# Numerical Analysis of a Plastic Deformation Composite Panel

**ADRIAN PRESURA***
Dunarea de Jos University of Galati, Faculty of Engineering, 47 Domneasca Str., 800008, Galati, Romania

*The paper is focused on the comparison between the experimental test and numerical analysis of a composite panel wich is subject to plastic deformation. An experiment was done on a sandwich panel made from steel – extruded polystyrene – steel, wich was subjected to a static plastic deformation with help of an sferical indenter. During the test the force applied to the indenter and the vertical displacement of the steel panels were measured. The experimental values were compared with the results of numerical simulation, wich was done with help of ANSYS-Static Structural module. Important aspects of FEA, such as material idealization, contact approach between bodies, mesh size and boundary conditions, are discussed and their influence on the results are highlighted. The results of this study are vey useful for investigation of beahviour at impact of composite materials such as steel-polystyrene sandwich panels.*

*Keywords: FEA, plastic deformation, composite panels, internal energy, experimental validation*

The main goals of this study were:

- to investigate the influence of different parameters of numerical analysis about the results, and
- to determine the level of approximation that can be achived between the FEA and experimental test.

In order to analyse the behavior of the sandwich panel under transverse force, first was designed and built an experimental stand, based on a screw press mechanism.

The stand is composed from (fig. 1):

- stand frame – made from UNP80 profile;
- screw mechanism–M24 screw;
- force transducer–50kN maximum;
- displacement transducer–300mm maximum;
- spherical bulb-ϕ60mm;
- sandwich panel with UNP60 profile frame.

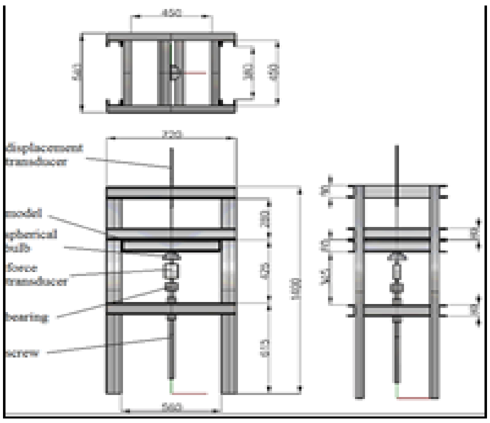


Fig.1 Experimental stand

The plastic deformation of the panel was done based on the vertical displacement of the spherical bulb, acted by rotating the screw.

## Experimental part

The test consist in plastic deformation of a composite panel made from (fig. 2):

- upper steel sheet – 1.5 mm thickness;
- extruded polystyrene – 20 mm thickness;
- lower steel sheet – 1.5 mm thickness.

During the experiment three values were measured:

- force applied to the spherical bulb;
- vertical displacement of the lower steel sheet, with help of the screw thread;
- vertical displacement of the upper steel sheet, with help of displacement transducer.

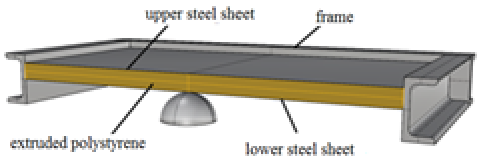


Fig.2 Sandwich panel

In figure 3 below is reprezented the panel mounted on the stand at the begining of the test.

Fig.3 Spherical bulb acting on the panel

The values recorded during the experiment are presented in the table 1 below.

