Vasile COJOCARU, Gabriel CACAREAZA

ANALYSIS OF STRESS STATE AND INERTIAL PROPERTIES OF A PENDULUM USED FOR CHARPY IMPACT TEST ON PLASTICS

Abstract: *The laboratory equipment used for testing the impact behavior of plastics uses low impact energies. In the last years were developed constructive solutions of monobloc pendulums, in order to ensure design and manufacturing simplicity. The paper presents the possibilities of optimizing a 15 J pendulum using 3D CAD design and finite element analysis. It highlights the possibilities for quick determination of the pendulum inertial properties ensuring the correlation between the position of mass center and the position of impact center of pendulum. A finite element analysis of maximum stress and displacement was performed correlated with the study of inertial properties.*

Key words: *pendulum, Charpy impact test, plastics, inertial properties, stress, displacements*

INTRODUCTION

The testing of impact properties of plastic materials is made on pendulum testers with low impact energy, generally between 0.5 J and 50 J [7]. The testers that ensure this range of values use pendulums with small dimensions. The potential energy provided by the pendulum is given by:

$$W = G \cdot h \tag{1}$$

where:

G [N] – the active weight of pendulum concentrated in the center of impact;

h [m] – the height measured from the horizontally axis of the tested sample to the center of impact of raised pendulum.

The Charpy pendulum testers are usually made of two main parts, an active weight and a support rod, assembled by threaded connections. The support rod must have a small mass in comparison with the active weight, in order to obtain the coincidence between the center of gravity of pendulum and the center of impact. Small weights corresponding to pendulums below 25 J involve the use of rods made on low density materials (carbon fiber, duralumin etc.).

In recent years, the companies from the industry of impact testers for plastics have introduced an alternative to the pendulum made of multi-components. This alternative solution is the design of pendulum manufactured from a single piece (monobloc pendulums) [8]. This alternative offers design and technological simplicity (this type of pendulums can be manufactured by laser or plasma cutting).

In the Strength of Materials Laboratory from the "Eftimie Murgu" University of Resita was developed, within a diploma project [1], a Charpy pendulum tester for plastic materials (figure 1), with impact energy up to 5 J, 7.5 J and 15 J.

This paper presents the design and optimization of 15J pendulum using the 3D model developed in SolidWorks. Solid Works combine in the unique interface 3D design and FEM analysis [5], [6].

Fig. 1 Pendulum tester

2. PENDULUM TESTER STRUCTURE

The 3D model of the designed pendulum tester is shown in Figure 2. On the base (1) is fixed the column (2). This column is the support for the pendulum axis (3). The pendulum (5) is fixed to the bush (4) by screwed joints. The tested sample is placed on the support (6). The impact energy is read on the dial (7) via the pointer (8) driven by the pendulum. The test area is closed by a protective screen (9).

The distance on the vertical axis between the sample center and the impact center of the raised pendulum is h=0.6 m. The radius of circle defined by the trajectory of impact center is calculated with the equation (2), where α=30º is the angle made by axis of raised pendulum with the vertical axis.

$$R = \frac{h}{1 + \cos\alpha} \tag{2}$$

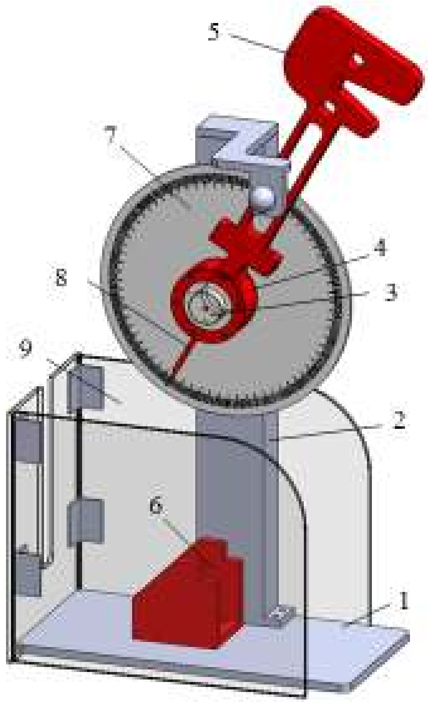

Fig. 2 CAD model of pendulum tester

The design of 15 J pendulum was carried out starting with the values of characteristic radius R = 321.54 mm and height h = 600 mm. This pendulum must have a weight G=25 N concentrated in the center of impact. The pendulum was designed in order to be processed on a laser cutting machine, without any further processing. In order to ensure this manufacturing condition, the thickness was established constant for the entire surface of the pendulum. The impact edge was designed removable and fixed on the pendulum frame by bolts.

