CILAMCE 2017
Florianópolis - Santa Catarina - Brazil

HARDENING PARAMETERS IDENTIFICATION BY CAUCHY OPTIMIZATION METHOD

Lucas Mangas Araújo

Bruno Arruda Silva

Lucas Queiroz Machado

Lucival Malcher

Lucasmangas96@gmail.com

Bruno.arruda.93@outlook.com

Lucasqmachado@gmail.com

Lucical.malcher@gmail.com

University of Brasília, Mechanical Engineering Department

Campus Universitário Darcy Ribeiro, 70910-900, Distrito Federal, Brasília, Brazil

Abstract. *The aim of this work is the determination of hardening parameters of engineering materials by the Cauchy Optimization method. The mathematical equation used in the fitting of the hardening curve was the Ludwick equation of 3 parameters, given its simplicity and ability to model the behavior of several materials. The objective function is based in the least squares method, due to its quick convergence in most optimization methods. The Finite Elements Method (FEM) was used to simulate the mechanical behavior of the specimen and to obtain the numerical response. Tensile tests were carried out in different materials to verify the analysis. The Cauchy algorithm developed showed a quick convergence, with satisfactory visual fitting and providing hardening parameters with an error less than 0.1 % and close enough of those obtained using classical methods of parameters identification.*

Keywords: *Cauchy Method, Finite Elements Method, Hardening, Optimzation*

1 INTRODUCTION

Hardening, also known as straightening, is a phenomenon that materials experience when their proportionality limit is transposed under a load or displacement, characterized by the presence of permanent strains, increase in mechanical resistance and hardness. The straightening of materials, despite being a classical field in engineering, remains one of the most studied subjects today. Information contained in the hardening process can be used in other important areas, such as damage mechanics. Thus, the identification of hardening parameters given certain elastoplastic model is fundamental.

With the constant development of new materials, a great deal of attention has been given in the implementation of optimization numerical methods to obtain more accurate parameters and with a shorter computational time. In this context, the gradient based methods proved to be a useful tool in this kind of analysis (Stahlschimidt, 2010). Among the studies in the area, stands out the works done by Abendroth and Kuna,(2003); Kleinermann and Ponthot,(2003). The first one presents parameter identification by small disk inlay test, using the constitutive models of Roussilier (Roussilier, 1987) and of Gurson (Gurson, 1987). The second one presents a parameter identification analysis by stamping tests, using the classical model of Von Mises and different types of gradient methods in the numerical analysis.

The aim of this present work is to obtain hardening parameters by tensile tests using the Cauchy Method (Cauchy, 1987). For simplicity, the equation uses was the Ludwick equation of 3 parameters (Ludwick, 1909). In addition to the optimization method, the Finite Elements Method was used to simulate the mechanical behavior

2 HARDENING AND ELASTOPLASTIC MODELS

The hardening phenomenon, as described previously, is characterized by the presence of permanent strains and enhancement of hardness and mechanical resistance. In stress versus strain graphics, this phenomenon is found in the portion above the initial yielding strength. This region in the stress and strain graphics is often modeled as a function of the true plastic strain:

$$\sigma_Y = \sigma_Y(\bar{\varepsilon}_P) \tag{1}$$

Many models have been described throughout history to model this curve in mathematical manner, each with its advantages and disadvantages. Due to the complexity of the phenomenon, many idealizations must be made in order to mathematical describe the hardening process. For the purposes of this work, it is considered that the straightening is non-linear and isotropic.

Table 1 presents the most significant elastoplastic mathematical models in the literature.

The mathematical model used in the present work, as said before, is the Ludwick equation. This model has as its vantage its simplicity and ability to model well, under certain limits, the behavior of several materials. In this model the parameters are the initial yielding strength,σ_{Y0}, the hardening isotropic module, H, and the hardening exponent, n.

The model, despite being mathematically simple as a vantage, fails by underestimating the triaxial stress states that occurs in the hardening process. Thus, in some cases, the equation doesn't capture well the straightening phenomenon in certain materials in which this process is more pronounced. For an analysis that takes in count this stress state, the Kleinermann and Ponhot is the most indicated (Stahlschimidt, 2010). Thus, the model used is:

$$\sigma_Y = \sigma_{Y0} + H\bar{\varepsilon}_P{}^{-n} \tag{2}$$

Table 1 Most important hardening mathematical models.

Model	Parameters	Model Name
$\sigma_Y = \sigma_{Y0} + H\bar{\varepsilon}_P{}^{-n}$	σ_{Y0}, H, n	Ludwick, 1909
$\sigma_Y = tanh\left(\frac{E\bar{\varepsilon}_P}{\sigma_{Y0}}\right)$	σ_{Y0}, E	Prager, 1938
$\bar{\varepsilon}_P = \frac{\sigma_Y}{E} + H\left(\frac{\sigma_Y}{E}\right)^n$	H, E, n	Ramberg-Osgood, 1943
$\sigma_Y = \mathrm{H}\bar{\varepsilon}_P{}^{-n}$	H, n	Holloman, 1944
$\sigma_Y = H(\varepsilon_s + \bar{\varepsilon}_P)^n$	H, ε_s, n	Swift, 1947
$\sigma_\mathrm{Y} = \sigma_{\mathrm{Y}0} + (\sigma_s + \sigma_{Y0})(1 - e^{-n\bar{\varepsilon}_P})$	$\sigma_{\mathrm{Y}0}, \sigma_s, n$	Voce, 1948
$\sigma_\mathrm{Y} = \sigma_{\mathrm{Y}0} + q_m(1 - e^{-d\bar{\varepsilon}_P}) + H\bar{\varepsilon}_P$	$\sigma_{\mathrm{Y}0}, q_m, d, H$	Mahnken, 2002
$\sigma_Y = \sigma_{Y0} + \xi\bar{\varepsilon}_P + (\sigma_\infty - \sigma_{Y0})\left(1 - e^{-\delta\overline{\bar{\varepsilon}_P}}\right)$	$\sigma_{\mathrm{Y}0}, \xi, \sigma_\infty, \delta$	Kleinermann-Ponhot, 2003