**Table 1**
EXPERIMENT MEASURED VALUES

| Dsipl. lower sheet | Correction | Displ. corrected lower sheet | Displ. upper sheet | Force | Distance between upper and lower sheets |
|---|---|---|---|---|---|
| [mm] | [mm] | [mm] | [mm] | [kN] | [mm] |
| 0 | 0.000 | 0.000 | 0 | 0 | 23.000 |
| 3.000 | 0.011 | 2.989 | 0.178 | 0.779 | 20.189 |
| 6.000 | 0.036 | 5.964 | 1.645 | 2.678 | 18.681 |
| 9.000 | 0.061 | 8.939 | 2.881 | 4.515 | 16.942 |
| 12.000 | 0.091 | 11.909 | 3.920 | 6.700 | 15.011 |
| 15.000 | 0.123 | 14.877 | 4.860 | 9.060 | 12.983 |
| 18.000 | 0.158 | 17.842 | 5.740 | 11.640 | 10.898 |
| 21.000 | 0.196 | 20.804 | 6.560 | 14.400 | 8.756 |
| 24.000 | 0.235 | 23.765 | 7.370 | 17.300 | 6.605 |
| 27.000 | 0.275 | 26.725 | 8.130 | 20.250 | 4.405 |
| 30.000 | 0.320 | 29.680 | 9.100 | 23.520 | 2.420 |
| 33.000 | 0.370 | 32.630 | 10.880 | 27.200 | 1.250 |
| 36.000 | 0.425 | 35.575 | 13.530 | 31.230 | 0.955 |
| 39.000 | 0.482 | 38.518 | 16.340 | 35.460 | 0.822 |
| 42.000 | 0.540 | 41.460 | 19.110 | 39.700 | 0.650 |
| 45.000 | 0.597 | 44.403 | 21.910 | 43.870 | 0.507 |
| 42.000 | 0.283 | 41.717 | 19.650 | 20.800 | 0.933 |
| 39.000 | 0.070 | 38.930 | 17.440 | 5.120 | 1.510 |
| 37.251 | 0.000 | 37.251 | 16.600 | 0.000 | 2.349 |

* email: adrian.presura@ugal.ro

Because the vertical displacement of the lower steel sheet was measured using the screw thread, a correction was applied to this values, that takes into account the screw and the stand own deformation. The correction is directly proportional to the force applied and has the maximum value of 0.68 mm at maximum force of 50 kN.

From the force-displacement diagram, reprezented in figure 4 below, it can be observed a much smaller deformation of the upper steel sheet, due to presence of the extruded polystyrene.

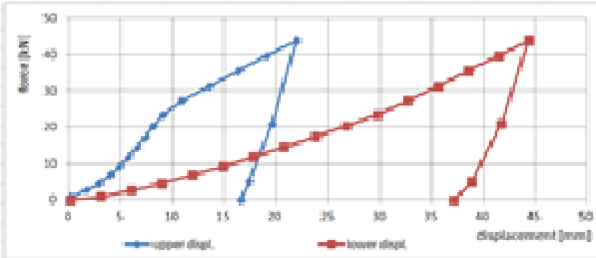


Fig.4 Force-displacement diagram

The same difference can be observed in the figure 5 below, where is presented the final plastic deformation of the lower steel sheet in the left and of the upper steel sheet in the right.

Fig.5. Panel plastic deformation

An important quantity of plastic deformation phenomena is the internal energy of the structure.

The energy-displacement diagram of the composite panel obtained after the experiment is depicted in figure 5 below.

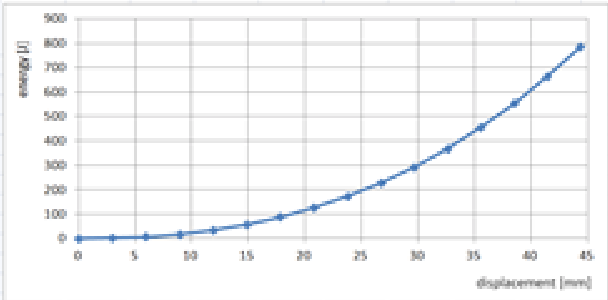


Fig.6. Energy-displacement diagram

Also as part of the experiment was the determination of the mechanical properties of the steel used at the fabrication of the upper and lower sheets of the sandwich panel.

A number of three specimens were tested on the tensile test machine as shown in the figure 7 below.

Fig.7. Tensile test of steel specimen

The geometry of the specimens were realized according SR EN100002-1:2001 – Annex B – Types of test pieces to be used for thin products: sheets, strips and flats between 0.1 mm and 3 mm thick (fig. 8).

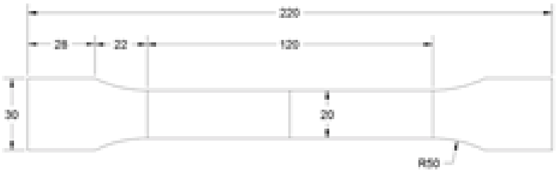


Fig.8. Steel specimen

The resulting force-elongation diagrams of the three specimens are figured below.