The design and analysis of the pendulum involved two main steps:

- The dimensioning of the pendulum so that for the imposed values of R and h, to ensure the weight G = 25 N and the coincidence between the center of impact and the center of mass;
- The analysis of the maximum stresses and strains of the pendulum for maximum impact loads assumption.

3. ANALYSIS OF INERTIAL PROPERTIES

The geometry of pendulum has been designed with a central cutting on the rod, in order to improve the inertial properties comparative to a single rod pendulum. The first shape of pendulum developed in [1] ensures the active mass of 2.5 kg (G≈25N) but with an offset distance between the center of mass and the center of impact (Figure 3).

Figure 4 shows the details of the inertial properties, for the pendulum with original dimensions. Coordinates are calculated relative to a local reference system, shown in Figure 3, a system which has the X-axis along the pendulum rod, Y-axis in the rotation plane of the pendulum and the axis Z perpendicular to that plane. It can be noted that the offset distance between the center of mass and the center of impact is 5.31 mm on the Y axis and 28.8 mm on X axis.

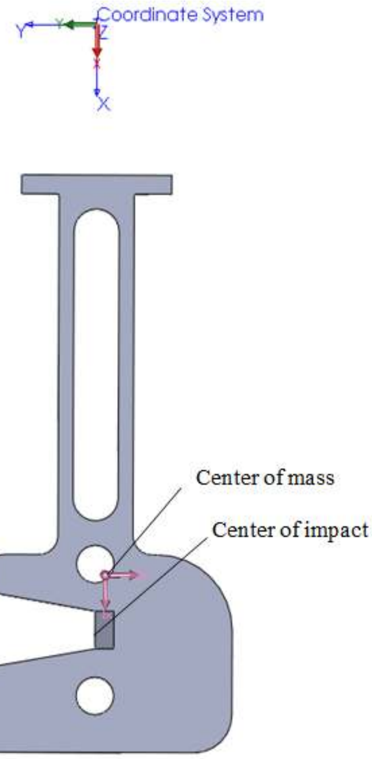

Fig. 3 Center of mass on first 15 J pendulum

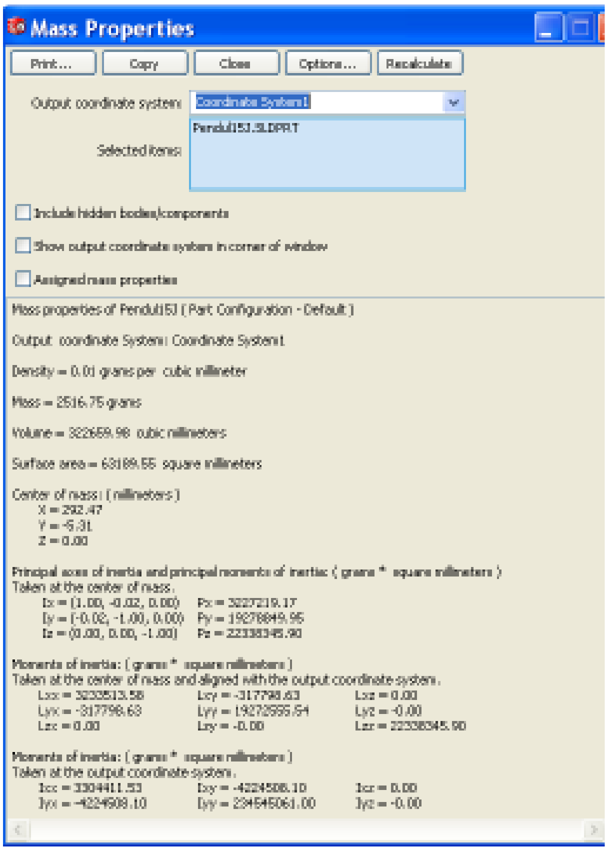

Fig. 4 Inertial properties on first 15 J pendulum

