IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 15, Issue 4 Ver. II (Jul. - Aug. 2018), PP 37-44
www.iosrjournals.org

Computer-Aided Manufacturing Processes: A Comparison AndDecision Model For Selecting An Analytical Or Numerical Approach To A Drawing Design

Moisés Xavier[1], Tamires Alves[2], Pedro Peixoto[3], Francisco Dos Santos[4]andGustavo Rodrigues[5]
[1,2,3,4,5]*(Department of Mechanical Engineering, Military Institute of Engineering, Brazil)*
CorrespondingAuthor:Gustavo Rodrigues (simao@ime.eb.br)

Abstract: *This article's main function is to analyze the possibilities and applications of numerical-computational analysis (in this case, the Abaqus software) to drawing considering both problem solving and project definition related to the manufacturing process. Therefore, this work starts with a comparison between analytical and numerical analysis of the process, mentioning the advantages and disadvantages, as well as the indicated application of each; and with a presentation of the Abaqus software potential and limitations. Next, a theoretical case study is proposed to better analyze this problem, testing the differences between the two methods (analytical and computational) of a specific problem solving in drawing process. The results are, finally, compared generating further conclusions.*

Keywords :*Drawing process, Numerical Analysis, Abaqus, Finite Element Method, Slab Method*

Date of Submission: 02-07-2018 Date of acceptance: 21-07-2018

I. Introduction

The drawing process (see schematic on Figure 1) is a manufacturing process, characterized by the application of a force after the drawing machine (or matrix), 'pulling' the metal through it and reducing, therefore, the transverse section of the material.

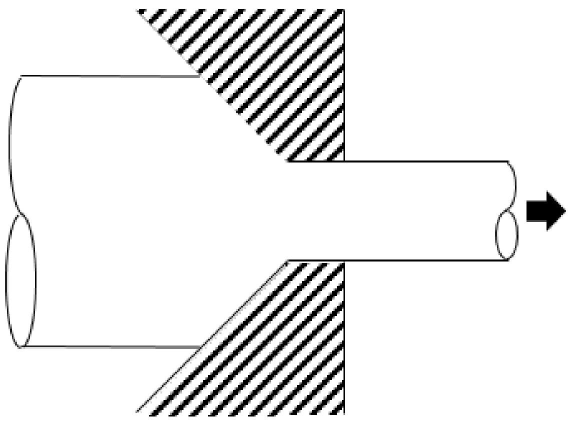

Figure 1. Scheme representing the drawing process. [1]

The drawing process aims to fabricate mainly: wires, narrow bars, or tubes. In the case of perforated tubes, an internal mandrel must also be employed.

Intermsofengineeringdesign,themainobjectivesrelated to this fabrication process are: to define how many drawing machines are necessary; and what is the amplitude of the tensionsinvolved.Forthat,therearetwomainmethods:

(1) Analytical Method: consists in solving the problem through a formula, with a usually large number of simplifications.

- Method whose application does not dependon geometric modeling usingcomputing;
- Quick application and adaptation in problems with unknownrequirements/situations*;*
- Good as a first estimate to aproject.

(2) Numerical Method: consists in obtaining an iterative solution, which requires, generally, the use of a computer to perform the integral analysis with certainty. It usually consists in the partition of the drawing metal (material) in a grid of finite elements — which may be continuously reduced. Each finite element has its own constitutive equation, and relates to its 'peers'.

Advantages:

- Considers the tension concentration in theanalysis;
- Allows the creation of geometries with higher complexity and larger optimizationpotential;
- Offers more precisesolutions.

Employing both these methods appropriately, a proper approach, in terms of engineering, would depend on the geometry complexity — and how much the geometry is known. If the geometry is simple or unknown, it is better to execute a first estimation using the analytical method — which is quicker in these situations; after that, the result may be confirmed or corrected by the numerical method, which will provide a more detailed analysis, considering the tensionconcentration—andothergeometricalcomplexities. Otherwise, if the geometry is both complex and known, the ideal approach is to compute the numericalsolution from the beginning — saving time regarding the analytical development. These aspects are summarized by thedecision model presented in Figure2.

