Modern Environmental Science and Engineering (ISSN 2333-2581)
June 2018, Volume 4, No. 6, pp. 508-515
Doi: 10.15341/mese(2333-2581)/06.04.2018/003
Academic Star Publishing Company, 2018
www.academicstar.us

Development Structural of A Wire Drawing and Extrusion Bench

Gilmar Cordeiro da Silva, Ana Caroline de Souza Couto, Daniel de Castro Maciel, Alysson Lucas Vieira, Ernane Vinicius Silva, and Norberto Martins

Pontifical University Catholic of Minas Gerais (PUC-MG), Brazil

Abstract: The present work aim consist to design and construction of a drawing wire and extrusion bench able to perform the metal forming process by drawing and extrusion. The design of the structure as well as detailing of mechanical components were designed with the use of SOLIDWORS software and the simulation of structural strain (FEA - Finite Element Analysis) with the aid of ABAQUS/Standard software.

Key words: wire drawing, extrusion, SOLIDWORKS, ABAQUS/Standard, finite element analysis

1. Introduction

Extrusion is the process by which a block of metal is reduced in cross-section by forcing it to flow through a die orifice under high pressure. It is accepted practice to classify extrusion operation according to the direction relationships between the material flow and the punch movement. The basic extrusion operations are backward-extrusion, forward-extrusion, and radial-extrusion. Drawing process is similar to extrusion, in that a length of metal is made to flow through a die opening and reduction is done over its cross section. The difference between drawing and extrusion is the application of force to the work piece. In extrusion the material is pushed through the die opening, in drawing the material is pulled through the die opening.

The finite element analysis is widely accepted and often used as an alternative to the experimental test method set out in many standards. The technique is based on the premise that an approximate solution to any complex engineering problem can be reached by subdividing the structure/component into smaller more manageable (finite) elements. The Finite Element Model (FEM) is analyzed with an inherently greater precision than would otherwise be possible using conventional analyses, since the actual shape, load and constraints, as well as material property combinations can be specified with much greater accuracy.

In the development of this work were used the computational resources of two softwares, SOLIDWORKS and ABAQUS, for the design and simulation of a wiredrawing and extrusion bench. The main purpose of using the software was to optimize and reduce the cost with materials as well as structural analyse of behaviour of bench when requested work efforts. For the simulation an "I" structural profile was used to verify your numeric response regarding the behaviour of the structure of the bench in different geometric combination of trusses.

2. Finite Element Method (FEM)

The advancement in software simulation combined with the finite element method tool (MEF) has significantly improved the manufacturing time of

Corresponding author: Gilmar Cordeiro da Silva, Doctor, Professor, research areas/interest manufacturing process and materials. E-mail: gilmarcord@gmail.com.

products and tools [1]. The analysis by (MEF) is a numerical simulation technique that predicts the efforts and possible component failures. The advantage of its use is in the reduction of costs in try-out, in the time of design and launch of the final product in the consumer market, and in addition, it allows the improvement of the projects through the simulation of concepts and scenarios before the decision making end. The simulation process is basically performed in 3 stages as illustrated in Fig. 1.

The pre-processing stage comprises the physical and mathematical modeling of the problem in question. In physical modeling, the geometries of the matrices and the initial part must be defined, using a graphical interface of the software or CAD systems. Then, depending on the geometry of the model and the loading conditions, the type of analysis is defined as Silva [2]. The processing is the stage where the computational calculation is performed with the problem divided into displacement increments. In this step, it is defined which solution to be adopted, that is, mechanical, thermal,

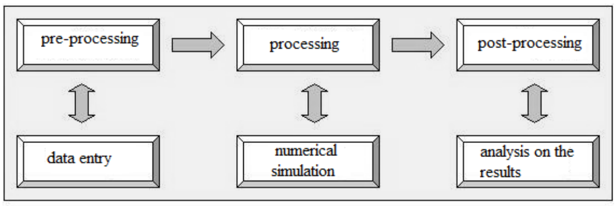


Fig. 1 Simulation Process [2].

thermo-mechanical, etc. Finally, the third stage, the post-processing, where the final data is read.