3 OPTIMIZATION METHODS AND ENGINEERING PROBLEMS

3.1 Optimization Problems

Engineering optimization problems are divided into two groups basically, the inverse and the direct ones, the first one being further subdivided into two another groups: the pre-form problems and the parameter identification. The direct problems are the most common in engineering. These kinds are the ones involving design, dimensioning or analysis of component or structure. In these cases, the initial geometry, the constitutive model that describes the phenomenon, the material properties and the boundary conditions are all known. The final objectives of these type of problems are the final geometry, stresses and strains by the end of the process and how they developed during the process. The inverse problems, on the other hand, seek to obtain the initial geometry, the boundary conditions, mechanical properties or the constitutive model of the process from the information at the end of the phenomenon.

The pre-form optimization problems seek to obtain the initial geometry that a certain component must have so it can attend a specific final geometry and the end of the studied process. In the parameter identification ones, the major objective is to achieve the constituve models or the parameters that describes the phenomenon.

The present work lies in the last group of inverse problems, and the constitutive models is already known. Thus, it is desired to obtain the hardening parameters of some ductile engineering materials by the use of data from tensile tests. As usual in engineering, this identification utilizes numerical methods.

3.2 Optimization Methods

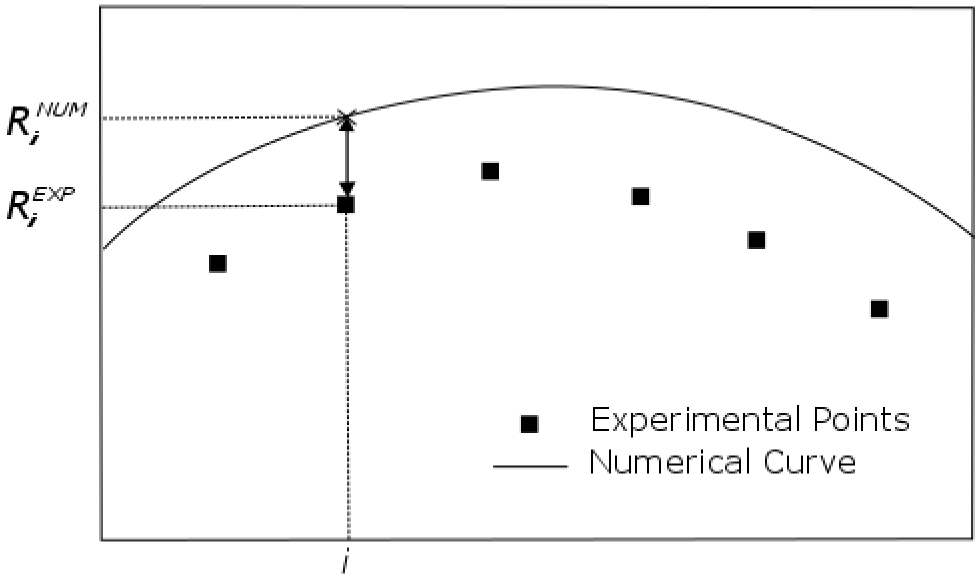
The optimization methods used in the in parametric identification analysis seek to achieve a vector $\boldsymbol{p} = (p_1, p_2, \ldots, p_n)$ capable of minimizing an objective function $f(\boldsymbol{p})$. This kind of function usually relates an numerical response $R^{NUM}(\boldsymbol{p})$ and an experimental response R^{EXP} . The response could be, for example, in means of force or hardness. Note that the numerical response is a function of the parameters, usually obtained based on the chosen constitutive model. For this work, the vector $\boldsymbol{p}$ is defined as follows

$$\boldsymbol{p} = (\sigma_{Y0}, H, n) \tag{3}$$

In the case of this work, R^{EXP} is obtained from the force data provided from the tensile tests carried out on the specimens of the selected materials. $R^{NUM}(\boldsymbol{p})$ is obtained by solving the direct problem using the Finite Elements Method (FEM). The objective function chosen to compare $R^{NUM}(\boldsymbol{p})$ and R^{EXP} is based in the least square method:

$$f(\boldsymbol{p}) = \sqrt{\frac{1}{N}\sum_{i=1}^{N}\frac{\left(R_i^{NUM}(\boldsymbol{p}) - R_i^{EXP}\right)^2}{R_i^{EXP}}} \tag{4}$$

Figure 1 illustrates this method



According to Kleinermann (2000), the objective function based on the least squares method is the most suitable choice due to its quick convergence in most optimization methods.

