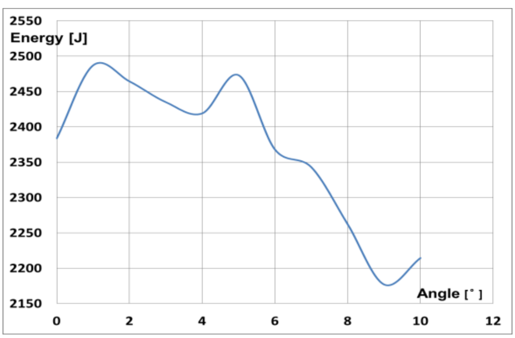
# IMPACT BEHAVIOUR OF SPOT-WELDED THIN-WALLED FRUSTA

**M. KOTEŁKO, A. MOŁDAWA**

Department of Strength of Materials, Łódź University of Technology
Stefanowskiego 1/15, 90-924 Łódź, Poland

## 1. INTRODUCTION

Dynamic response of structures in the plastic range is a significant problem in the case of energy absorbers. Such a structural member converts totally or partially the kinetic energy into another form of energy. One of the possible design solutions is the conversion of the kinetic energy of impact into the energy of plastic deformation of a thin-walled metallic structural member. There are numerous types of energy absorbers of that kind cited in the literature. Among others, there are compressed thin-walled frusta (truncated circular cones or prisms) [1], currently used as impact attenuation members in car structures. A designer of any impact attenuation device must meet two main requirements. The initial collapse load must not be too high in order to avoid unacceptably high impact velocities of the vehicle. On the other extreme, the main requirement is a possibly highest energy dissipation capacity, which may not be achieved if the collapse load of the impact device is too low. The shape of the thin-walled member and the manufacturing technology (e.g. spot welding) influence substantially quantities mentioned above. There are very few fragmented information about an influence of spot welding on the crushing behavior of thin-walled members subjected to axial impact [2]. In the paper the results of the parametric study into the energy absorption capacity, ultimate load and mean crushing load of thin-walled prismatic frusta subjected to axial impact compressive force are presented. Particularly, the influence of a diameter and number of spot welds was investigated as well as preliminary parametric analysis into an optimal angle of inclination of the frustum wall was performed.

a)


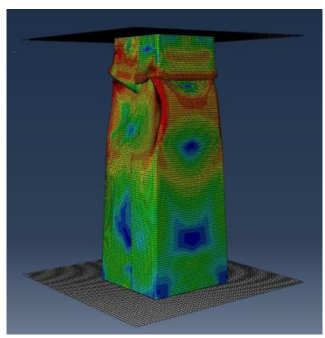
b)


Fig. 1. Numerical results a) - energy absorption versus wall inclination angle (8 spot welds of diameter d = 8 [mm]) - b) - numerical model

## 2. NUMERICAL ANALYSIS

The subject of investigation was a thin-walled prismatic frustum on square foundation shown in Fig.1, under axial dynamic compressive force. The numerical explicit dynamic analysis of crushing behaviour of the frustum was carried out using FE commercial code ABAQUS. An FE model of the frustum (Fig. 1b) was created using 4-node shell elements S4R. A column model situated between two rigid elements R3D4. Between upper and lower rigid elements and the column Tie links were applied. The rate-dependent material elasto-plastic model, taking into account the strain rate and strain hardening has been applied. Material parameters were determined in tensile and compressive tests. It was steel Dual Phase DP 800, with main parameters: E= 210 GPa, Re= 590 MPa. There, calibration of the material model was carried out using ABAQUS program [3]. The spot welds were modelled using the rigid beam elements, situated between two overlapping column surfaces.

## 3. RESULTS OF ANALYSIS AND FINAL REMARKS

Results of preliminary analysis showed that the diameter and number of spot welds had a minor influence on the crushing behaviour of examined frusta.

The further analysis was focused on the parametric study into an optimal angle of frustum wall inclination with respect to the energy absorbed. The results show the optimum of magnitude about 5º (Fig. 1) for the examined wall thickness. The ultimate load decreases linearly with the increase of the angle of inclination. The research presented in the paper is a part of a wider project. Further investigation will be continued into an experimental validation of theoretical results.

## REFERENCES


[1] Alghamdi, Collapsible impact energy absorbers: an overview, Thin-Walled Structures, 39, 2001, pp. 189-213.

[2] Rusiński E. et al., Tests of thin-walled beams joined by spot welding. J. of Materials Processing Technology, 157-158, 2004, pp 405-409.

[3] Abaqus 6.13 Documentation, Abaqus Online Documentation, Simulia, Dassault Systems, 2013.


Finite Element calculations carried out in Abaqus by CYFRONET AGH. MNiSW/IBM_BC_HS21/PŁódzka/013/2013