For the extrusion and drawing operations, the technical conditions determining the quality of operations include the extrusion and drawing ratio, the product profile, tool and the friction conditions at the part/material interface. The optimization of the parameters of work is of fundamental importance.

The design of the extrusion/drawing processes by trial and error can be time consuming and costly. An alternative has been the finite element method (FEM), with satisfactory results.

Soares et al. [3] investigated the influence of several parameters on the appearance of residual stresses in finished products. The authors used numerical simulations using finite element methods and compared the drawing forces obtained by the software with those obtained by the empirical equations of Sachs and Avitzur.

Jiayong Si et al. [4] also carried out a study focused on the finite element analysis of TiAl alloy extrusion.

Process variables were evaluated based on simulation results, such as extrusion force, die loading during the process and strain distributions. Fig. 2 shows the simulation results of the TiAl bar with its deformations. It was evidenced points with maximum deformation in the two materials at different extrusion rates.

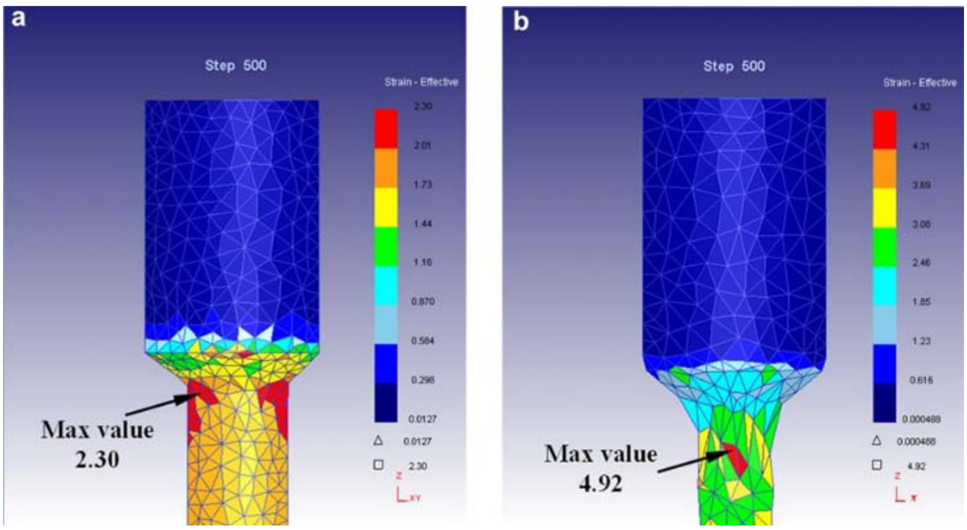


Fig. 2 Effective strain distributions for different extrusion ratio (a) *R*¼3, a¼60°; (b) *R*¼6, a¼60°.

L. G. Deshpande, N. M. Sawalhi, and R. B. Randall [5], used the finite element technique to simulate rolling failures in gearboxes. The casing of the gearbox composed of welded steel sheets was modeled using solid and shell elements. The model was supported by rubber cushions that were simulated using spring elements at the corners of the enclosure.

3. Cold Drawing/Extrusion

For the purpose of minimizing costs, maximizing productivity, being more competitive on global market and avoiding raw material losses when using no specified products, plastic deformation process has been modernized on the last years. Researches have been done about the process variable parameters and their influence on finished products. So, a big progress on cold drawing techniques have been verified on the last years, through automation and continuous thermal treatment improvements, making standardization and finished products quality better. It is a mechanical process which guarantees dimensional precision and best mechanical properties to the manufactured pieces [6]. Cold drawing is a conformation process where a bar or a tube is pulled through a tool called drawing die, made from a high wear resistance material like hard metal (tungsten carbide), diamond (natural or polycrystalline) and some ceramic materials (zircon oxide, titanium carbide-nitrate, etc.). The drawing die has a cylindrical external format, containing a center hole for passing the wire. This hole has a decreasing diameter, presenting a conical form profile.

