TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics, Mechanics, and Engineering
Vol. 57, Issue III, September, 2014

NUMERIC SIMULATION OF MATERIAL BEHAVIOUR IN MANDREL BENDING

Cristel MUREŞAN, Gheorghe ACHIMAŞ

Abstract: Finite Element Method (FEM) is the most efficient method of numerical analysis in engineering. Simplicity of principles and basic concepts of the finite element method is a significant advantage for anybody using this method. FEM has continuously been developed as a numerical calculus method in a broad range of domains and applications of the deformable body mechanics using the displacement method, beginning with the need for instruments able to describe complex mechanical system, and currently reaching a level of development which allows for solving differential equations systems with complex derivatives.

Main advantages of FEM include: flexibility, possibility of modelling bodies which are physically not homogenous, as well as easy implementation in general calculus programs. The object of present paper is to extend the understanding of phenomena occurring in large diameter pipe bending using the mandrel pulling method.

Keywords: bends, connections, bending, numerical simulation, finite elements.

1. INTRODUCTION

Basis of FEM has been originally formulated in 1943 by the German mathematician Richard Courant (1888-1972), who combined Ritz method with numerical calculus in problems related to variational calculus and minimisation, to find satisfactory solutions for vibration systems analysis [4]. Beginning with the'70s, FEM has been used for solving most complex problems in continuous elastic structures, ranging from civil or industrial or water dams engineering to maritime ships and space ships. Concepts include certain hypotheses, general aspects, simplifications and approximations which, if ignored, would cause errors in finite elements modelling and analysis [6]. FORGE® application is a landmark in numerical simulation of metal deformation processes. FORGE enables the simulation of hot, semi-hot and cold forging of ferric and non-ferric metal. FORGE performs calculations for plants and multi-object structures. Numeric calculus may be performed both in 2D and 3D [2].

The application includes a module for viscous-plastic resolution for hot forming and forging, as well as an elasto-viscous-plastic module for cold deformation, studying the evolution of a range of physical parameters, such as the distribution of equivalent deformation for von Mises stress, temperature, pressure etc.

Reliability of automated discretisation and rediscretisation algorithms secure the application's stability both for simple geometric configuration and complex configurations. The application allows for using a very large number of finite elements which produces very accurate calculations that are the strong point of this numeric application, namely the reduction of calculation time [2].

2. STAGES OF PROBLEM SOLVING USING FEM

PRE-PROCESSING: is preparation of finite element model to be transferred to the solver [2].

PROCESSING: perform numeric calculation based on a predefined calculation model in FORGE® solver [2].

POST-PROCESSING: analysis of numeric calculation results, and their interpretation and processing and presenting relevant results in graphic form or images/maps presenting the distribution of parameters on certain sections [2].

2.1 Creating the geometric simulation model

Based on the concept formulated by Avitzur [1], who stated: “As regards the plastic deformation processes, models to provide an accurate description are not yet available, however approximations and simplifications of certain techniques may be applied”, we have reduced the base model due to its size and consequently the estimated completion time.

The modelling of pipe bending process requires the preparation of a model for the system being studied in order to simplify the problem in question. Such simplification regards the geometry of tools, geometry of heating system and of the product to be formed. Geometric model used the values described in [3].

Figure 1 presents the geometric model used for analysis, with D_e 108 x 3.6 mm given the tube size D_e 96.1 x 3.6 mm and D_i 89.00 mm, the other elements are designed such as to form an ensemble (Figure 2), and a second model based on design criteria in Figure 6.2, values of German manufacturer Schmidt & Clemens GmbH.

Figure 3 presents the geometric model used for analysis, with D_e 168.3 x 4.5 mm given the tube size D_n 114.3 x 4.5 mm and D_i 105.00 mm, the other elements are designed such as to form an ensemble.

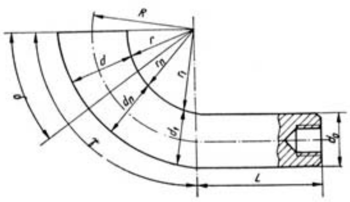

Fig. 1 Mandrel used for bending pipes by pulling [3]