In single variables functions, the conditions for a certain point to be a minimum is that the first derivative at the point must be zero, and that the second derivative at the same point must be positive. For the case of many variables functions, it is required that the gradient of the function calculated at the candidate to a minimum $\boldsymbol{p}^*$ is zero:

$$\nabla f(\boldsymbol{p}^*) = 0 \tag{5}$$

However, the described condition doesn't guarantee that $\boldsymbol{p}^*$ is a minimum point. Analogously to one variable functions, another condition is required. It is necessary that the Hessian matrix calculated at $\boldsymbol{p}^*$ obeys to the following condition:

$$\boldsymbol{d}^{\boldsymbol{T}}\boldsymbol{H}(\boldsymbol{p}^*)\boldsymbol{d} > 0 \tag{6}$$

Where $\boldsymbol{H}(\boldsymbol{p}^*)$ is the hessian matrix calculated in $\boldsymbol{p}^*$ and $\boldsymbol{d} = \boldsymbol{p} - \boldsymbol{p}^*$. It is worth noting that this condition is only true if $\boldsymbol{H}(\boldsymbol{p}^*)$ positive definite. Many times, as in the case of this job, $f(\boldsymbol{p})$ is not known analytically in terms of the interest parameters. Thus, in the optimization process, it is not possible to analytically solve the problem, requiring numerical resolution.

3.3 Cauchy Method

The optimization numerical methods based on gradient usually have the following format:

$$\boldsymbol{p}_{k+1} = \boldsymbol{p}_k + \alpha_k \boldsymbol{d}_{\boldsymbol{k}} \tag{7}$$

Where $\boldsymbol{p}_{\boldsymbol{k}}$ is the current iteration point, $\boldsymbol{p}_{\boldsymbol{k+1}}$ is the following iteration point, α_k is the optimal step length and $\boldsymbol{d}_{\boldsymbol{k}}$ is the search direction in the current iteration. The iterative process terminated when a predefined stop criterion is satisfied. The main difference between the gradient methods lays in how the search direction is calculated. In the present, the method used is the Cauchy Method, also known as the steepest descent method. The analysis consists in using the gradient as the search direction, since the gradient vector has the property of always be in the direction of maximum increase of the function. Since it is desired to minimize $f(\boldsymbol{p})$, the search direction is defined as the reciprocal of the gradient. Therefore, $\boldsymbol{d}_{\boldsymbol{k}}$ is calculated as follows:

$$\boldsymbol{d}_k = -\nabla f(\boldsymbol{p}_{\boldsymbol{k}}) \tag{8}$$

When the partial derivatives of f in relation to each parameter are not available or are difficult to calculate, the finite difference method is used to estimate each derivative and consequently the gradient of f. In any gradient-based optimization method, it is necessary to calculate the optimal step length. This is necessary due to the fact that the gradient property of pointing to a maximum or a minimum is a punctual characteristic. Therefore, α_k shows how much one must walk in the direction $\boldsymbol{d}_k$. Mathematically, α_k is a scalar that minimizes the following function of one variable.

$$f(\boldsymbol{p}_{\boldsymbol{k}} + \alpha \boldsymbol{d}_{\boldsymbol{k}}) = f(\alpha) \tag{9}$$

Notice that f is the objective function. The achievement of α_k that minimizes Eq. 9 is done in a numerical way. Since the analytical optimization is impractical. The one dimensional optization method used to calculate was the quadratic interpolation method, which can be found in more details in the book "Engineering Optimization" (Rao, 2009), pages 273-280. Since the Ludwick's parameters have very different orders of magnitude, $\boldsymbol{\alpha}_k$ was chosen as a vector, where each element consists of a step length related to each parameter.

The Cauchy method was chosen due to its easy programing and its relative independence in relation to the parameters initial guess. However, the method doesn't have a low convergence rate compared to other gradient-based methods. For example, the BFGS method has superlinear convergence in the vicinity of $\boldsymbol{p}^*$, however the initial guess must be relatively close to $\boldsymbol{p}^*$. The gradient-based methods have the advantage over univariate methods, such as the golden section method, for example, due to the fact that they do not need a range of values on which the searched parameter must lay for convergence.

Finally, gradient-based methods have the disadvantage of often obtaining local minimum points and global ones. To solve this kind of problem, it is commonly used hybrid methods, which uses genetic methods associated with gradient-based methods.

4 METHODOLOGY

The experimental data used to validate the numerical analysis was obtained from tensile tests performed in the following materials:

- 4340 Steel, with annealing heat treatment;
- 4340 Steel, with Normalizing heat treatment;
- R4 Steel, as received supplier;
- R4 Steel, with quenching and tempering heat treatments;
- U2 Steel, as received by supplier;
- U2 Steel, with heat treatment;

The tensile tests were carried out on specimens shown in Fig. 2. The tests also had the aid of a clip gauge with a length of L_0=25 mm. The initial diameter of useful section is d_0=8 mm.