According to Bresciani [7], the drawing die has to be constructed with the purpose of dealing with high productivity and low wear, large reductions, getting a better result for superficial finishing and having a better dimensional precision. The passage of the wire causes its transversal section reduction. This operation is commonly done on a cold process, occurs a modification on wire material's mechanical properties because of work-hardening [8]. Modifications that occur on ductility reduction and mechanical resistance increase direction. Thus, it is possible of needing annealing thermal treatment on every part of cold drawing process, decreasing cold drawing effects, and providing it more ductility, so that it can handle the process for longer. Deformation preponderant efforts are compression efforts, done by the tool orifice walls when line is passing through a traction effort applied on its axial direction, from an external origin. Since external effort is by traction, and deformation efforts are by compression, cold drawing process is classified as an indirect compression process.

According to Akikazu et al. [9] and Wang and Gong [10], deformation can be reached when combining traction and compression tensions, whose are created by traction force on drawing die's way out and its geometry. Considering its main tensions, different failing criteria can be used. A metal draining is determined by microstructure grain movement, ruled by shearing tensions. Thus, plastic deformation beginning can be determined by maximum shearing tension, based on Tresca criteria Cetlin [11].

Extrusion is one of the most used conformation processes among different processes. A wire is placed in a die and pressed by the punch, causing the metal to flow through the die opening. It can be classified in the front, rear and lateral. The lateral extrusion, sometimes referred to as radial, offers a product with the central protrusion having full or segmented protrusions. The research work for lateral extrusion is not as abundant as for forward and inverted extrusion. It is accepted to classify the extrusion operation according to the relationship between the material flow and the punch movement [2]. Fig. 3 shows a schematic drawing of the geometry of a forward, reverse and radial extrusion tool.

4. Experimental Tests

The material used for the bench construction and simulation in ABAQUS/STANDARD software was ASTM-A36 structural steel, whose mechanical properties are presented in Table 1.

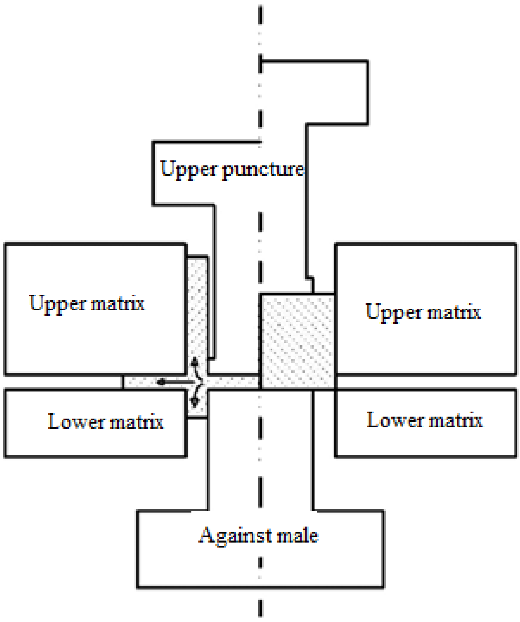


Fig. 3 Schematic drawing of the geometry of a forward, reverse and radial extrusion tool [2].

Table 1 Mechanical properties of ASTM-A36 material.

Material	ASTM- A36
Poisson	0.32
Young Module (GPa)	200
Density (Kg/m^3)	7850
Yeld Stress (Mpa)	250
Ultimate Stress (Mpa)	400

The structural simulations were done in ABAQUS/STANDARD software in version 6.14 and the bench components were simulated in SOLIDWORKS software version 2016. The operating system used was WINDOWS 10 with the 1.60 GHz INTEL® processor, with a 2.00 GB RAM. The structural "I" profile was used in the manufacturing and simulation with dimensions shown in Fig. 4, and the process load was 40 ton.

With the section dimensions and the material mechanical properties, the contour conditions were defined. Fig. 5(a) shows the bench attachment points and Fig. 5(b) shows the application load points, where the hydraulic cylinder will be positioned.

The cylinder holder was simulated in SOLIDWORKS software. The support design, the constrains and the load points were defined according to Fig. 6(a). The mesh has 22,500 tetrahedral elements with the total number of nodes 36,026. Fig. 6(b) shows the complete drawing structure which was simulated.

