behaviour simulation. Contact elements are generated in common boundaries between the PCB and the housing – pins/holes and supporting ribs/bottom PCB surface. Meshed structure is shown on Fig. 2 as to present the density of the mesh. The meshed structural model contains 161 000 nodes and 70 000 elements approximately.

The materials used in the design are two types – POM – for the housing, and common for PCBs properties (material FR-4), as its behaviour is not in the focus of this study. Material properties of POM, used in the presented analyses are defined as follows: Young modulus: E = 2.7 G Pa; Poisson's ratio: μ = 0.34; density: ρ = 1 410 kg/m^3.

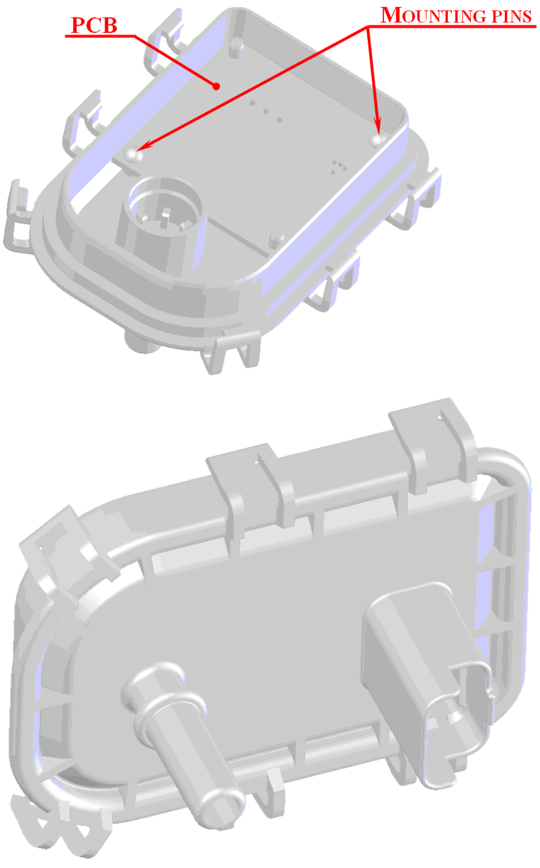

Fig. 1. Geometry model of examined electronic device.

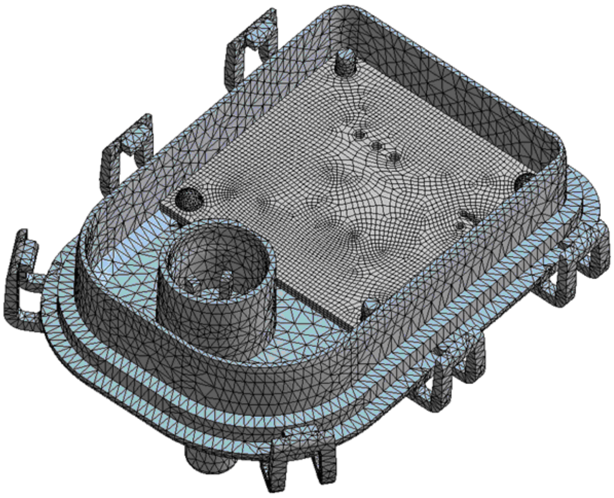

Fig. 2. Generated mesh model.

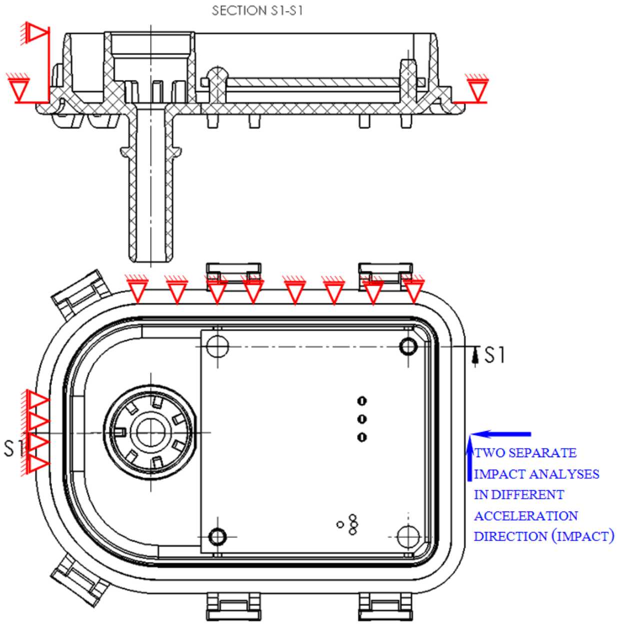

Fig. 3. Applied boundary conditions.

The applied boundary conditions are shown on Fig. 3. They represent impacts at both horizontal directions subsequently and additionally for the vertical *X* axis. The degrees of freedom represent mounting of the housing by its edge surfaces. The vertical support is applied on the complete flange area, while in horizontal plane only two surfaces are constrained – one at each direction. All constraints are unidirectional.