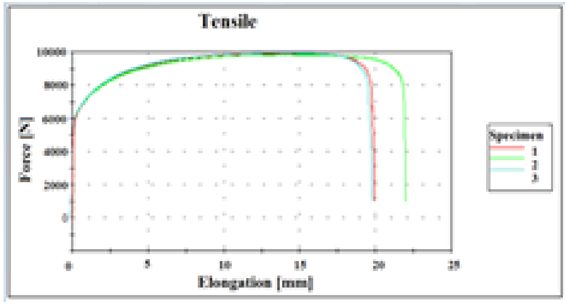


Fig.9. Force-elongation diagram-steel

The mediate value of the three curves presented above was the final stres-strain diagram of the steel used at the numerical simulation.

The mechanical characteristics of the extruded polystyrene was taken from Kulzep, the afferent stress-strain diagram is reprezented in figure 10 below.

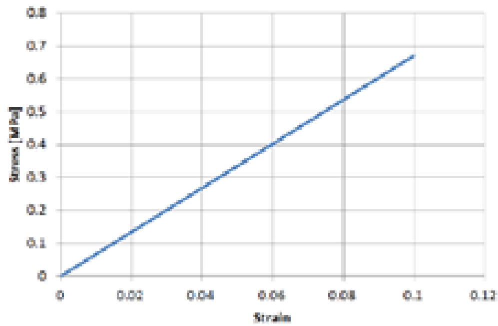


Fig.10. Stress-strain diagram-polystyrene

*Numerical analysis*

For numerical analysis of plastic deformation experiment, it was used a model for quarter of the panel and spherical bulb, in order to reduce the computational time (fig. 11).

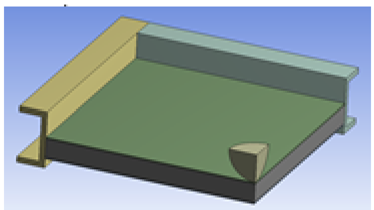

Fig.11 Model used for simulation

Having as reference the force and vertical displacement values measured during experimental test, it was investigated the influence of different parameters of numerical analysis:

- material idealization
- contact approach between bodies
- mesh size
- boundary conditions.

The scope of this comparative investigation was to establish the numerical simulation parameters that lead to best approximation of experiment results.

Material idealization

Two different idealization were approached (fig. 12):

- bilinear isotropic hardening;
- multilinear isotropic hardening.

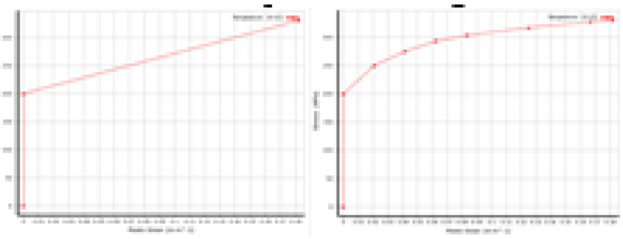

Fig.12. Material idealization bilinear (left) and multilinear (right)

The mechanical characteristics of the tested steel were:

- Young's modulus = $2.1\times10^5$ N/mm$^2$;
- yield strength $R_{eH}$ = 200 N/mm$^2$;
- tensile strength $R_m$ = 335 N/mm$^2$;
- ultimate elongation A = 0.182.

The numerical simulations revealed the following results presented in table 2 below.

**Table 2**

BILINEAR-MULTILINEAR COMPARISON

| | Experiment | Simulation | |
|---|---|---|---|
| | | Bilinear | Multilinear |
| Force [kN] | 43.87 | 37.32 | 39.68 |
| Diff. [%] | - | 14.9 | 9.6 |
| Displ. [mm] | 44.4 | 38.3 | 38.1 |
| Diff. [%] | - | 13.7 | 14.2 |
| Time [s] | - | 684 | 984 |
| Diff. [%] | - | - | 43.9 |

The comparative energy-displacement diagram for experiment and simulation with bilinear and multilinear idealization is presented in figure 13 below.