The movement of the mass center has been achieved by modifying the geometry of the pendulum body and by reducing the width in the rod area. These changes led to a decrease of the total mass of pendulum. In order to obtain the required mass were added two interchangeable weights (figure 5).

Figures 6 show the new values of inertial characteristics for the modified form of the pendulum. The pendulum mass is 2.501 kg. The offset on Y axis between the mass center and the impact center has been removed and the offset on X axis was reduced to 7.7 mm, ensuring the placement of the center of mass to the edge of impact.

The position of mass center was computed also for the entire assembly of pendulum tester aiming to verify system stability.

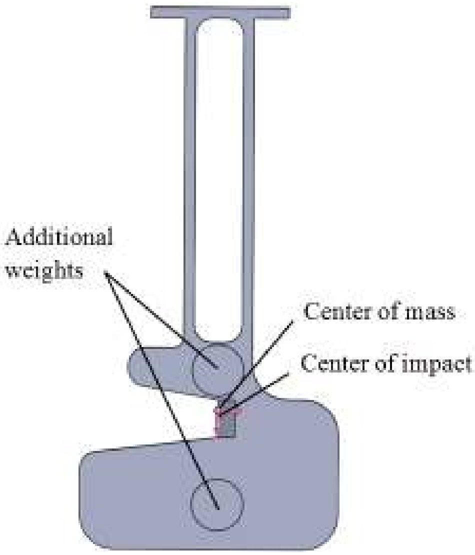

Fig. 5 Center of mass on modified 15 J pendulum

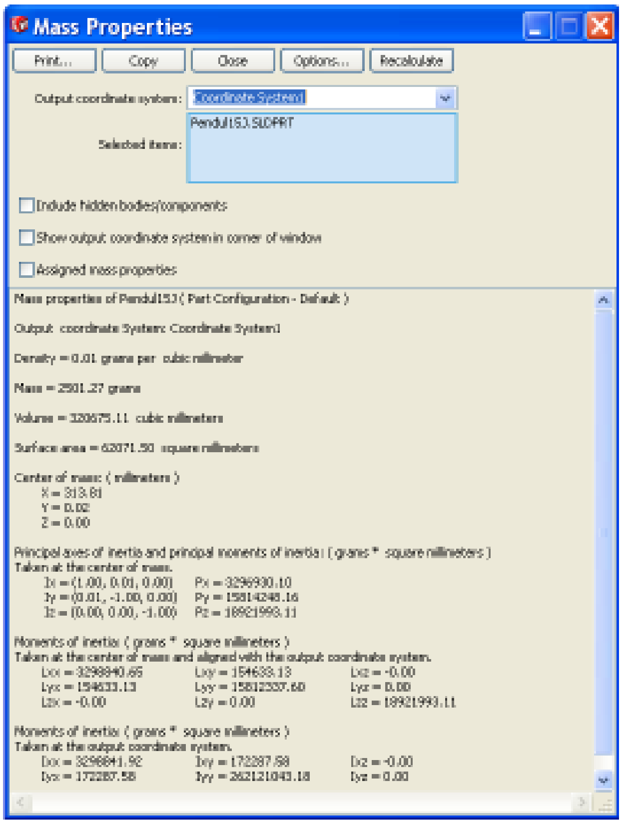

Fig. 6 Inertial properties on modified 15 J pendulum

4. FEM ANALYSIS

The change of pendulum dimensions and shape was performed concurrently with the analysis of the stresses and strains. The stress and strains were calculated for a limit loading hypothesis corresponding to the assumption that the pendulum hit an oversized steel specimen (a case of accidental loading). For the FEM analysis was defined a static study using the impact force corresponding to 15J pendulum.