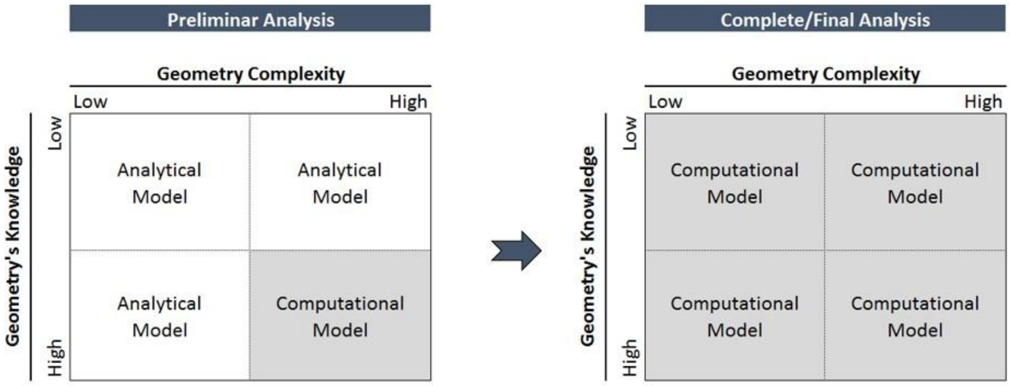

Figure 2. Decision tool to support choice between analytical or computational

Noticethat,despitethecomputation analysisbeingmore precise (when used with high levels of grid refining), itdoes not eliminate, in all cases, the advantages of the analytical problemsolving— itisquitetheopposite,asitsusereduces the analysis time significantly,by:

(a) Providingpreliminaryresultsofthenumberofdrawing machines necessary to achieve the requirements;and

(b) Verifying if the requirements are too extreme for the machinery, generating tensions that are too large (not supported), with short analysisperiod.

II. Theory

The analytical approach focus on evaluating the tension required to accomplish the drawing process. In this work, the Slab Method is employed. This method is derived from the force balance over a finite element of the material under deformation, as shown in Figure 3.

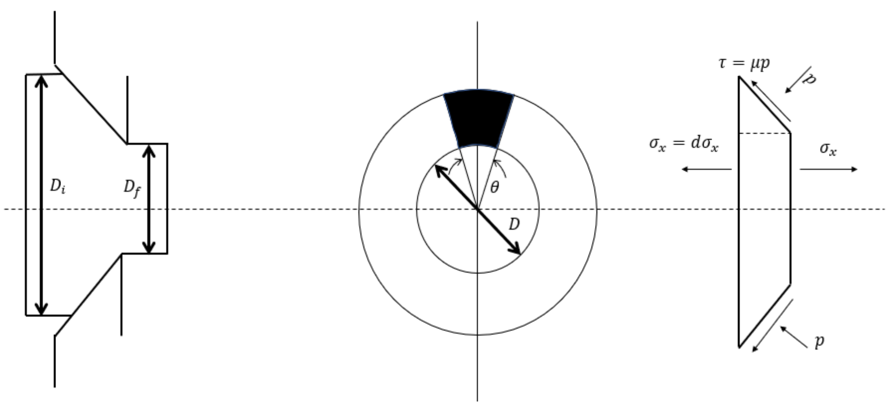

Figure 3. Geometries related to the drawing model

The force balance throughout the axial direction gives:

$$\frac{d\sigma_x DdD}{2} + \frac{D^2 d\sigma_x}{2} + pDdx \tan\alpha + \mu pDdx \quad (1)$$

The geometry of Figure 3 associated to the Tresca criteria, and adopting $B = \mu \operatorname{cotg} \alpha$, yields the following expression:

$$\frac{d\sigma_x}{B\sigma_x - Y(1+B)} = \frac{2dD}{D} \quad (2)$$

In (2), Y is the average yield strength, μ is the friction coefficient, $D_f$and $D_i$are, respectively, the final and initial diameters of the bar being deformed. With the boundary condition of D = $D_i$for σ_x= 0, the integration of this expression results in the equation for the drawing tension.

$$\sigma_f = \frac{Y(1+B)}{B}\left[1 - \left(\frac{D_f}{D_i}\right)^{2B}\right] \quad (3)$$

When a tension is added backwards (σ_{back}), boundary condition turns into D = $D_i$for σ_x= σ_{back}. So the equation becomes:

$$\sigma_f = \sigma_{back}\left(\frac{D_f}{D_i}\right)^{2B} + \frac{Y(1+B)}{B}\left[1 - \left(\frac{D_f}{D_i}\right)^{2B}\right] \quad (4)$$

The numerical analysis is carried out through the Finite Elements Method (FEM). The FEM consists of dividing the material in shorter pieces, called elements. The object is now described by the sum of the properties of its elements. The greater is the number of elements, the accurate is the simulation,andtheprocessingtimeaswell,sincetheamount of calculations is proportional to the number ofelements.