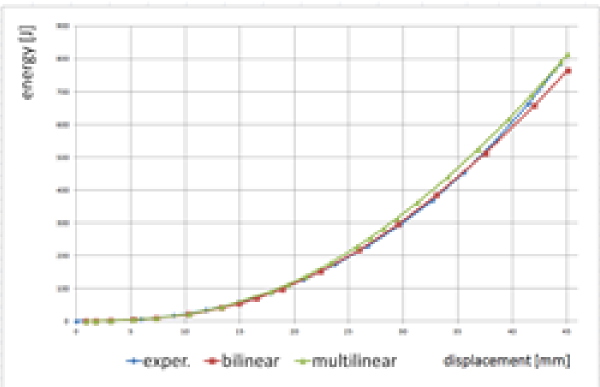


Fig.13 Internal energy bilinear and multilinear

The following conclusions can be drawn:

- the multilinear idealization leads to more precisely force simulation (5% better than bilinear);
- both solutions lead to similar displacement results;
- the computation time is significant bigger (44% to bilinear) for the multilinear idealization;
- the bilinear idealization is more conservative, leading to a total energy with 3% smaller than the experiment.

Contact between bodies

Two different contact approaches between shperical bulb and the model were investigated:

- frictionless;
- frictional, with friction coefficient 0.3.

From the analysis results, presented in table 3 below, can be observed:

- an insignificat difference of 1% for the force calculation;
- practical no difference for vertical displacement ;
- an important major computational time for the frictional approach (31% higher).

**Table 3**

BODIES CONTACT COMPARISON

| | Experiment | Simulation | |
|---|---|---|---|
| | | Frictionless | Frictional |
| Force [kN] | 43.87 | 36.92 | 37.32 |
| Diff. [%] | - | 15.8 | 14.9 |
| Displ. [mm] | 44.4 | 38.26 | 38.3 |
| Diff. [%] | - | 13.8 | 13.7 |
| Time [s] | - | 523 | 684 |
| Diff. [%] | - | - | 30.8 |

Mesh size

Four different mesh sizes were investigated: 45 mm, 22 mm, 11 mm and 7 mm (fig. 14).

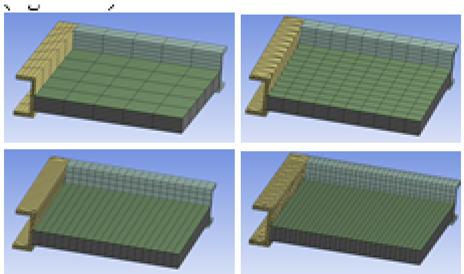

Fig.14 Mesh sizes

The difference of plastic deformation between the biggest mesh size 45 mm and the smallest mesh size 7 mm can be observed in figure 15 below.

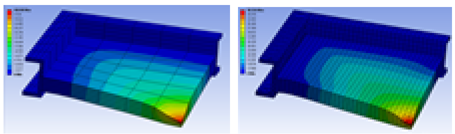

Fig.15 Plastic deformation for 45 mm mesh (left) and 7 mm mesh (right)

The comparative energy-displacement diagram for experiment and simulation with differet mesh sizes is presented in figure 16 below.

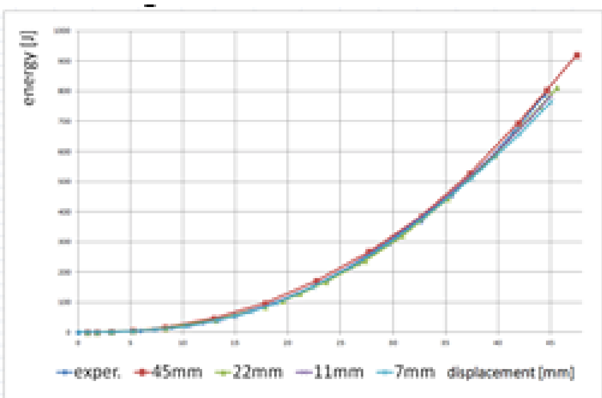


Fig.16 Internal energy for different mesh sizes

From the calculations results presented in table 4, for different mesh sizes, it can be concluded that the 45 mm mesh size offers the closest results for maximum force and final deformation and the 7 mm mesh size leads to best energy approximation .

**Table 4**
MESH SIZE COMPARISON

| | Exper-iment | Simulation | | | |
|---|---|---|---|---|---|
| | | 45 | 22 | 11 | 7 |
| Force [kN] | 43.87 | 46.6 | 40.2 | 38.7 | 37.3 |
| Diff. [%] | - | 6.2 | 8.3 | 11.8 | 15.0 |
| Displ. [mm] | 44.4 | 40.9 | 38.6 | 38.1 | 38.3 |
| Diff. [%] | - | 7.9 | 13.1 | 14.2 | 13.8 |
| Time [s] | - | 20 | 58 | 289 | 684 |

Boundary conditions

Two different boundary conditions have been used:

- all displacements and all rotations fixed on the panel contour (fig. 17);

- vertical displacement fixed on the panel frame (fig. 18).