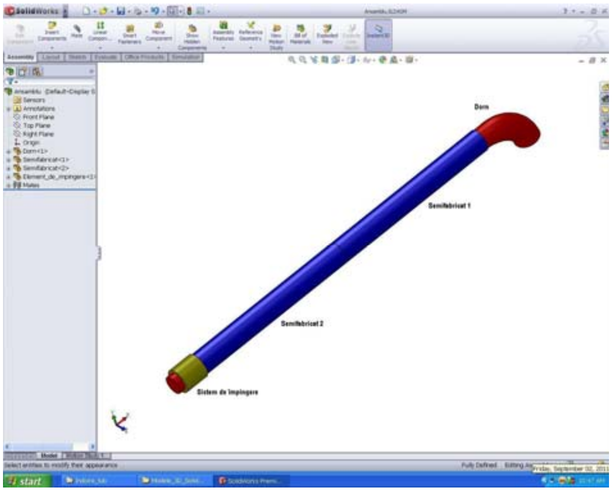

Fig. 2 Geometric model used in numeric analysis**.**

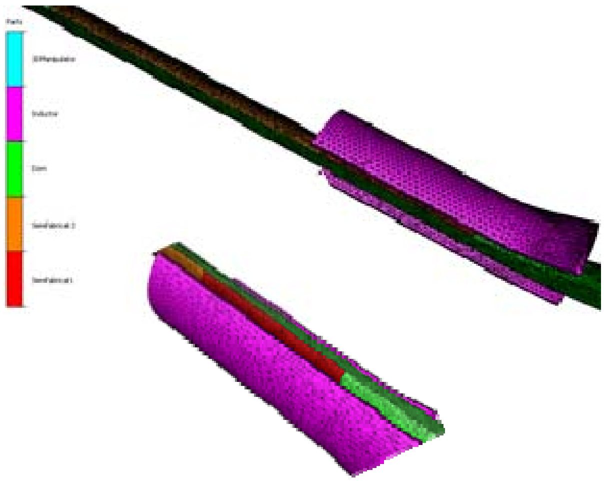

Fig. 3 Second geometric model used in numeric analysis**.**

For the discretisation of the ensemble under study, a tetrahedral mesh was used. For the first stage, a coarse tetrahedral mesh was used, in order to perform fast calculation. Under the action of deformation stress, the semisolid material is being deformed, as well as the discretisation mesh, since FORGE® approaches the forging process as a Lagrange process (i.e. mesh volume is deemed to be equal with the material volume). The Lagrange formulation considered the discretisation as being “material”, since the mesh follows the material during its deformation (flow). Consequently, the quality of elements located in the highly deformed areas may degrade quickly. When the mesh would become too deformed, a new discretisation will be performed (automated discretisation).

For the product with the length of 700 mm, the discretisation mesh comprised about 21176 elements and 6191 nodes. Considering the large number of finite elements in the analysed ensemble (Fig. 4), (the mandrel with an approximate length of 4000 mm has a very large discretisation mesh), generating an excessive duration of calculation, as well as the symmetry of the ensemble, the geometric model and the related discretisation mesh have been reduced (fig. 5).

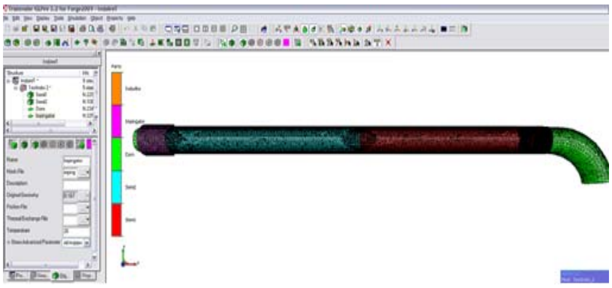

Fig. 4 Discretisation mesh for the mandrel – blank product ensemble**.**

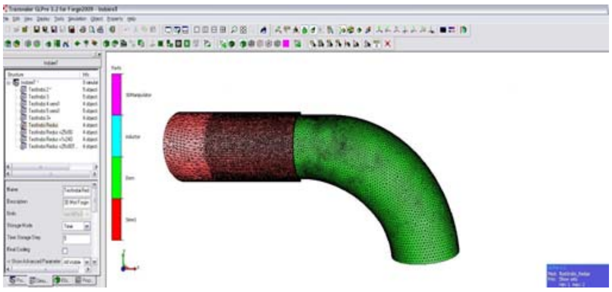

Fig. 5 Discretisation mesh for reduced ensemble of mandrel and blank product**.**

The reduced geometric model is presented in figure 5. In this case the mandrel has a length of about 1000 mm (the active part of the mandrel is unchanged), and the length of the blank product is of 175 mm. The remaining geometric characteristics (as per extended model) were not modified (inside and outside diameter of the blank). To simulate the induction heating process an element was created, named "inductor" which was given specific characteristics.

Data regarding the deformation equipment (hydraulic pushing press), deformation mould (mandrel) and blank product (pipe), as well as the conditions of the testing are fed into FORGE ® calculation code, following standard procedure. A time increment was imposed for the numeric analyses performed.