III. Analytical Vs. Computational Solutions

This article proposes a case study about drawing, considering that this is the most efficient method of accomplishing two objectives:

(1) To expose a clear comparison between the analytical and computational solutions of a practical project problemregardingthedrawingprocess,withemphasis to the advantages of eachmethod;

(2) To show the potential applications of the Abaqus software,usedinthenumerical-computationanalysis regarding fabrication processes — in this case, drawing.

This way, the following sections present the proposed theoretical problem, used as a case study, and then provide the analytical and computational solutions.

Proposed Theoretical Case Study

Analuminumbarmusthaveitsdiameterreducedfrom3.0cm to 2.4cm. The matrix's angle is of 45º, and the friction coefficient μ=0.15. The yield tension for the material may be approximated by Y = 270 MPa. For those conditions, the case study consists inanswering:

(a) what is the resulting drawingtension;

(b) analytically, which is the minimum tension when two drawing machines are employed, consecutively, in the process.

Analytical Solution

The first objective (a) related to the analytical solution is to find the tension in the drawing process, resulting from the reductioninthetubediameter.Thattensioncanbecalculated usingtheequation1.TheBfactormaybecalculatedfromthe matrix angle (2α)and the friction factor (μ).The relation is described by the followingequation:

$$B = \mu\cot\alpha = 0.15\cot 22.5^\circ = 0.3621 \quad (5)$$

Substituting this value on equation 3, it is possible to find the resulting drawing tension:

$$\sigma_f = \frac{270(1+0.3621)}{0.3621}\left[1 - \left(\frac{2.4}{3.0}\right)^{0.7242}\right] = 151.56\text{MPa} \quad (6)$$

Noticethat,fromequation3,itisrelativelyfasttofindthe tension — even considering that this result is a (reasonable) approximation.
Turning to the second problem (b), we now consider two drawing machines used to make the same reduction. The minimum tension occurs when the tensions in both the machines are equal. This is clearly observed when we see that,byreducingthetensioninanyofthedrawingmachines, the tension in the other will unequivocally rise. Therefore, for a constant tension, the diameter reductions willbe:

$$\left(\frac{D_f}{D_i}\right)_1 = \left(\frac{D_f}{D_i}\right)_2 \rightarrow \frac{D_{half}}{3.0} = \frac{2,4}{D_{half}} \rightarrow D_{half} = 2.68\text{cm} \quad (7)$$

That gives means to calculate the tension. Note that the process's tension is the same in both the machines. Finally, using the result in equation 3, the tension is:

$$\sigma_f = \frac{270(1+0.3621)}{0.3621}\left[1-\left(\frac{2.68}{3.0}\right)^{0.7242}\right] = 78.86\text{MP} \quad (8)$$

It is interesting to notice that this value (equation 8) is not half of the first tension calculated in equation 6 (indeed, it is larger).

Computational Solution

The numerical simulation accomplished with the software Abaqus CAE aims to reproduce the conditions of a true wire drawing process, by employing Finite Element Method (FEM). The essential parameters to be determined for the modelinvolvetheattributionofdie/billetgroupgeometry, that in this case was adopted as an axisymmetric set, which means that the calculations are performed based on a plane section of the model, considering axial symmetry. This procedure allows extrapolation of results, since a minorquantity of variables can be solved and repeated, reducing the time required to obtain a solution [2].
The die/billet geometry consists of an aluminum wire, modeled as a deformable body, and the spinneret, modeled as a discrete rigid body, with a view to its extremely high hardness when compared to the wire. The assembly resultingfromdrawingbothpartsinAbaqus,andthedetailed Sketchesincludingdimensionsofthemodelareillustratedin Figure4.

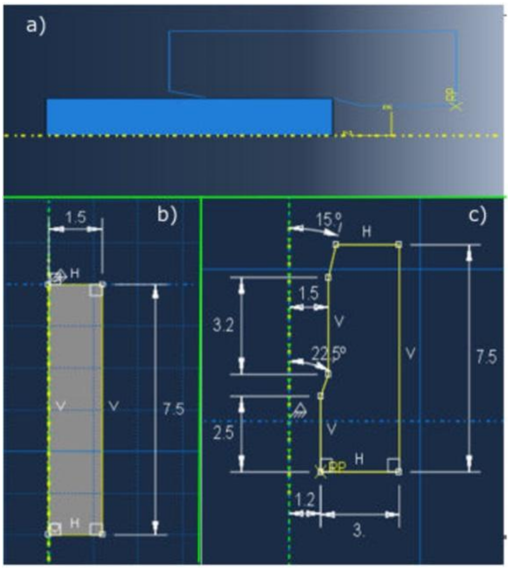

Figure 4. Scheme containing assembly (a) and sketches with dimensions of the parts (b) and (c)

In this configuration, the wire drawing diameter passing through the die is to be reduced from D_i= 3 cm to D_f= 2.4 cm. The wire length, estimated at L = 7.5 cm, should be enough in order to present a region exempted from plastic deformation after the simulation.