Six analyses are performed as follows:

- three initial – each in separate planar direction and in vertical direction – to determine the structural stiffness in proper direction;
- three final – in both perpendicular planar directions and in vertical direction too – to determine final stress distributions at the moment of impact.

4. SIMULATIONS RESULTS

Sample results from the performed first three analyses are shown. The performed simulations for determining the impact accelerations in both planar directions are for *initial* acceleration value and the results are shown as deformations in acceleration directions for both horizontal plane axes – *Y* and *Z* accelerations applied – on Figs. 4 and 5 below. These figures mainly shows the character of deformation and also helps to determine the rigidity of examined structure – in each of modelled directions of impact.

Thus, the results show the following maximal displacement values for the main body of the casing:

- in-plane axes:
  - o *Y* axis → $c = 3.08 \cdot 10^6$ N/m; $a_{final} = 3\ 022$ m/s^2;
  - o *Z* axis → $c = 2.4 \cdot 10^6$ N/m; $a_{final} = 2\ 670$m/s^2;
- vertical *X* axis → $c = 2.27 \cdot 10^6$ N/m; $a_{final} = 2\ 594$ m/s^2.

Different values in certain directions in fact are due to the different rigidity of the structure.

Calculated accelerations values are used as input data for the next three simulations – each in separate acceleration direction. The results of the performed new steady-state (impact) analysis, using determined in previous step acceleration values along three main axes, are shown as equivalent (von Mises) stress distributions on Figs. 6, 7 and 8.

All the three sides of the housing are examined under impact loads, reproduced by applied accelerations subsequently. Maximal equivalent stress for the in-plane accelerations directions occur in the pins (especially in transverse direction) – about 8 MPa. Vertical direction of impact results in about 6 MPa. Generally, there are no stress values over 10 MPa, and no critical behaviour is found for the examined design.

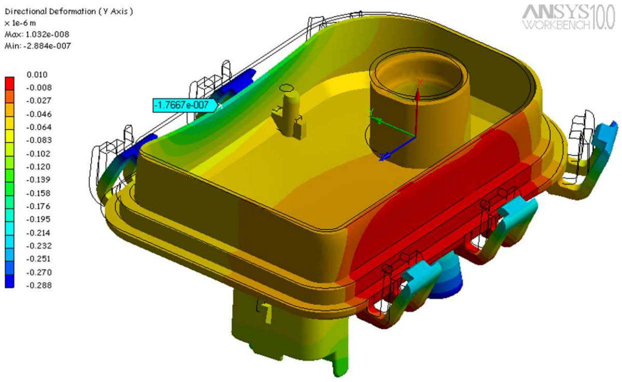

Fig. 4. Transverse direction of impact; in-plane directional deformations, m.

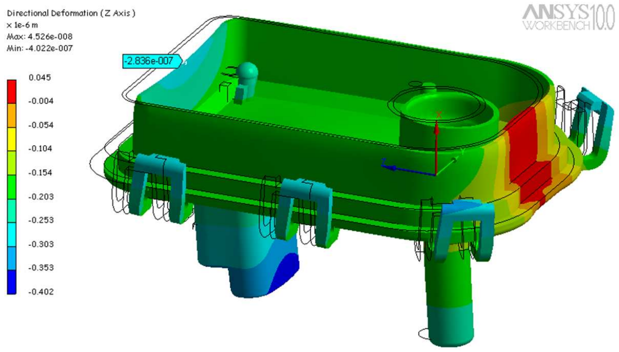

Fig. 5. Longitudinal direction of impact; in-plane directional deformations, m.

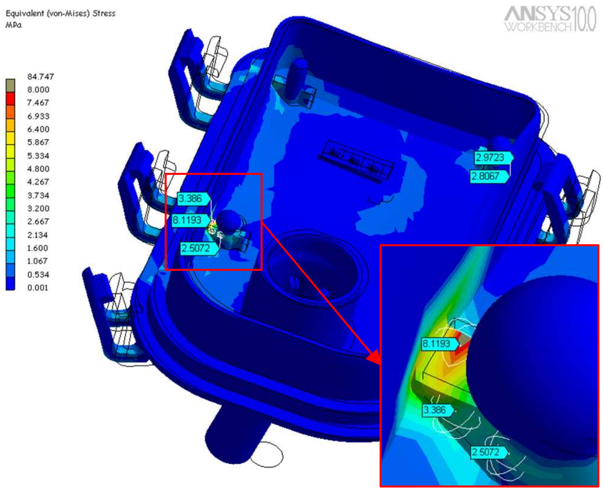

Fig. 6. Impact final analysis along in-plane *Y* axis: equivalent stress distribution, MPa.