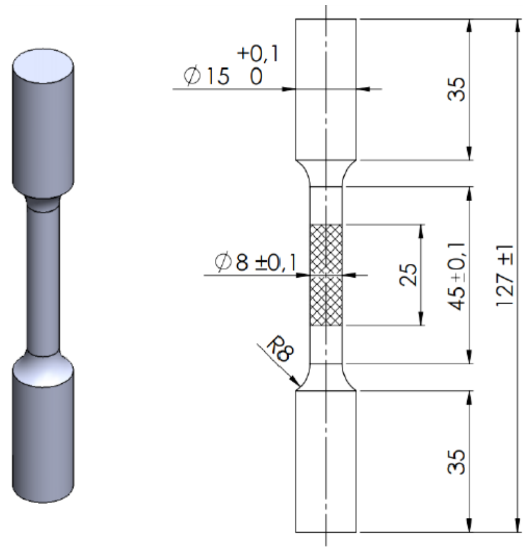


Figure 2 Dimensions of the specimens used in the tensile tests. The useful data is located on the hatched area

With the experimental data, the hardening curve was generated. The curve starts with the initial yielding strength of the material, obtained by parallel line of 0.2% of deformation method, and ends at the ultimate tensile strength. In all cases, the engineering stress and strain data were transformed into real stress and strain. Finally, the strain plastic component can be calculated as the difference between total and the elastic strains.

For comparative purposes and in order to estimate preliminary values of the studied parameters, a nonlinear fit was applied to the experimental data, the MATLAB ® *nlinfit* function and considering the Ludwick's equation as the fitting function. This methodology is usually called the classical method. In both classical and optimization methods, σ_{Y0} was fixed, so the analysis parameters were the isotropic hardening modulus H, and the hardening parameter exponent n. This was done to accelerate the calculation process.

After the experimental and pre-analysis, an algorithm was developed to obtain H and n by the Cauchy Method. The objective function used is the same as given by Eq. (4) and the simulation of mechanical behavior was conducted using the Finite Elements Method with the aid of the commercial program HYPLAS © (Souza Neto *et at,* 1996-2001). The program was made to receive as input σ_{Y0}, H, n and R_i^{EXP}, and from these entries obtain $R_i^{NUM}(\boldsymbol{p})$ (numerical force), which is used in the evaluation of Eq. (4). The mesh made for the simulation was a not well refined one consisting of 561 nodes and 500 elements. The mesh was made this way to save processing time. The mesh size does not usually have a great impact on the final parameters values (Stahlschimidt, 2010). The experimental responses used to feed the iterative process are provided from the hardening curve generated, but in terms of displacement and force. The starting point of the curve is the force and displacement associated with the initial yielding strength and the last point is the force and displacement connected with the ultimate tensile strength. The program was tested for each of the selected materials, changing only the mesh and the experimental data for each one.

For the modeling, it is considered some simplifications. First, the one-dimensional tensile test problem can be considered to be axisymmetric, so in the finite elements analysis, the situation can be thought as a two-dimension problem rather than a three-dimensional one, and only one quarter of the specimen must be modeled. Figure 3 illustrates the modeled region and the mesh used. It is worth noting that due to these simplifications, the experimental displacements must be divided in half in the file containing this data.

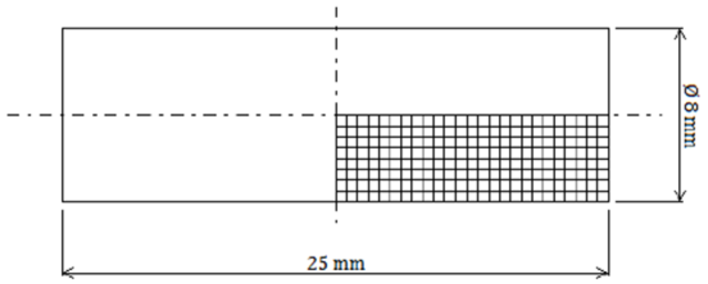


Figure 3 Illustration of the axisymmetric model showing the modeled region

5 RESULTS

Table 2 reports preliminary values of σ_{Y0}, H, n obtained using the *nlinfit* function. Two things are noted from this table. First, for all materials analyzed, the hardening modulus H is greater than the initial yielding strength σ_{Y0} of the respective material. Further, the hardening exponent n is in the range of 0 to 1. At this stage of the analysis using nonlinear fitting (classical method) the hardening data is provided for the function in terms of true stress and strain. It is important to emphasize that this analysis of obtaining parameters by means of nonlinear adjustment, although simple, is not so reliable, since the physical problem is not taken into account during the process.

Material	σ_{Y0} (MPA)	H (MPa)	n
Annealed 4340 Steel	495.0866	1081.3000	0.5499
Normalized 4340 Steel	690.4439	1389.8000	0.4002
Received R4 Steel	692.1300	821.2199	0.3414
Heat Treated R4 Steel	902.510	2091.1000	0.9913
Received U2 Steel	352.0000	888.7226	0.4443
Heat Treated U2 Steel	895.5400	1420.3353	0.8524

Based on the findings, the numerical simulation was done following a systematic logic. It was decided that the initial trial for H would be equal to the initial yielding strength, and two values of n would be tested, 0.1 and 0.5. The stopping criterion for the Cauchy method was chosen as the difference between the objective function of the current iteration less the previous one and the tolerance was of 0.1 %. In contrast to the methodology used to generate Table 2, the optimization method implemented is more realistic and reliable, since the physical problem is considered by means of the numerical response in finite elements. In addition, this methodology allows the use of other mechanical tests as experimental response.

The results obtained by the developed algorithm are in Table 3 and the plots comparing the experimental and numerical responses are shown in Fig. 4 trough Fig. 10. From Table 3and the plots shown, it is noticed that for a given initial guess of the parameters, there was a better convergence. This is due the fact that gradient-based methods have a dependence on the initial trial for its convergence. Although Cauchy's method does not have such a strong dependence, it still exists. This can be observed from the fact that when the initial estimation for the parameters was closer to its true value, the convergence is faster. It is also observed that convergence process is dominated by the exponent n. Regarding the processing time, the convergence took 32 min on average and when the initial trial of n was closer to its optimal value, the method converged faster. The optimal value is approximated by the values given in Table 2.