Figure 7 (a) shows the loads and fixtures used in FEM simulations. The constraint (fixed geometry) was applied on the contact surface with the bush. The impact force was applied to the edge of impact, along Y axis and the own load was applied in the center of mass along the rod (appropriate to the moment of impact).

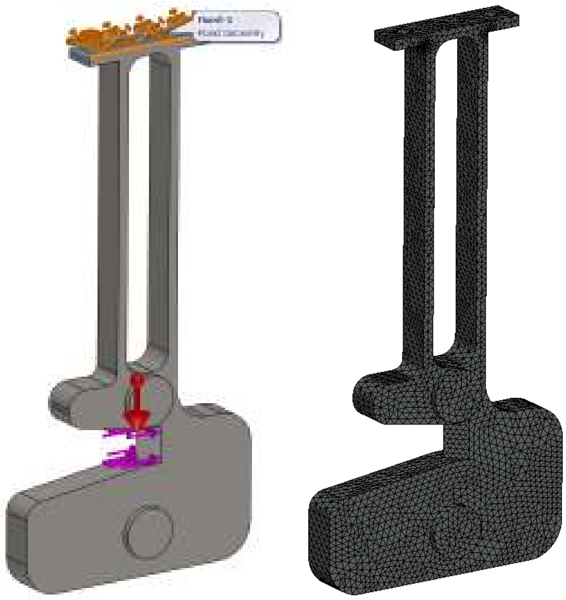

Fig. 7 FEM analysis of pendulum:
a) loads and fixtures, (b) mesh;

The convergence of results was analyzed by running twelve FEM simulations, where was varied the size and the number of finite elements. It is noted (Figure 8) that the strains and stresses varies in a small range (179.9 MPa to 196.6 MPa for equivalent von Misses stress), although the number of finite elements is increased approximately 50-fold (from 2175 to 114206). The results presented below and analyzed in this paper are related to maximum mesh (Figure 7 b).

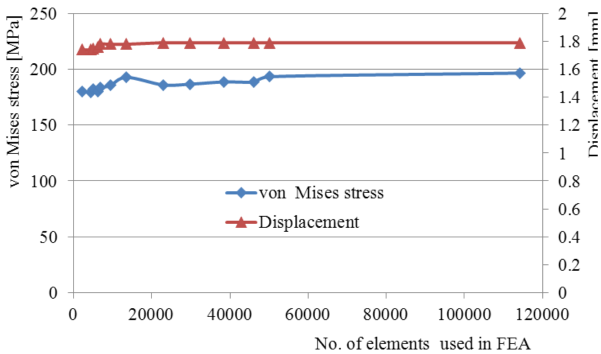

Fig. 8 Stress and displacement variation related to number of element used in FEM study

The maximum values for stress (Figure 9) occur on the rod of pendulum near the corner radius. The increase of corner radius would decrease the maximum stress [2], [4] but this increase is not possible because of the positions of screwed connection of the pendulum with the bush. Maximum von Mises equivalent stress is 196.6 MPa. The safety factor related to the yield strength is c = 1.39 (the material used for the manufacturing of pendulum was the unalloyed steel C45). This value is considered acceptable on the mechanical parts to which regular monitoring can be done [3].

The maximum values for resultant displacement (Figure 10) are relatively high, U_{res} = 1.79 mm, but these values correspond to a case of accidental application and does not affect the measurements performed on plastics samples.

