https://dx.doi.org/10.22161/ijaers.72.10 **ISSN: 2349-6495(P) | 2456-1908(O**

Stamping set modeling to aluminum coins manufacturing through 3D software CAD/CAE

Santos, Diego Barbosa Pratis[1], Maia, Luis Henrique A.[2], Martins, Paulo Sérgio[3], Freitas, Júlio César Lemos de[4], Drumond, Ramon Martins[5], Cheikh, Elhadji[6], Pereira, Otávio dos Santos[7]

[1]Me Student, Mechanical Engineering Department, Pontifical Catholic University of Minas Gerais, Belo Horizonte, Brazil
[2]Professor, Mechanical Engineering Department, Pontifical Catholic University of Minas Gerais, Belo Horizonte, Brazil
[3,4]Professor, Mechanical Engineering Department, UNA, Belo Horizonte, Brazil
[5,6]Me Student, Mechanical Engineering Department, Federal Center for Technological Education of Minas Gerais, Belo Horizonte, Brazil
[7] Me Student, Mechanical Engineering Department, Federal University of Minas Gerais, Belo Horizonte, Brazil

***Abstract* —The time-to-response, level of accuracy and its consequent quality required to manufactured goods have demanded from engineers efficient methods to get a reasonable design. Following this trend finite elements methods have been used for years in order to predict design constrains and front loading potential issues prior to the manufacturing. Among several manufacturing processes this paper will highlight stamping process in a high level way and how finite elements simulations can support analysis of displacement and deformation of parts when an external load is submitted. It was used SolidWorks software to generate the meshes and perform the simulation.**

***Keywords*— Stamping, modeling, CAD, simulation, finite elements, SolidWorks**

I. INTRODUCTION

Computational tools application to simulate structure behavior have been increasing in engineering field. The reason of that is the design accuracy increase since the behavior data acquisition is independent of prototype construction that generates cost reduction. Furthermore a better material choice and components dimensioning assertiveness can be achieved due to availability of resources and multiple analysis. This work presents the development and outcomes generated from a software CAD simulation of an experimental pneumatic stamping to manufacturing of coins.

II. FINITE ELEMENT SIMULATION

Widely used in structural analysis, independent of geometry complexity, the finite element method is a mathematical analysis based on the part division into a known amount of discrete elements connected by nodes called mesh [1]. The structure analysis can be enhanced by the usage of 3D modeling even the calculation is too complex to be executed analytically. [2] have declared that finite element method corresponds to differential equations calculation which through numerical approximations, it becomes possible to perform complex analyzes of stresses, heat transfer, fluid flow, electromagnetism, among others.

The model of part division is represented in the following picture where it is possible to see the a generated mesh.

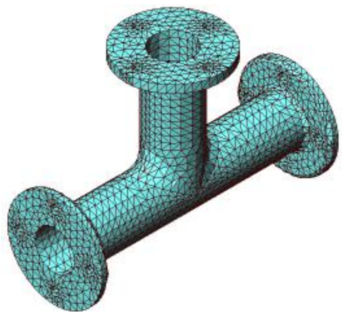


Fig. 1: Mesh representation in a generic part

Meshes can take different formats such as triangular, tetrahedral, quadrilateral and hexahedral. Finite element software usually has a library of elements for analysis and choice of mesh type [3]. When a deep analysis is required it is possible to increase density to get mesh refinement by increasing the number of divisions which generates element size reduction in regions close to large stress gradients. The criteria to mesh refinement is based on convergence tests where the stress gradients is compared among different zones. When a high level of stress is verified a refinement still need to be made [1].

https://dx.doi.org/10.22161/ijaers.72.10 ISSN: 2349-6495(P) | 2456-1908(O

III. STAMPING PROCESS

According to [4], stamping process consists of a set of operations in which it is possible to obtain a part in the desired geometry, without the production of chips typical in removal material processes. The stamping cutting tools are known as cutting die and are basically constituted by a die and a punch [5]. The following Figure 2 shows a generic stamping die schematics.