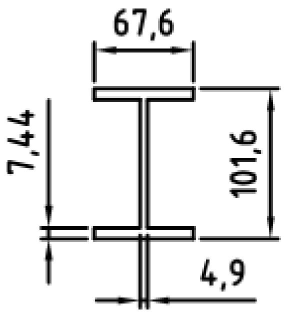
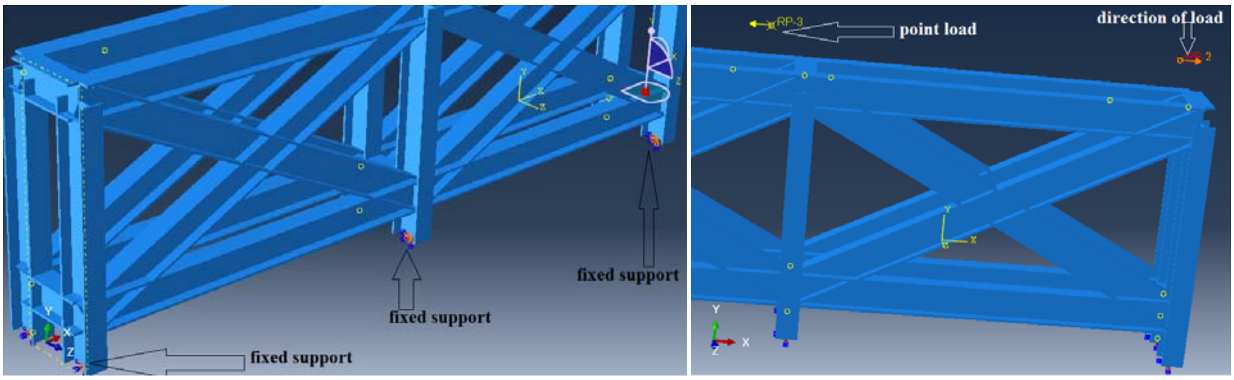


Fig. 4 Bench profile dimensions (mm).



(a) (b)

Fig. 5 (a) Bench constrains points, (b) load points.

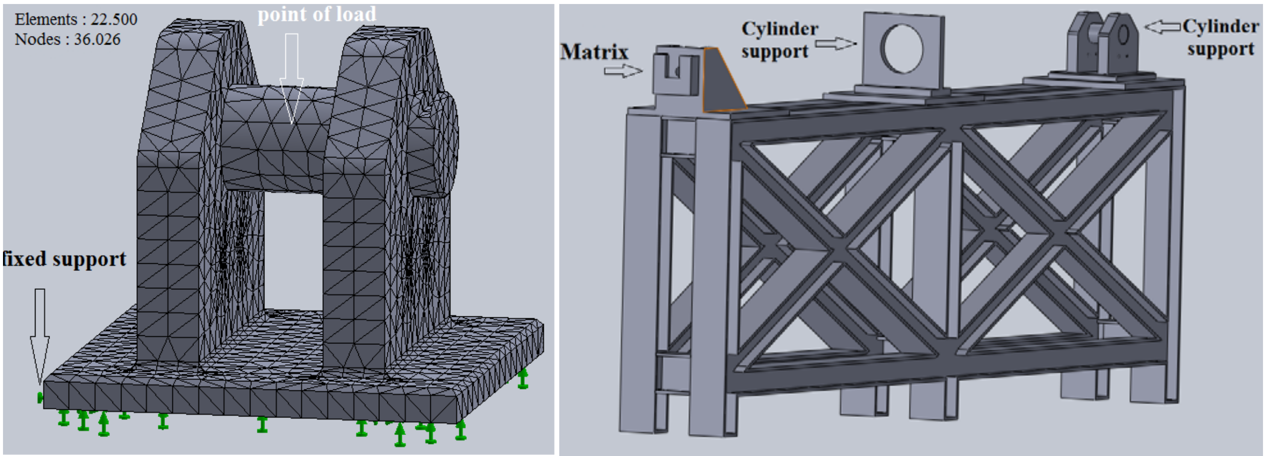
5. Results and Discussion

The initial bench design was a simple structure with a single bracing to minimize weight and withstand loading. According to Fig. 7, it is possible to observe that the VonMises stress at the most critical point exceeded the material yield stress by 80.28%.