Once the interest geometry and the wire material have been defined (for a rigid body this definition is not possible in the software), it is now necessary to attribute the known properties to the material aluminum. In this case, those properties are listed in Table 1, for the chosen material aluminum 6061 alloy.

Table 1. Properties of material aluminum and values inserted in Abaqus program

	Property value	Abaqus Input
ρ (density)	2700 kg/m^3	2.7
E (Young Modulus)	69 GPa	0.69
ν (Poisson Coefficient)	0.34	0.34
Y (Yield Stress)	270 MPa	0.0027

Whenintroducingthosevaluesintheprogram,theunitsmustbeadjustedtothesameconsideredinthesketchingoftheset die/billet,whichweretakenincentimeters,asAbaqusworks with dimensionless base. The units were adjusted according to the standardized system presented in Table2.

Table 2. Units adjustment obtained from [3].

Mass	Length	Time	Force	Stress	Energy
kg	M	s	N	Pa	Joule
kg	mm	ms	kN	GPa	kN-mm
G	cm	s	dyne	$dyne/cm^2$	erg
G	cm	μs	10^7N	Mbar	10^7 N-cm
G	mm	s	10^{-6} N	Pa	10^{-9} J
G	mm	ms	N	MPa	N-mm
ton	mm	s	N	MPa	N-mm
$lbf\text{-}s^2/in$	in	s	lbf	psi	lbf-in
slug	ft	s	lbf	psi	lbf-ft

When it comes to the material modeling, 6061-aluminum was incorporated through Johnson-Cook elasto-viscoplastic material model. It includes the effect of linear thermo- elasticity, yielding, plastic flow, isotropic strain hardening, strain rate hardening, softening due to adiabatic heating and damage [4]. The equivalent von Mises stress of the Johnson- Cook model is expressed as

$$\sigma = (A + B\varepsilon_P^n)\left(1 + C\left(\ln\frac{\dot{\varepsilon_P}}{\dot{\varepsilon_0}}\right)\right)\left(1 - \left(\frac{T - T_r}{T_m - T_r}\right)^m\right) \quad (9)$$

The Johnson-Cook parameters employed for the numerical analysis are listed in Table 3

Table 3. Bilinear Johnson-Cook material properties for Al-6061, obtained from [5]

A (MPa)	B (MPa)	n	C_1	C_2	m	T_m	ε_0	ε_P
270	154.3	0.2215	0.002	0.1301	1.34	925	1	597.2

Another important aspect to be explored in the simulation is the establishment of interactions between the parts. To do so, the master and slave surfaces are defined as shown in Fig.2, where master surface is formed by a die segment (in red), whilst slave surfaces are formed by billet profiles (in magenta). The friction coefficient adopted was μ = 0.15, considering penaltyformulation.

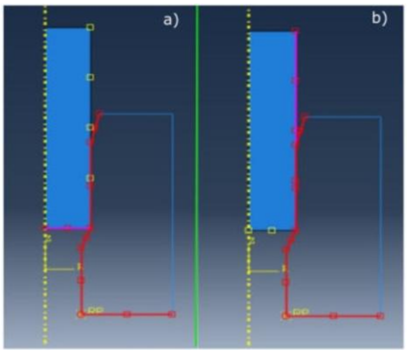

Figure 5. Interaction between the billet and die.a)the interaction is taken between a billet horizontal slave surface and the die master surface; (b) the slave surface taken is a vertical one

For the numerical simulation, a step according to the increments described in Table 4 was created.

Table 4. Increments considered for numerical analysis in step 1

InitialIncrement	MinimumIncrement	MaximumIncrement	TimePeriod
0.1	10^{-5}	1	1

In this step, a displacement of 5 centimeters was applied to the wire frontal face in the negative direction of y-axis, while the die was maintained crimped through a previously defined reference point. The wire surface was restricted to movejuxtaposedtothemodelaxis,inawayitonlytranslates in y-axis. Such boundary conditions were then inserted into the model as shown in the Figure6.