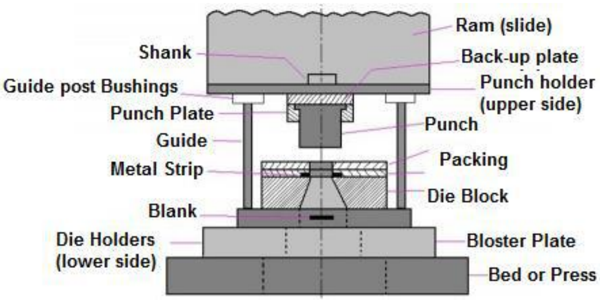


Fig. 2: Generic stamping die schematics

According to Chiaverini [6] to determine the effort required to cut the material, Equation (1) below is used:

$$Q = p.E.\theta_C \qquad (1)$$

Q is the force required for cutting (N), p is the cutting perimeter (mm), E is the thickness of the plate (mm) and Theta is the shear stress (N/mm).

IV. METHOD

To design any structure it is necessary to use the principle of the balance of forces, represented in the Equation 2:

$$\Sigma FX=0 \ \Sigma FY=0 \ \Sigma FZ=0 \ \Sigma MX=0 \ \Sigma MY=0 \ \Sigma MZ=0 \qquad (2)$$

They refer to the sum of forces and moments in the X, Y, and Z axis to determine the forces acting on the material. The dimensions of the elements, deflection and stability depend on the internal loads and also on the type of material they are made of. Von Mises was the criteria failure adopted for modeling. It uses the maximum distortion energy theory applied to ductile materials, in this work AISI 1020 steel.

The 3D CAD software SolidWorks was used in this work. The behavior evaluation of the components from the submitted efforts was made through the SimulationXpress feature available in this software. A finer mesh usually results in a more accurate solution. However, as a mesh becomes thinner, the calculation time also increases. The Figure 3 shows the meshes generated by the software of the main components of the experiment.

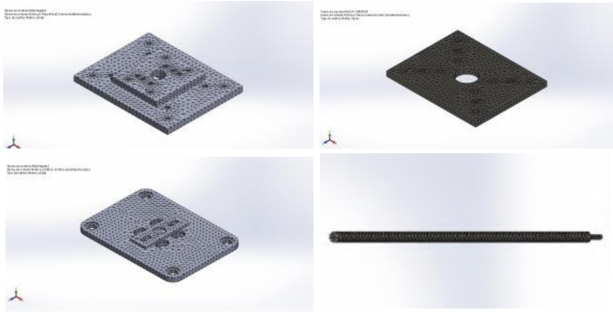


Fig. 3: Meshes on main components of the experiment

In the simulation were used for the structure, in addition to AISI1020, steel threaded rods M10, plain, pressure washers and nuts, according to DIN912. The structure was composed of a base plate, an upper plate and a movable plate or hammer, as well as threaded bars and guide bars. The shear force required to obtain the final product calculated from the Chiaverinni equation was 1100N approximately. Based on that the base of stamping should be designed to withstand both the impact load and the static loading of the press stamping tool. The hammer was designed to give balance to the movements of the upper set of stamping, at the time of cutting, to avoid misalignment and assure process high quality. The upper part is designed to support a cylinder which characteristics are shown in the Table 1.

Table 1: Cylinder main characteristics

Diameter (mm)	63
Young's modulus (E)	200GPa
Tensile strength (S_r)	420.51 MPa
Elastic limit (S_e)	351.57 MPa
Specific mass (ρ)	7900kg/m^3
Shear modulus (G)	77 GPa
Diameter (mm)	25
Length (mm)	25
Weight (kg)	3.85

Table 2 shows the dimensions of the designed press.

Table 2: Overall dimensions of the designed press

Width (mm)	250
Length (mm)	200
Height (mm)	210

Table 3 shows the characteristics of the base and top plate.

https://dx.doi.org/10.22161/ijaers.72.10 ISSN: 2349-6495(P) | 2456-1908(O

Table 3: Characteristics of the base and top plate

Width (mm)	250
Length (mm)	200
Thickness (mm)	12.5
M10 holes to threaded bars fixture (qty)	4
Ø 8mm holes to linear guides fixture (qty)	4
Base - Oblong hole to tool fixture and material drain (qty)	1
Top - Fixture cylinder hole (qty)	1

Table 4 shows the characteristics of the movable plate.