A second attempt was made to verify a use of double bracing at the critical points for the localized stresses distribution, as shown in Fig. 8. The critical VonMises stress was below the material yield stress by 2.28%.

The simulation was performed in the SOLIDWORKS program in the Fig. 9 shows the simulation results, the critical VonMises stress value was 351.7 MPa.

The SOLIDWORKS simulation showed some important points, where the stress presented a value close to the yeld stress. In Fig. 10, it shows the points in the support where it presented greater stress.



(a) (b)

Fig. 6 (a) Cylinder holder mesh, SOLIDWORKS, (b) Bench structure, SOLIDWORKS.

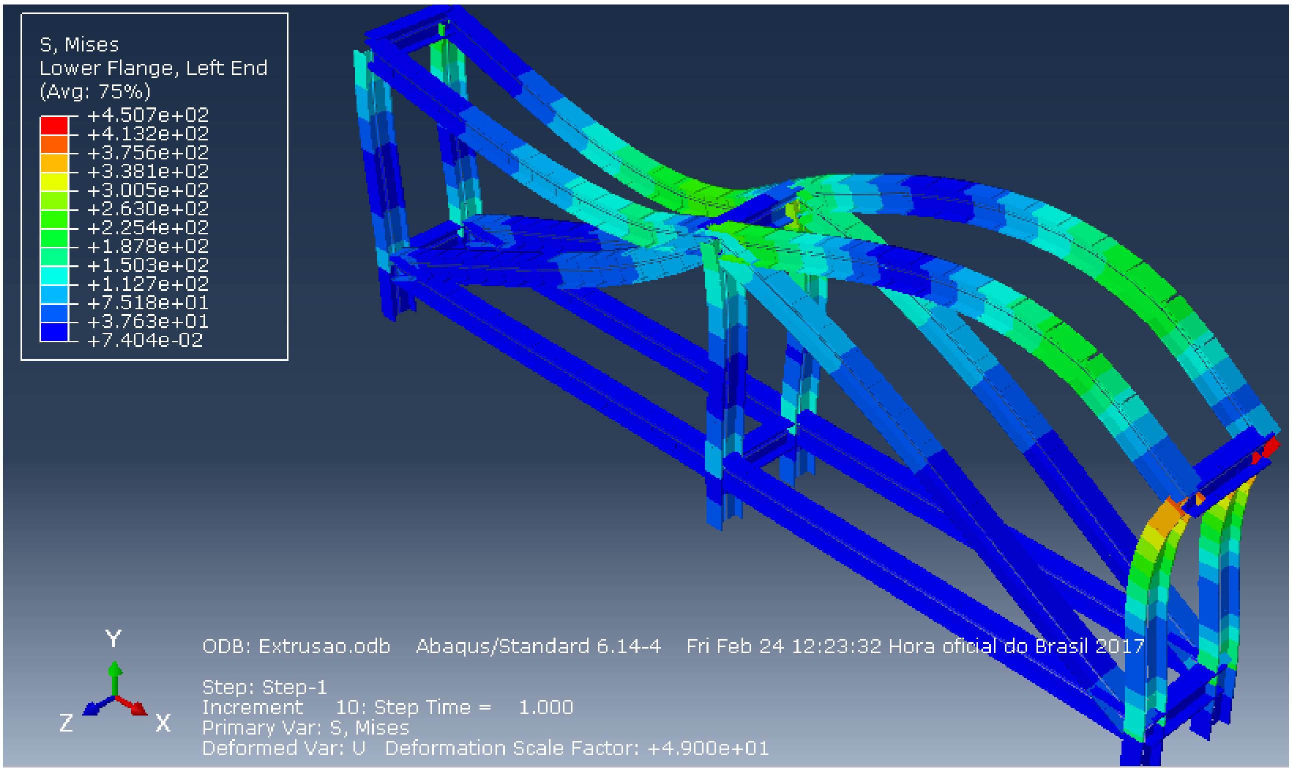


Fig. 7 Bench extrusion simulation process.