For some materials there was no convergence or there was convergence, but the results did not match the expectations. These were the cases of Heat Treated R4 and U2 Steels. There are two explanations for the problem. The first one is that gradient-based methods are unable to distinguish a local minimum for a global one, so there is a possibility that the process converge to local minimum point. The second one is that the Ludwick model considers a homogeneous strain state, disregarding the triaxial stress state that arises in the necking region.

Although the results were reasonable, it is possible to see that there is room for improvement. The Ludwick model may not be the most appropriate to describe the hardening process, thus other models should be tested. Another issue to be analyzed is fixing the initial yielding strength, how it would interfere in the analysis if it wasn't fixed. In addition, the finite elements software used is not an optimized one, so it is worth trying to use other commercial programs such as ABAQUS ®. Finally, other mechanical tests may be performed as experimental data for comparison.

Table 3 Values found after the numerical analysis for different initial guesses.

Material	Initial n	σ_{Y0} (MPa)	H (MPa)	n	Time(Min)	f
Annealed 4340 Steel	0.1	495.0866	670.9811	0.4018	36.33	0.0498
	0.5	495.0866	915.4984	0.5182	28.95	0.0367
Normalized 4340 Steel	0.1	690.4439	901.4706	0.2765	32.23	0.0449
	0.5	690.4439	1671.3908	0.4482	31.27	0.0248
Received R4 Steel	0.1	692.1300	700.6590	0.2854	23.89	0.0243
	0.5	692.1300	1342.000	0.4511	30.21	0.0232
Heat Treated R4 Steel	0.1	902.5100	-	-	-	-
	0.5	902.5100	-	-	-	-
Received U2 Steel	0.1	352.0000	628.0338	0.3367	43.51	0.0484
	0.5	352.0000	925.2986	0.4677	30.79	0.0277
Heat Treated U2 Steel	0.1	895.5400	-	-	-	-
	0.5	895.5400	-	-	-	-

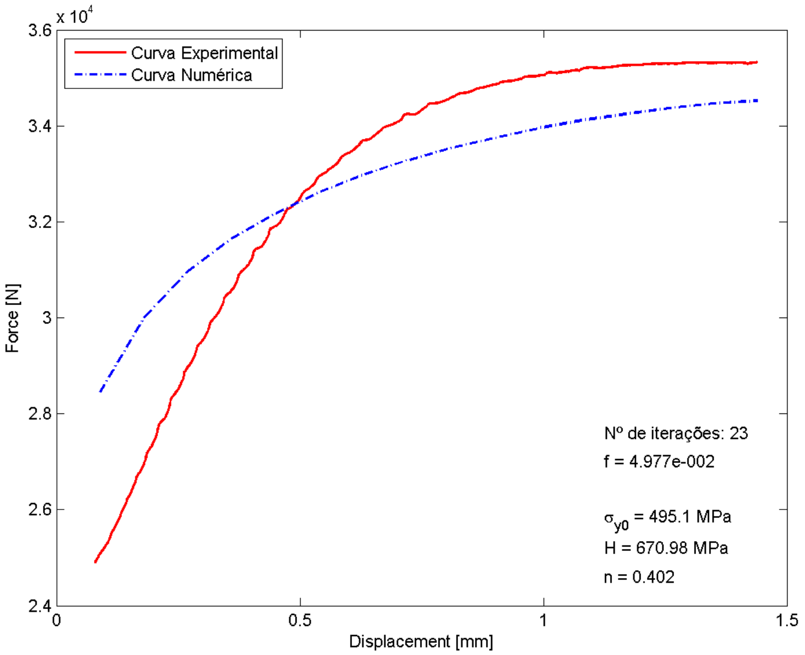


Figure 4 Comparison between numerical and experimental curves with n=0.1 for the annealed 4340 Steel

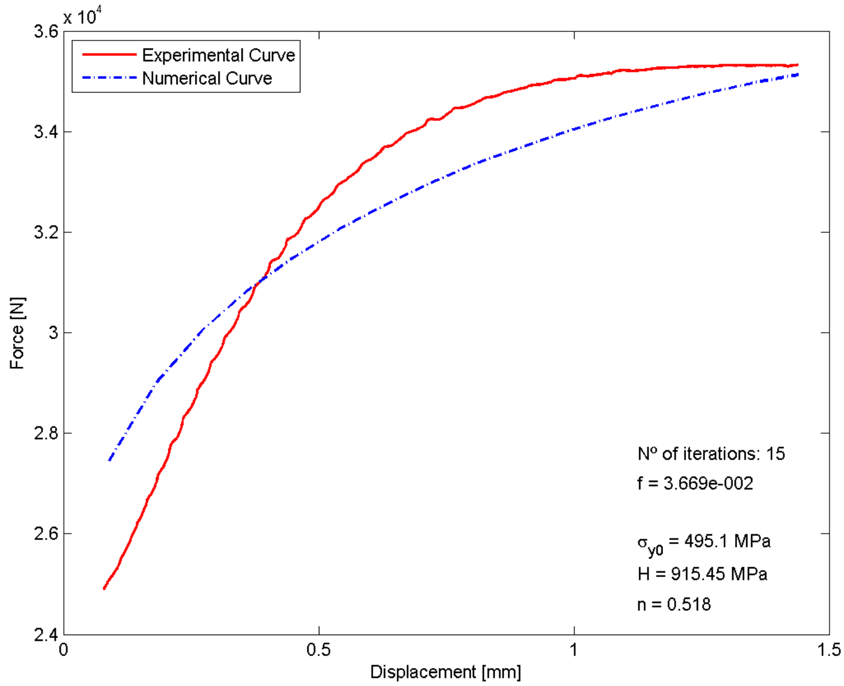


Figure 5 Comparison between numerical and experimental curves with n=0.5 for the annealed 4340 Steel