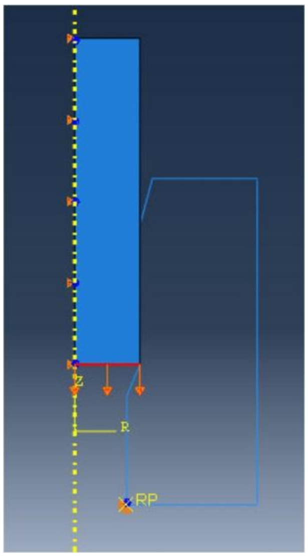

Figure 6. Boundary conditions applied to the model

Finite element mesh in this model is composed of two types of elements: CAX4R for the wire, with two- dimensional elements, 4-node and reduced integration, as it provides the advantage of less computational cost and the tendency of making the elements more flexible [2]; RAX2 for the spinneret, with 2-node linear axisymmetric rigid link, which is a typical element type for discrete rigidbodies.

IV. Results

Various numerical simulations were carried out in order to obtain the wire drawing stress corresponding to the die exit region for a reduction of the wire diameter from D_i= 3 cm to D_f= 2.4 cm. From the mesh definition, this stress was calculated for the following element sizes until convergence of results: 1 cm, 0.5 cm, 0.1 cm and 0.05 cm.

The results found can be observed in Figure 7, Figure 8, Figure 9 and Figure 10for each of the element sizes, according to both von Mises formulation and Tresca criterion. The stress along the wirecan be visualized in form of gradually colored bands showing the tension state at the whole part. By analyzing the drawings, it can be concluded that the solutions are coherent and converged to a tension of 150 MPa according to von Mises and 158.1 MPa according to Tresca, which is a pretty similar result in relation to the obtained through analytical calculations: 1.03% of difference for von Mises and 4.32% forTresca.