Table 4: Characteristics of the movable plate

Width (mm)	180
Length (mm)	130
Thickness (mm)	8
Ø 8mm holes to linear guides fixture (qty)	4
Holes to stamping tool fixture (qty)	4

The Figure 4 represents the press assembled based on the listed characteristics.

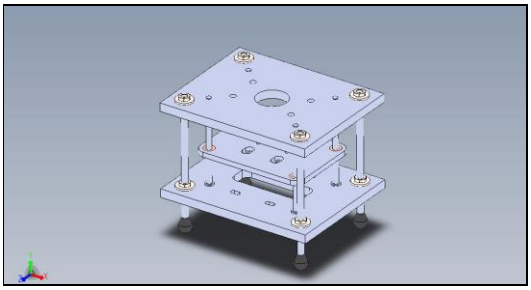


Fig. 4: Assembled press

In order to carry out the simulation, some simplifications were proposed in the model. The system stems and the upper plate were not considered since the load could be applied from the movable plate. Static analysis and an external load of 1100N were used.

The results obtained were the displacement and deformation of the parts subjected to the application of the load, besides the safety factor distributed throughout the different regions of each one of the pieces.

Displacement and deformation of parts under the load applied were the main outcomes analyzed by the comparison to the yield stress of AISI 1020 steel.

V. RESULTS AND DISCUSSION

The analysis of the base plate shown in Figure 5 has shown a concentration of stresses close to the through holes of the threaded rod and to the stamping tool die fixing holes.

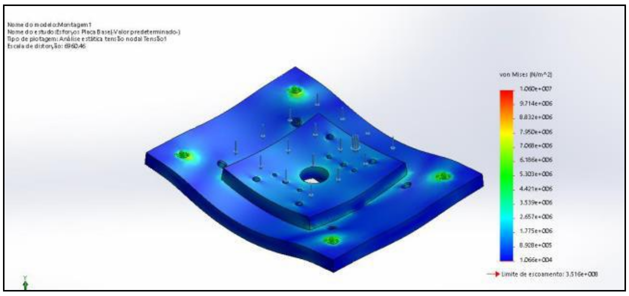


Fig. 5: Bed plate – concentration of stress

The following Figure 6 shows the behavior of the part from displacement standpoint.

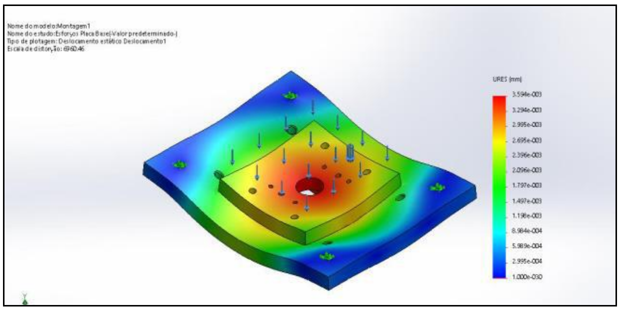


Fig. 6: Bed plate – displacement analysis

The software reproduces the images in a larger proportion in order to allow a better visualization of the displacement. Same happens to the analysis of concentration efforts.

The following analysis was performed to verify the safety coefficient of the part (Figure 7). The red region is the one that suffers the greatest efforts and tensions.

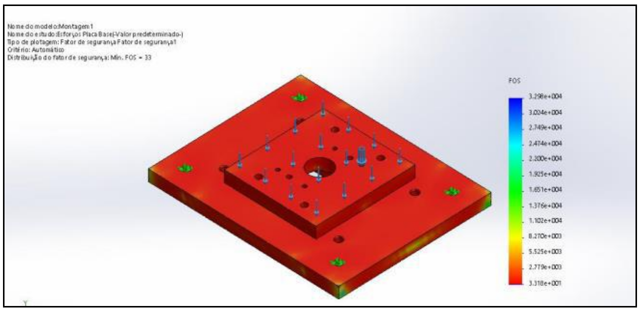


Fig. 7: Bed plate – safety coefficient analysis

It was observed in the upper plate that the highest level of tensions are located near to the passage holes of the threaded base, as shown in the Figure 8.