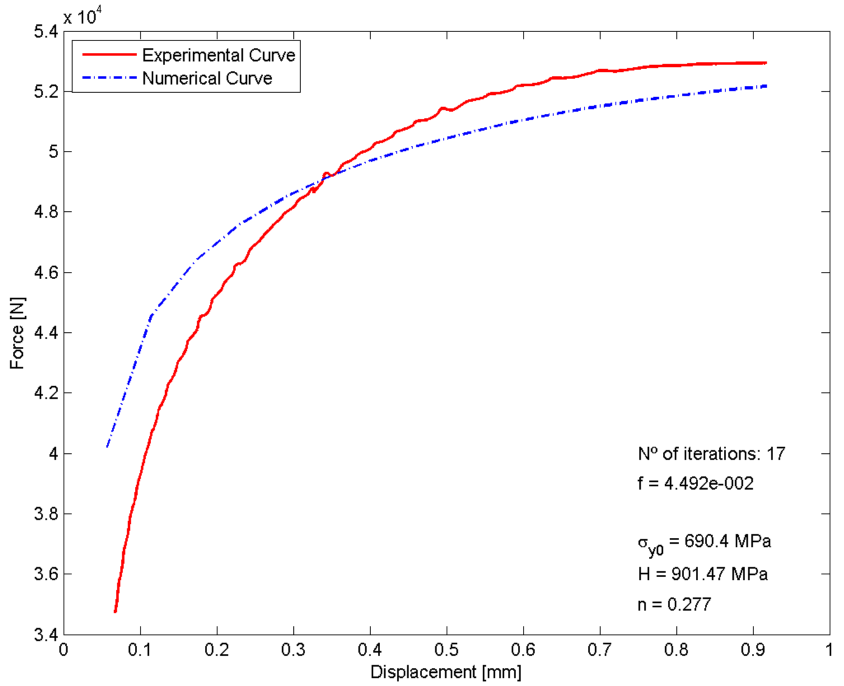


Figure 6 Comparison between numerical and experimental curves with n=0.1 for the normalized 4340 Steel

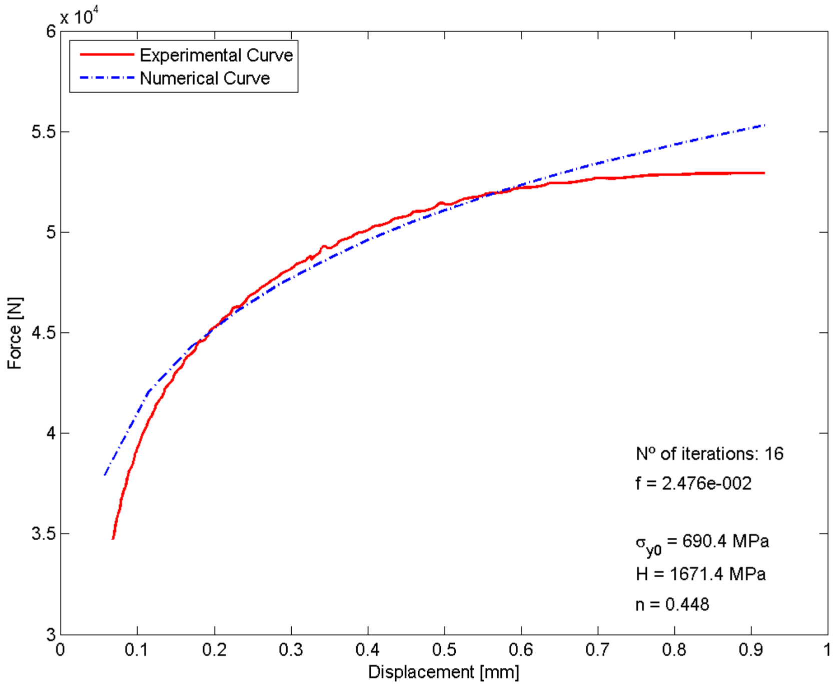


Figure 7 Comparison between numerical and experimental curves with *n*=0.5 for the normalized 4340 Steel