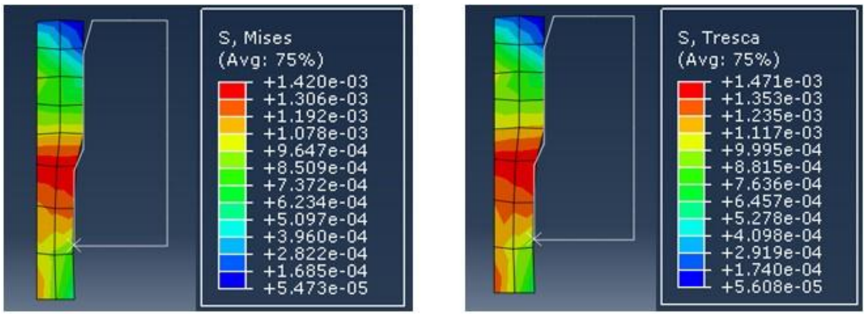

Figure 7. Mesh 1 cm a) Tresca, b) von Mises

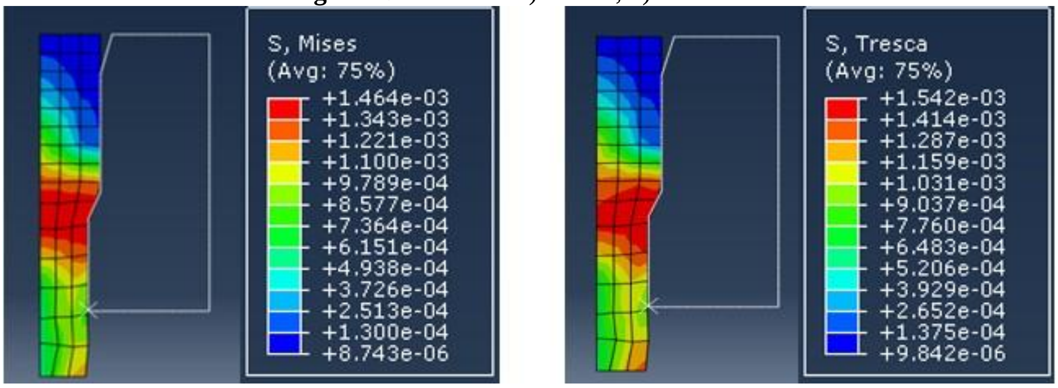

Figure 8. Mesh 0.5 cm a) Tresca, b) von Mises

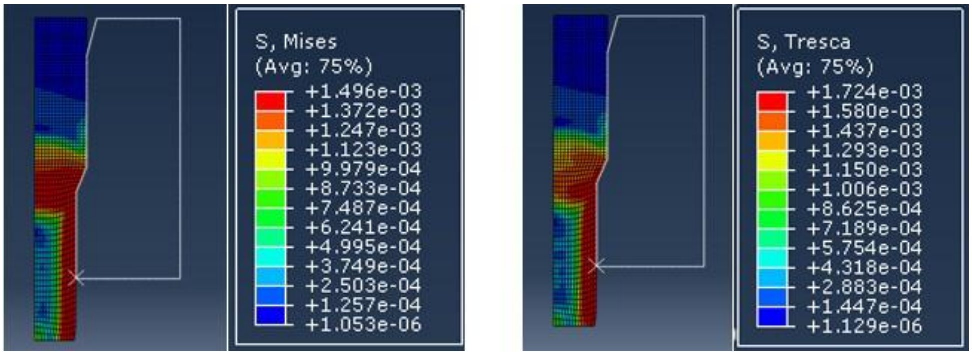

Figure 9. Mesh 0.1 cm a) Tresca, b) von Mises

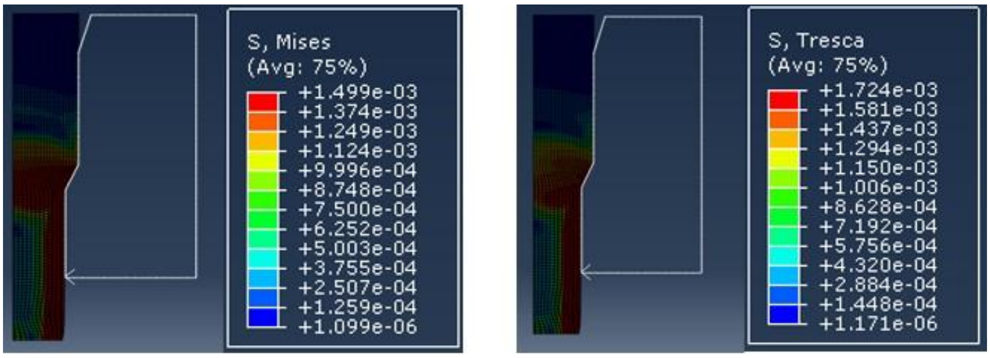

Figure 10. Mesh 0.01 cm a) Tresca, b) von Mises

Due to the mesh refinement, a considerable influence in the results can be noticed, especially because of the choice of employing reduced integration and a mesh with few elements. A difference of 5.2% between the values obtained using 1 cm and 0.05 cm element sizes for von Mises stress criterion and of 14.4% for Tresca stress criterion, somewhat higherforthesameelementssizesconsidered,wasobserved. With this study, the calculations previously performed by analytical method could be validated. Nonetheless, it must beconsideredthatthefiniteelementmethodisapproximated and does not give us an exact answer, but in this case a satisfactory result was reached, corresponding to the actual and practical necessities.

ThenumericalsimulationswereruninanIntelInsideCore i5-3210M processor workstation with 2.5 GHz CPU and 8 GB RAM Memory. The simulation time of the mesh 0.01 cm was approximately 5minutes.

V. Conclusion

As discussed, the numerical methodology based on finite elements and proceeded through the Abaqus software has validatedtheanalyticalone,sincetheresultsforthedrawing tension were very similar, with a difference of 1.03% (von Mises) and 4.32% (Tresca) when compared with the analytical approach. In this work, the decision of using the analytical or numerical approach would be easy, since the geometry is pretty much simple. In another processes, it is recommended to apply the decision toolprovided.

References

[1]. M.P. Groover, *Introdução aos Processos de Fabricação* (Rio de Janeiro, RJ: LTC Editora, 2014).

[2]. J.P. Lottermann, *Análise numérica e experimental do processo de trefilação de fios aplicados a cabos condutores,*master diss., Universidade de Brasília, Brasília, DF, 2017.

[3]. P. BARRET, *Tips & tricks for consistent FEA units,* 2017. [Online]. Available: https://caeai.com/blog/tips-tricks- consistent-fea-units

[4]. M Iqbal, S Khan, R Ansari, and N Gupta, Experimental and numerical studies of double-nosed projectile impact on aluminum plates, *International Journal of Impact Engineering, 54,* 2013, 232-245.

[5]. A. A. Dehkharghani, *Tuning johnson-cook material model parameters for impact of high velocity, micron scale aluminum particles,* master diss.,Northeastern University, Boston, Massachusetts, 2016.