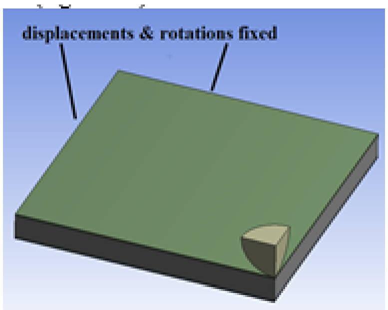


Fig.17. Panel contour fixed

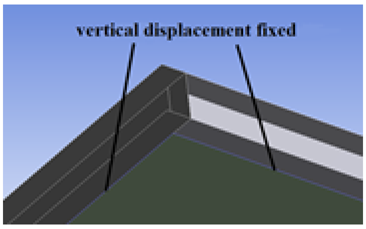


Fig.18. Frame vertical displacement fixed

The numerical analysis results for the two different boundary conditions are presented in table 5.

**Table 5**
BOUNDARY CONDITIONS COMPARISON

| | Exper-iment | Simulation | |
|---|---|---|---|
| | | Panel contour | Panel frame |
| Force [kN] | 43.87 | 61.8 | 46.6 |
| Diff. [%] | - | 40.9 | 6.2 |
| Displ. [mm] | 44.4 | 45.0 | 40.9 |
| Diff. [%] | - | 1.4 | 7.9 |
| Time [s] | - | 16 | 20 |
| Diff. [%] | - | - | 25.0 |

From the above results the following conclusions can be drawn:

- the first boundary condition, all displacements and rotations fixed on panel contour, had an substantial negative influence on force value

- the displacement value was better approximated by panel frame boundary condition.

## Conclusions

From the comparative analysis of experiment and numerical simulation resultes, the following conclusions can be drawn:

- bilinear idealization of the material leads to good results with low computation time;

- frictionless contact between spherical bulb and model offers very close results to frictional contact, but with significant smaller calculation time;

- bigger mesh size led to better approximation of maximum force and final deformation, but 7 mm mesh size led to better approximation of internal energy;

- the two different boundary conditions investigated had an greater influence on contact force.

Based on this research, the level of approximation for FEA was found to be of 5-10%, for plastic deformation of a composite panel.

*Acknowledgements: The present research has been supported by Ship Design Group Galati.*

## References

1. *** SR EN100002-1:2001, Annex B, Types of test pieces to be used for thin products: sheets, strips and flats between 0,1 mm and 3 mm thick.
2. *** Kulzep. (2001). Verhalten von ausgeschäumten Schiffsstrukturen bei Kollision und Grundberuhrung. Schriftenreihe Schiffbau. [Arbeitsbereiche Schiffbau], TU.
3. *** ANSYS. (Release 12.0. 2009).
4. CHIRICA, I., 1997, Elasticitate - Fundamente. Exemple. Aplicatii, Editura Tehnica, Bucuresti, ISBN 973-31-1129-5.
5. IONEL GAVRILESCU, 2004, Plasticitate. Fundamente. Calcul numeric, Editura fundaþiei universitare Dunãrea de Jos, 300 pag., ISBN 973-627-177-3,
6. BEZNEA E.F, CHIRICA, I., 2010, Structuri compozite, Editura Galati University Press, Ed. Galati University Press, ISBN 978-606-8008-86-8.
7. CHAKRABARTY J., Theory of plasticity, Elsevier Butterwoth-Heinemann 2006– ISBN 978-0-7506-6638-2
8. CORMOS, R., PETRESCU, H., HADAR, A., ADIR, G. M., GHEORGHIU, H., Finite element analysis of the multilayered honeycomb composite material subjected to impact loading, Mat. Plast., **54**, no. 1, 2017, p. 180.
9. F. ROTARU (PARASCHIV), I. CHIRICA, E-F. BEZNEA, Influence of the Honeycomb Geometry on the Sandwich Composite Plate Behavior, Advanced Materials Research, ISSN: 1662-8985, Vol. 1143, 2017, pp 139-144 Trans Tech Publications

Manuscript received: 13.02.2017