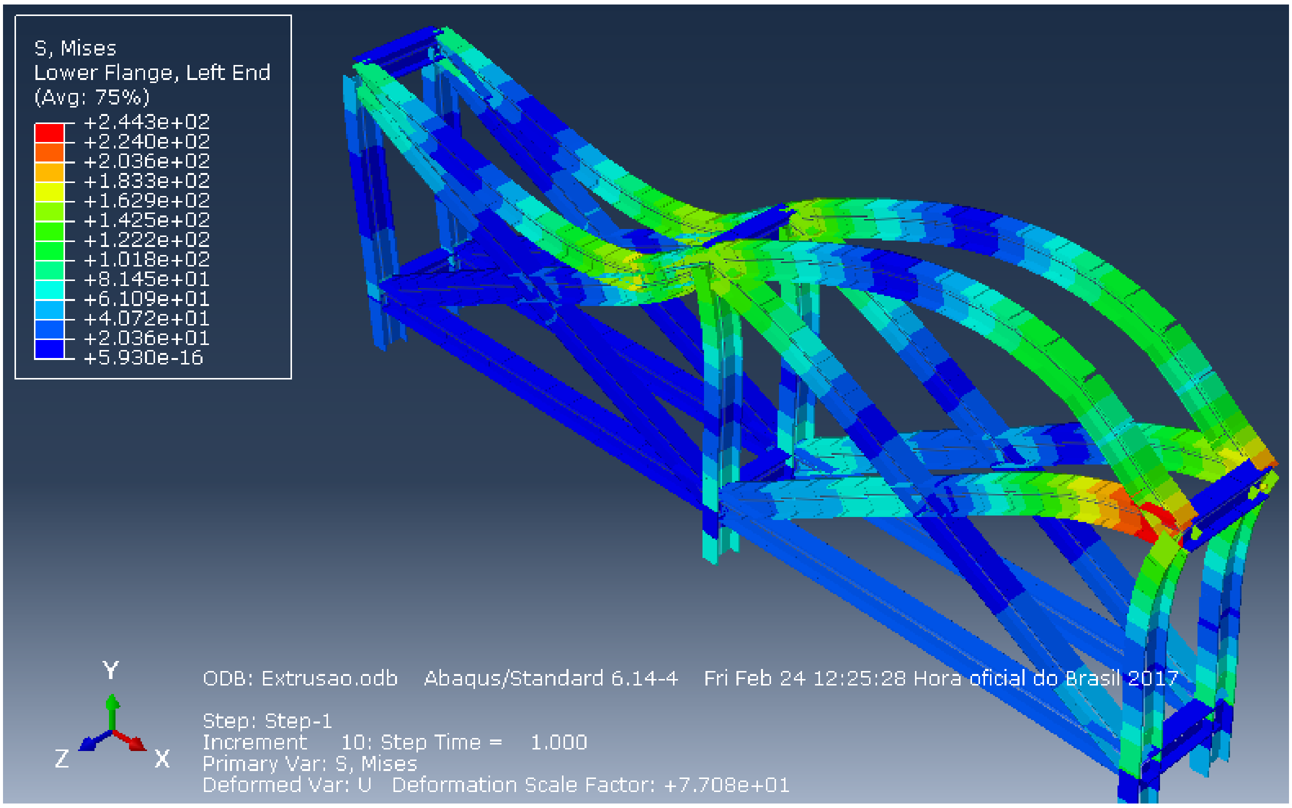


Fig. 8 Extrusion process simulation with double bracing.