https://dx.doi.org/10.22161/ijaers.72.10 ISSN: 2349-6495(P) | 2456-1908(O

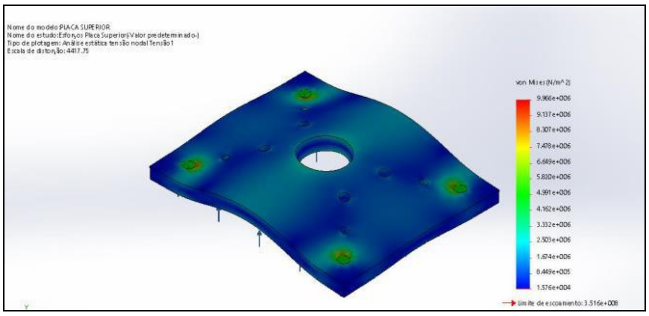


Fig. 8: Upper plate – tension distribution

Figure 9 evaluation shows the variation of the displacement when the upper is under load.

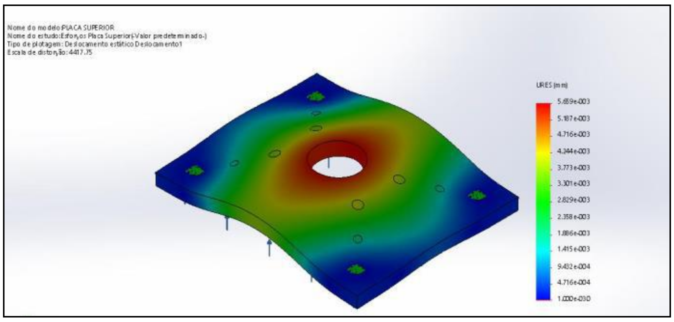


Fig. 9: Upper plate – displacement variation

Figure 10 shows the regions where the maximum and minimum safety factors of the upper plate can be found.

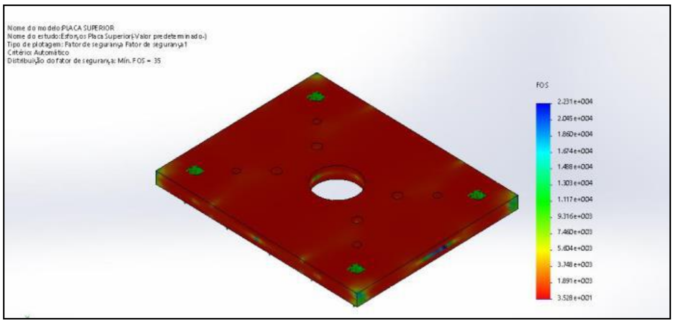


Fig. 10: Upper plate - safety coefficient analysis

Related to the analysis to the movable plate Figure 11 shows the regions that suffer the greatest tensions are located in the center of the part, where are the holes for the fixation of the tool of stamping, fixation of the tip of the rod of the cylinder besides the region of the fixing holes of the guide rod.

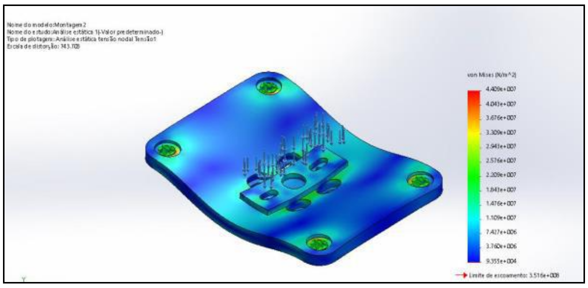


Fig. 11: Movable plate – tension distribution

Figure 12 shows a greater displacement variation in the central region of the part.

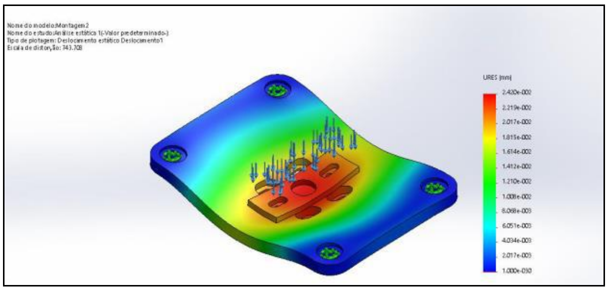


Fig. 12: Movable plate – displacement variation

The safety factor of the mobile plate was analyzed according to Figure 13.