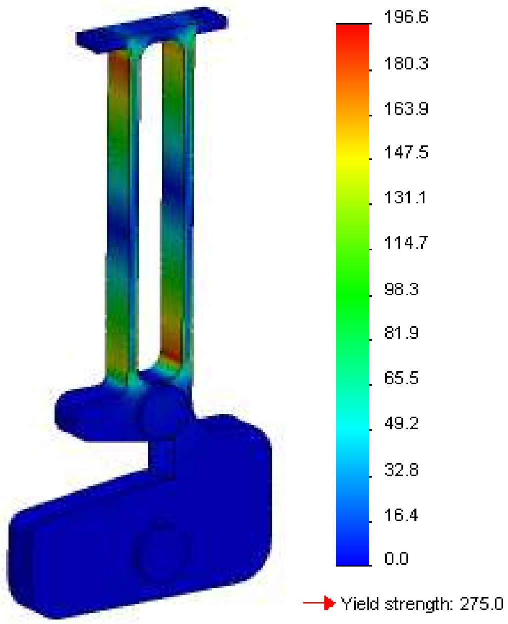

Fig. 9 Equivalent von Mises stress diagram

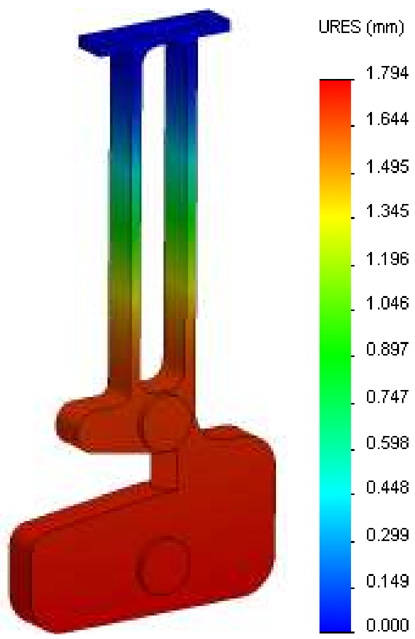

Fig. 10 Resultant displacement (U_{res}) diagram

5. CONCLUSIONS

The paper presents the analysis of inertial characteristics and FEM analysis of stress and strain for a pendulum used to test impact behavior of plastics by Charpy method.

The development of 3D models of mechanical systems in a CAD software that allows both geometric modeling and finite element analysis lead to multi-criteria optimization. The goal of the optimization process of the pendulum was to achieve the inertial properties imposed by standards correlated to a stress and strain state in allowable limits.

The analysis of inertial characteristics performed in CAD software allows the calculation of mass, center of mass position and moments of inertia. This analysis is important for achieving imposed geometric characteristics and to assess the stability of an assembly.

The influence of meshing on stress and deformation values was analyzed through a study of convergence. This study showed that stress variation is under ten percent for a 50 times increase of the number of finite elements.

REFERENCES

[1] Cacareaza, G. (2016). *Design of a Charpy Impact Pendulum for plastics*, Diploma project, project coordinator: Cojocaru V., “Eftimie Murgu” University of Resita

[2] Cojocaru, V. (2013). *Influence of the rib radius on the maximum stresses and deformations of a corner bracket*”, Analele Universitatii Eftimie Murgu din Resita, Fascicula I, vol. XX, no.1, pp. 71-76

[3] Hanel, B. (2003). *FKM Guideline – Analytical strength assessment of components in mechanical engineering*, VDMA Verlag GmbH, Frankfurt

[4] Muhs, D., Wittel, H., Jannasch, D. (2008). *Roloff/Matek Machine Elements (Organe de masini)*, Vol. I+II, Editura Matrix Rom, ISBN 978-973-755-412-3, Bucuresti

[5] Nedelcu, D. (2011). *Digital Prototyping and Numerical Simulation with SolidWorks*, Eurostampa, ISBN 978-606-569-276-3, Timisoara

[6] Shih, R. (2012). *Introduction to finite element analysis using SolidWorks Simulatio*n *2012*, Ingram, ISBN 978-1-58503-704-9

[7] *** Zwick/Roell, *HIT Pendulum Impact Testers from 5 to 50 J*, FP 5732.0116, www.zwick.com

[8] *** Instron, *CEAST 9000 Series, Pendulum Impact Systems*, www.instron.com; Accessed: 2016-10-12

Authors:

Lecturer Dr. Eng. Vasile COJOCARU, “Eftimie Murgu” University of Resita, Department of Mechanical Engineering and Material Science, E-mail: v.cojocaru@uem.ro.

Eng. Gabriel CACAREAZA, Master student, “Eftimie Murgu” University of Resita, Department of Mechanical Engineering and Material Science.