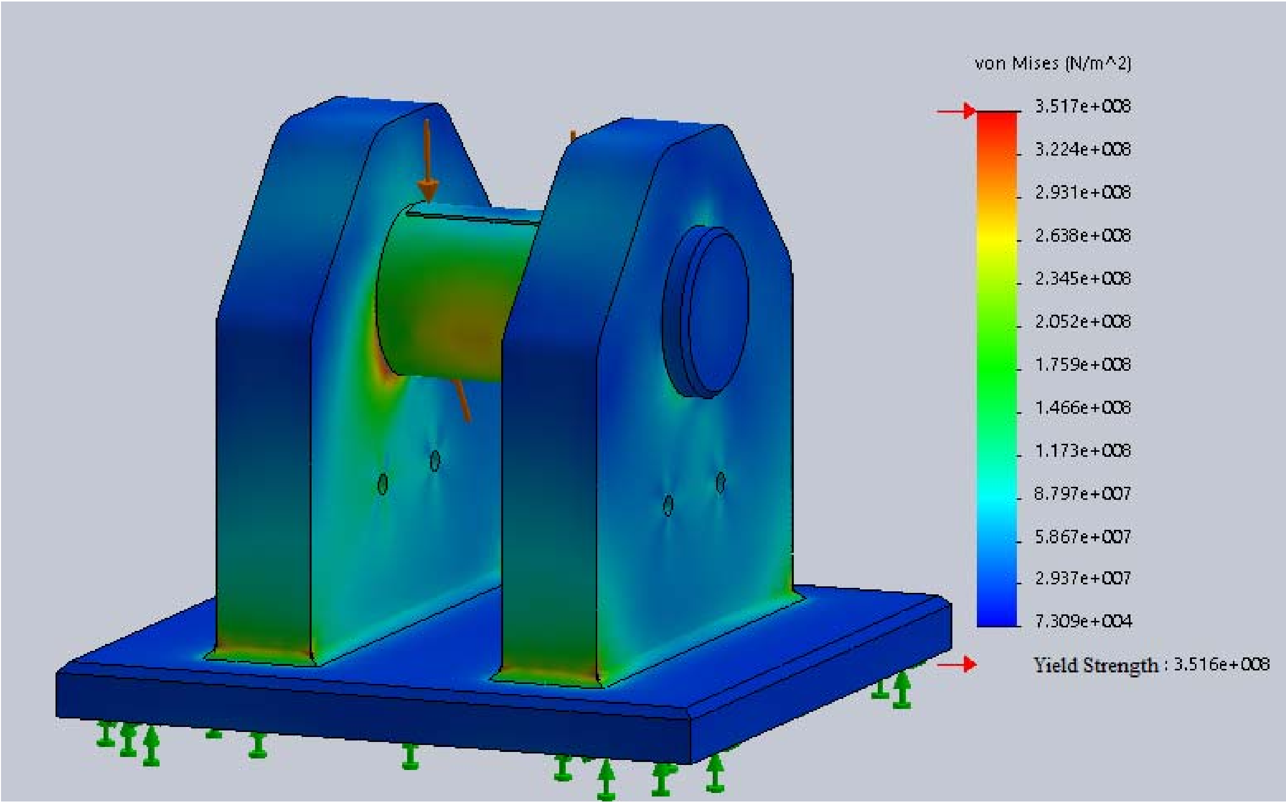


Fig. 9 SOLIDWORS stress results.

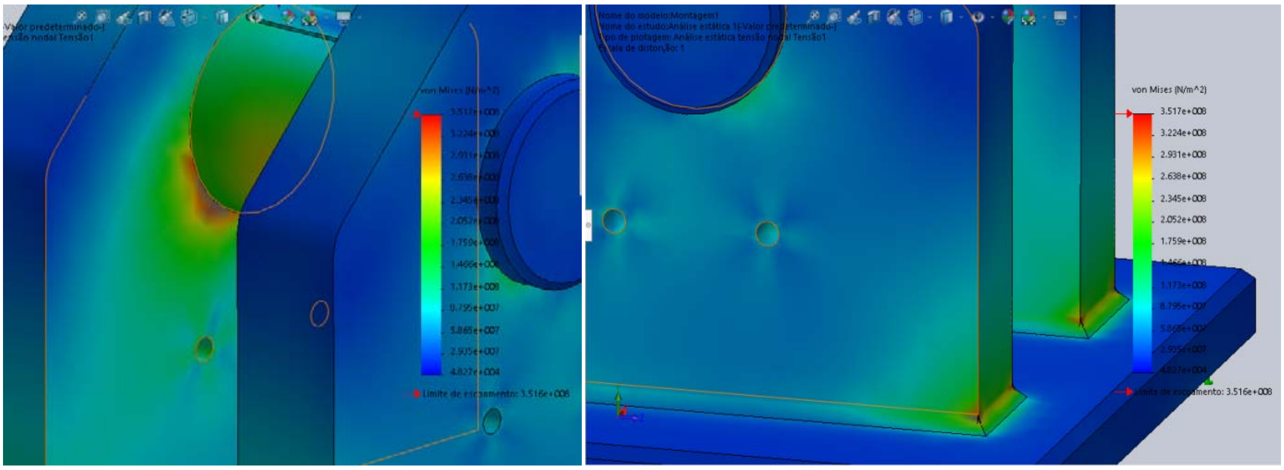


Fig. 10 SOLIDWORKS, Higher stress points.