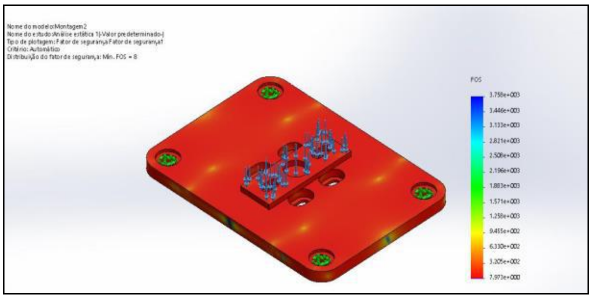


Fig. 13: Movable plate – safety coefficient analysis

The threaded rod is manufactured in AISI 1045 steel, where its mechanical properties are shown in Table 5.

Table 4: Mechanical properties of AISI 1045 steel

Young's modulus	205GPa
Tensile strength	625MPa
Yield limit	530Mpa
Specific mass	7850Kg/m³
Shear modulus	80GPa

The threaded rods were 10 mm in diameter with load applied of 500N (single bar analysis). Figure 14 shows the strain variation of the rod.

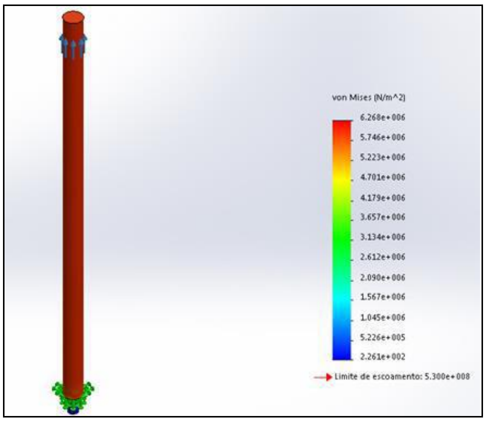
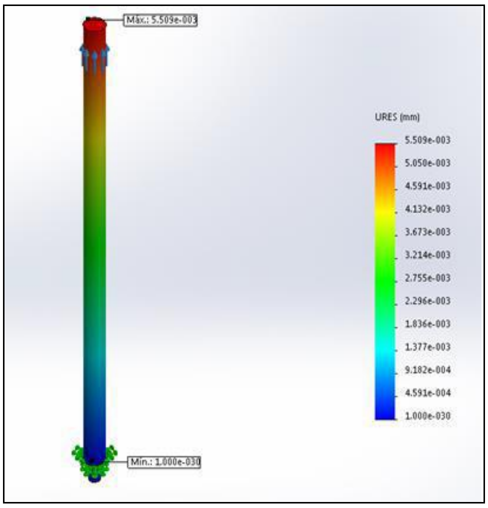


Fig. 13: Threaded rod – strain variation

Figure 13 shows how the rod varies in its displacement.



VI. CONCLUSION

Based on the outcomes from the simulations performed was possible to confirm that the material (AISI 1020 steel) is capable to support the external load applied from material resistance standpoint. The displacements obtained do not represent a significance considering that the highest value founded in the movable plate was around 0,02420mm. CAD design is the base to CAE simulation since the combination of plates from the proposed press interact between it selves distributing the load as soon as it is applied. Finite Elements Methods applied to stamping process evaluation has demonstrated an efficient method considering time-to-response, accuracy level and robustness.

REFERENCES

[1] Norton, R. L. (2003). Projeto de máquinas uma abordagem integrada. Porto Alegre, BR, Eds. Bookman.
[2] Fish, J.; Belytschko, T. (2009). Um primeiro curso em elementos finitos. Rio de Janeiro, BR, Eds. Ltc.
[3] Budynas, R. G.; Nisbett, J. K. (2011). Elementos de máquinas de Shigley: Projeto de engenharia mecânica. Porto Alegre, BR, 8th ed., Eds. Bookman.
[4] Junior, I.; Caversan (2012). Tecnologia de estampagem. Centro Paulo Souza. São Paulo, BR, pp. 82.
[5] Rocha, O. F. L. D. (2012). Conformação mecânica. [s.l].
[6] Chiaverini, V. (1997). Tecnologia mecânica: processos de fabricação e tratamento. 2nd ed., São Paulo, BR, eds. Markron, pp. 165.