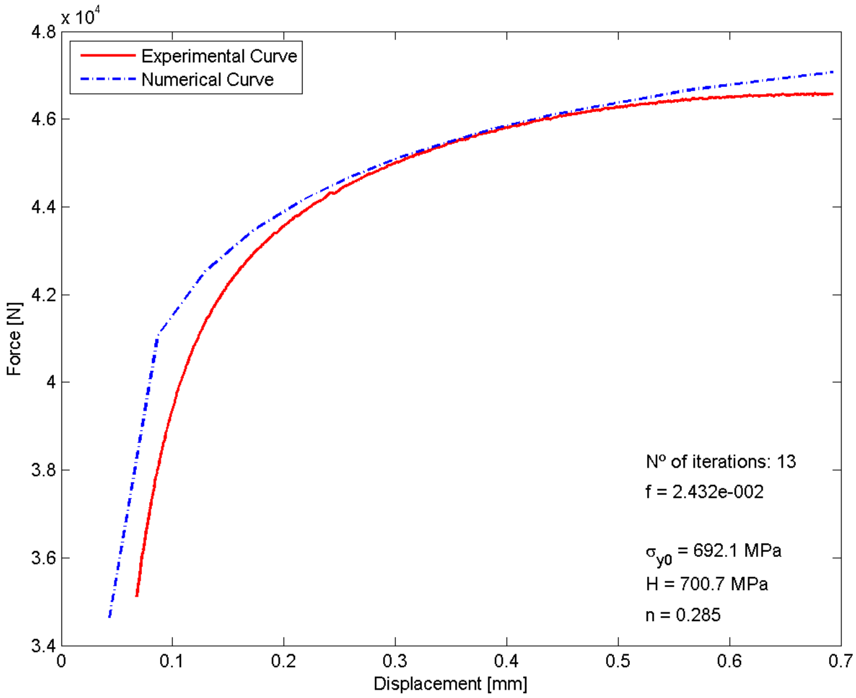


Figure 8 Comparison between numerical and experimental curves with *n*=0.1 for the received R4 Steel

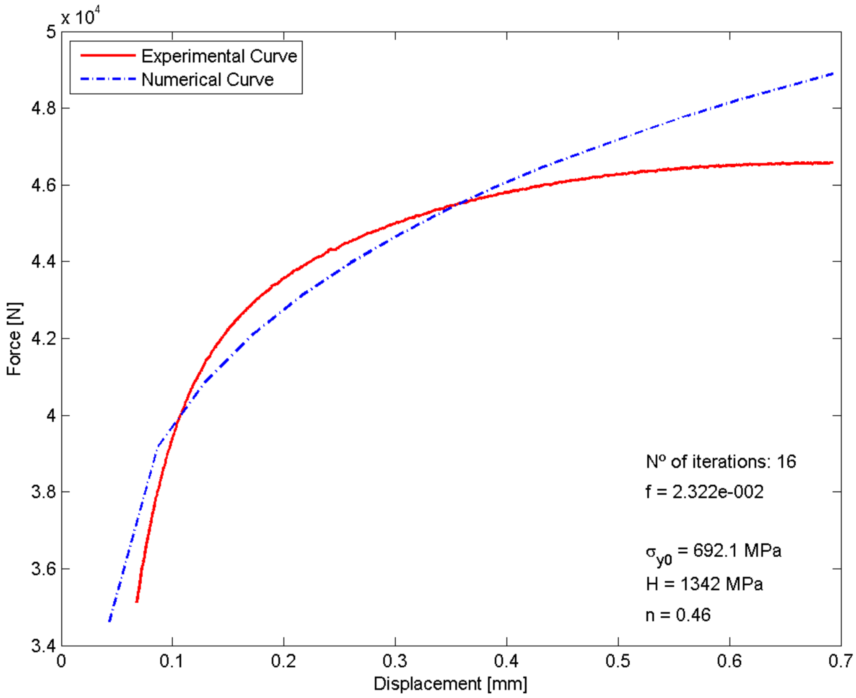


Figure 9 Comparison between numerical and experimental curves with *n*=0.5 for the received R4 Steel

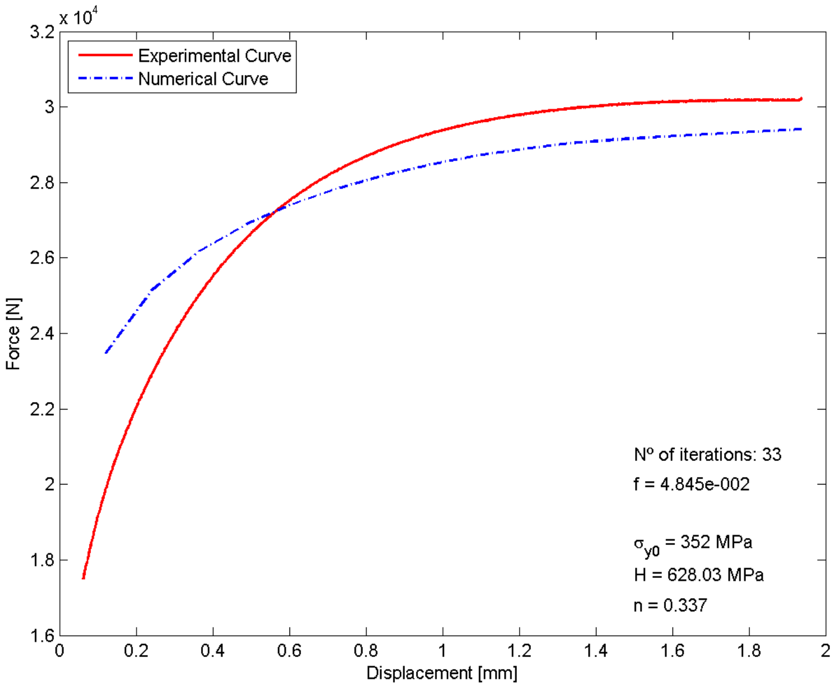


Figure 10 Comparison between numerical and experimental curves with *n*=0.1 for the received U2 Steel