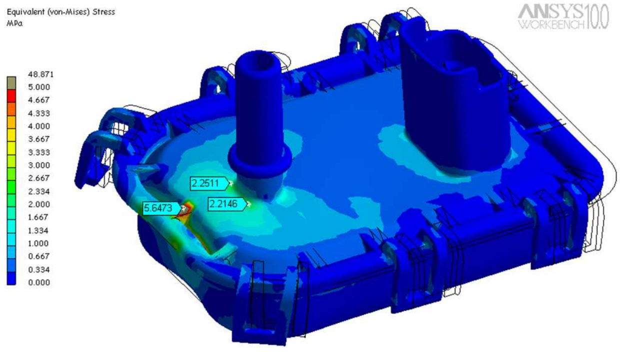

Fig. 7. Impact final analysis along in-plane *Z* axis: equivalent stress distribution, MPa.

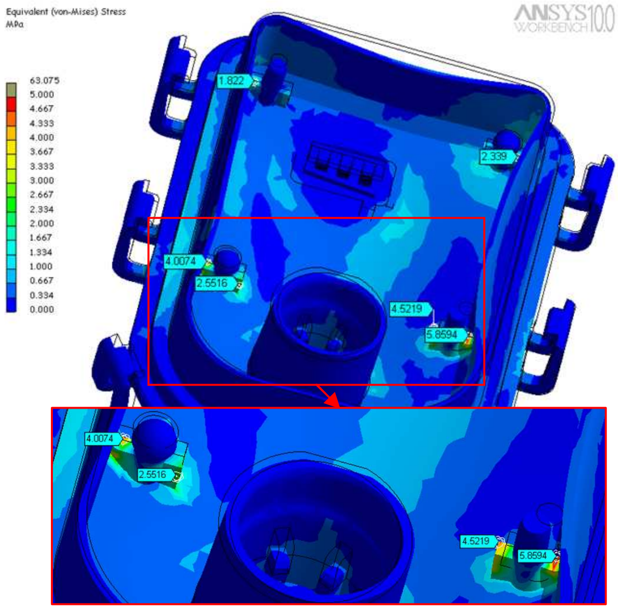

Fig. 8. Impact final analysis along vertical *X* axis: equivalent stress distribution, MPa.

5. CONCLUSIONS

Dynamic behaviour assessment of a structure under impact loads is typical nonlinear, explicit solution problem, which involves significant project resources. Demonstrated approach in this study uses linear technique to reproduce same results and is based on simple "worst case" type assumption for energy transfer during the examined process.

Solution linearity decreases significantly needed time and computational resources on the price of receiving worst scenario results. This allows multiple design variants to be explored, without spending a lot of resources - both computational and human. Additional advantage is that nonlinear results accuracy very often is comparable to linear substitution model.

Generally, the proposed approach allows the designer to evaluate even conceptual model, on its early stage, to compare and to find optimal solution among concurrent designs, on a very cheap price (time), compared to conventional nonlinear techniques. This technique could be concurrent even in some final stage design assessments, especially for large assemblies, subjected on impact loads, where ordinary explicit solution would be too expensive.

Future work in this direction would include verification of the proposed approach by a standard explicit analysis and by performed experimental studies over the same design.

ACKNOWLEDGEMENTS: This research study is performed by the support of project DUNK-01/03 "University Scientific and Research Complex" of National Science Fund, Ministry of Education, Youth and Science, Bulgaria.

REFERENCES

[1] D. Neumayer, M. Chatiri, M. Höermann, *Drop Test Simulation of a Cooker Including Foam Packaging and Prestressed Plastic Foil Wrapping*, 9th International LS-DYNA Users Conference, pp. 18-33 – 18-40, 2006, Detroit, USA.

[2] G. Todorov, Y. Lai, K. Kamberov, *Development of Collaborative Approach for Virtual Prototyping based on 3D kernel and FEM analysis*, Wissenschaftlichen Konferenz "Technik und Wirtschaft in der globalen Krise", pp. 11–16, Nov. 2009, Sofia, Bulgaria.

[3] L. Qiao, U. Zencker, G. Wieser, and H. Völzke, *Numerical Safety Assessment of a Transport and Storage Cask for Radioactive Materials without Impact Limiters by the 0,3m Drop Test onto an Unyielding Target*, 9th International Conference on Computational Structures Technology, pp. 64–68, Athens, Greece, September 2–5, 2008.

[4] H.-L. Wang , Sh.-Ch. Chen, L.-T. Huang, Y.C. Wang, *Simulation and Verification of the Drop test of 3C Products*, 8th International LS-DYNA Users Conference, pp. 14-7 – 14-18, 2004, Dearborn, USA.

[5] A. Connel, *The Rise and Rise of Virtual Prototyping*, Time-Compression Technologies, vol. 12, iss. 3, 2004, pp. 37–43.