6. Conclusion

In the bench simulation with only one bracing, was present a critical stress greater than the ASTM-A36 material yield stress, therefore a modification was necessary in the structure in order to support the requested load. After the results it was possible to verify that the use of a double bracing improved the distribution of the maximum VonMises stress in 84%. In the SOLIDWORKS with the mesh obtaining 22,500 elements and 36,026 nodes had its value of 351.7 MPa. Through the simulations made in the cylinder holder were pointed out regions where failures could occur, so the FEA tool was of great importance in the work.

Through the study it was possible to predict failures that could occur during the material extrusion process, so that it could solve the problems pointed out even before the bench was fabricated. These computational tools are fundamental to the engineering area, which always seeks to optimize resources, reducing material costs, labor costs, time and resources consumed in experimental tests.

Acknowledgments

The present research project was funded by the Pontifical Catholic University of Minas Gerais, Project number 2016/11050-S2. The authors also acknowledge funding from CNPQ — Conselho Nacional de Desenvolvimento Científico e Tecnológico

References

[1] P. F. Rodrigues Das, Aplicação da estampagem incremental no fabrico de moldes: Estudo de viabilidade técnico-económica para industrialização, , Dissertação (Mestrado) — Faculdade de Engenharia da Universidade do Porto Integrado em Engenharia Mecânica, Porto, 2013, p. 91.

[2] Silva, Gilmar Cordeiro da. Simulação numérica e experimental no processo de extrusão indireta a frio, considerando o efeito do caminho de deformação/Gilmar Cordeiro da Silva. Belo Horizonte, 2011, p. 176.

[3] Soares Carla A. T. et al., Influência do ângulo e do coeficiente de atrito nas tensões residuais geradas na trefilação de barras de aço AISI 1045, *Estudos Tecnológicos em Engenharia* 9 (2013) (2) 54-62, accessed on 11 Apr. 2016, available online at: http://revistas.unisinos.br/index.php/estudos_tecnologicos/article/view/ete.2013.92.01/3833&gt.

[4] Jiayong Si, Fan Gao, Pengbiao Han and Ji Zhang, Simulation on extrusion process of TiAl alloy, *Intermetallics* (Oct 2010) 169-174.

[5] L. G. Deshpande, N. M. Sawahli and R. B. Randall, Application of finite element model updating and reduction techniques to simulate gearbox bearing faults, *Australian Journal of Multi-Disciplinary Engineering* 10 (2013) (2) 109-119.

[6] G. E. Dieter, *Metalurgia Mecânica*, Guanabara Dois (Ed.), Rio de Janeiro, Brasil, 1986, p. 561.

[7] F. Bresciani, *Conformação Plástica do Metais* (5th ed.), Da Unicamp (Ed.), Campinas, São Paulo, Brasil, 1997, p. 383.

[8] Corrêa, Elaine Carballo Siqueira. Aspectos do Encruamento de Metais Previamente Deformados a Frio. 2004. 262f. Tese (Doutorado) - Programa de Pós-Graduação em Engenharia Metalúrgica e de Minas,

Universidade Federal de Minas Gerais, Belo Horizonte, 2004. pp. 233.

[9] N. Akikazu, Y. Takaki and K. Maszuri, Behaviour of residual stress and srawing stress in conical-type and cauch-type die drawing FEM simulation and improvement, *Wire Journal International* l (2007) 72.

[10] Z. Wang and B. Gong, Residual stress in the forming of materials, in: G. E. Totem, *Handbook of Residual Stress and Deformation of Steel*, ASM International, 2002, p. 141.

[11] H. Helman and P. R. Cetlin, *Fundamentos da conformação mecânica dos metais*, São Paulo: Artliber, 2005.