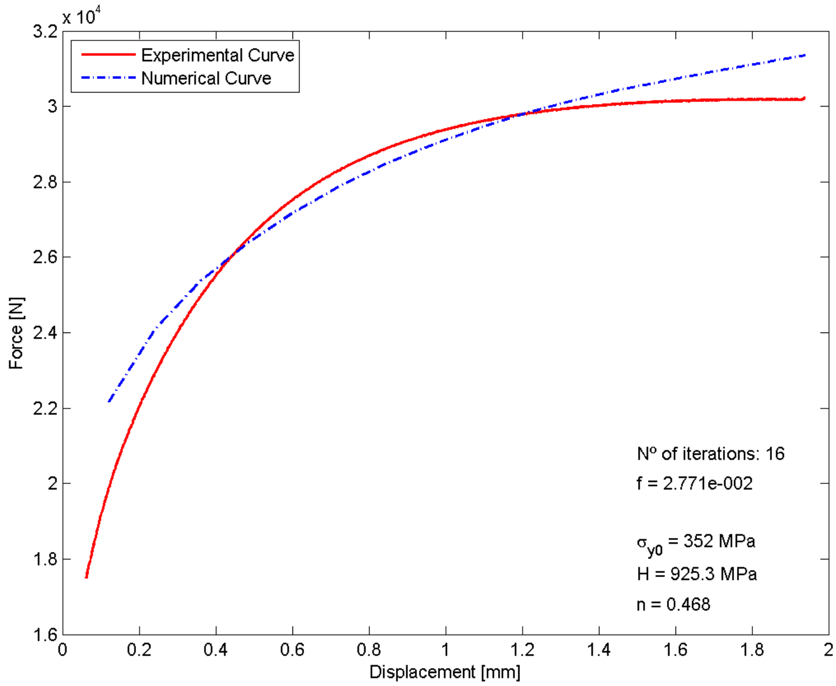


Figure 10 Comparison between numerical and experimental curves with *n*=0.5 for the received U2 Steel

Finally, Table 4 shows a relative comparison between the preliminary values of the parameters, obtained by the classical method, and the final parameters values with better convergence, obtained after optimization process.

Table 4 Comparison between *H* and *n* obtained by the *nlinfit* function and the optimization method.

Material	*H*	n
Annealed 4340 Steel	15.3 %	5.8 %
Normalized 4340 Steel	20.3 %	12.0 %
Received R4 Steel	14.7 %	16.4 %
Received U2 Steel	4.1 %	5.3 %

6 CONCLUSION

The optimization method by the Cauchy Algorithm presents interesting results for most of the analyzed materials. Although the process doesn't show a single solution, it is observed that when the initial vector guess was close to the optimal value, in particular when n was close to its true value, the curve fitting was reasonable, despite the curves shown in Fig.4 to 7 are in terms of force and displacement. Comparing the best results obtained for each material due to respect of those achieved by the *nlfintit* function, it is observed that both results are relatively close, as it can be seen in Table 4. However, the process using optimization methods is closer to reality, because as stated earlier, it simulates the physical problem in the numerical response.

The implemented methodology shows potential to be used in association with cheaper mechanical tests than tensile tests. Even using tensile tests to provide experimental data, the method only require one test, thus making the parameter identification by optimization quite cheap.

For future works, other gradient-based methods should be tested, such as the BFGS method, as well as to improve the algorithm using a more optimized finite element software such as ABAQUS ®, in addition with the implementation of a genetic method in association with the gradient-based one, to avoid local minimum points. Finally, other mechanical tests, such as hardness tests, should be used in the analysis for comparison purposes and to make the process even cheaper.

7 REFERENCES

Arruda, B. S *Implementação de método numérico multidimensional para determinação de parâmetros de encruamento de materiais dúcteis por ensaio de tração.* 2017. 84 p. Dissertation (Graduation in Mechanical Engineering)-Department of Mechanical Engineering, University of Brasília, Brasília, Federal District.

De Souza Neto, E. A, Peric, D; Owen, D. R. J. *Computer Methods For Plasticity.* Singapore: John Wiley & Sons, 2008, 791 p.

Kleinermann, J P.; Ponhot, J. P. Parameter identification and shape/process optimization in metal forming simulation. *Journal of Materials Processing Technology,* v 139, p. 521-526, Elsevier, 20 Aug 2003.

Kuna, M; Abendroth, M. Identification and validation of ductile damage parameters by small punch test. In *European Conference on Fracture,* 15, 2004, Stockholm CD ROM Proceedings Stockholm: European Structural Integrity Society, p 1-8.

Machado, L Q. *Determinação de parâmetros de encruamento de materiais dúcteis através de um ensaio de dureza.* 2016. 77 p. Dissertation (Graduation in Mechanical Engineering)-Department of Mechanical Engineering, University of Brasília, Brasília, Federal District.

Rao, S. S. *Engineering Optimization:* theory and practice. 4 ed. Hoboken, New Jersey, John Wiley & Sons, Inc, 2009

Stahlschmidt, J. *Estudos de identificação de parâmetros elasto-plásticos utilizando métodos de otimização.* 2010. 99 p. Mastership thesis (Mastership in Materials Science and Engineering)-Technology Science Center, Santa Catarina State University, Santa Catarina