2.2 Inputting the limit conditions

2.2.1 Defining friction conditions

For defining friction conditions between surfaces in contact, Coulomb equations were used [2]:

$$\tau = \mu\sigma_n \quad \text{if} \quad \mu\sigma_n < \bar{m}\frac{\sigma_0}{\sqrt{3}} , \qquad (2.1)$$

$$\tau = \bar{m}\frac{\sigma_0}{\sqrt{3}} \quad \text{if} \quad \mu\sigma_n > \bar{m}\frac{\sigma_0}{\sqrt{3}} , \qquad (2.2)$$

where: μ is the friction coefficient; $\bar{m}$ is Tresca's friction factor.

2.2.2 Defining heat transfer conditions

During the plastic deformation process, the elements of the mechanical system work at high temperature, consequence of the external heat source – inductor. Therefore, thermal stress and deformation will occur in these elements' structure, besides the mechanical stress and deformation.

In the heat module, the energy dissipated by friction will be included in calculation, therefore the diffusion into the components shall be defined. The mechanical system is heated by an inductor, thus a heat exchanging interface was defined, as an element of induction heating. The value of heat flow was included, as well as the transfer coefficient between tool and material, and the temperature of the blank product. In the thermal-mechanical calculation, the heat transfer with the environment interface was taken into account. If initial exterior temperature is not constant, it can be calculated at a certain moment with the following equation [2]:

$$T = (\theta_o + 273) * exp^{\frac{\beta}{t}} \qquad (2.3)$$

where θ_0 constant; β time factor.

Thermal calculation is performed incrementally. Time step may be calculated automatically, according to variation of product's internal temperature. A maximum temperature increment may be defined, as well as a maximum time step. According to the product's internal temperature variation, the time step shall be adjusted in order to conform with the maximum temperature increment.

2.2.3 Selecting the material behaviour model

Avitzur [1] mentions that in regard to plastic deformation processes, the available material behaviour models may include approximations and simplifications of certain behaviours.

Metal behaviour during deformation is described by the fundamental equations of continuous media mechanics: mass conservation and displacement conservation. These are complemented by behaviour equations (rheological, tribological) and limit conditions.

Numeric calculations performed in FORGE®, had the rheological module based on a Hansel – Spittel evolution, defined by (2.4) [2]:

$$\sigma = A * e^{m_1 T} * T^{m_9} * \varepsilon^{m_2} * e^{m_4/\varepsilon} * (1+\varepsilon)^{m_5 T} * e^{m_7 \varepsilon} * \dot{\varepsilon}^{m_3} * \dot{\varepsilon}^{m_8 T} \quad (2.4)$$

where: A is solid consistency $[mm \cdot kg \cdot s]$; - ε - equivalent deformation (total deformation), - $\dot{\varepsilon}$ equivalent deformation velocity $[s^{-1}]$, - T - temperature $[^o]$, - $m_{1...9}$ sensitivity parameters.

Material consistency depends on thermal and mechanical conditions, which may vary. Equation applicable in such cases is: [2], [5]

$$A(T,\bar{\varepsilon}) = A_0 * (\bar{\varepsilon} + \bar{\varepsilon}_0)^n * e^{\frac{\beta}{T}} \quad (2.5)$$

where A_0 - constant; - $\bar{\varepsilon}_0$ - cold-hardening regularisation factor; - n - cold-hardening exponent; - β - temperature factor; - T temperature in Kelvin.

When such conditions are constant, the consistency may be defined with a constant value.

Variables used in numeric calculations were nodal variables or application's internal variables (e.g.: temperature, pressure, pressing velocity etc.).

2.2.4 Inputting process conditions

Actual process conditions were observed in the numeric analysis, parameters values are presented in Table 1.

Table 1.
Applied process parameters values

Pushing rate	2.5 mm/s
Mandrel's initial temperature	20 °C
Heating period	60 s; 90 s
Friction parameters	$\bar{m} = 0,15$ $\mu = 0,075$
Heat transfer parameters	Heat transfer coefficient = 2000 W/m^2K Diffusion coefficient = $11763.62 Wm^{-2}K^{-1}s^{1/2}$

3 SIMULATION RESULTS FOR HOT BENDING BY MANDREL PULLING

Numeric calculations based on a Norton-Hoff material behaviour model, with parameters presented in Table 2, allowed for studying the evolution of material flow in conditions imposed by the actual process. Values of these model parameters were automatically generated by the application.

Table 2
Model parameters used in numeric analysis

Solid consistency A [Mpa.s]	m_1	m_2	m_3	m_4
1499	-0.0027	-0.1265	-0.12651	-0.0596

3.1 Effect of mandrel temperature on material's bending behaviour

The effect of pre-heating temperature of the deformation moulds (mandrel, in this case) is very significant in regard to materials' deformation behaviour.

Two cases have been studied as regards the mandrel temperature in the beginning of deformation process, namely cold mandrel pulling (temperature of 20°C, ambient temperature) and pre-heated mandrel pulling (temperature of 600°C).

For the second particular case the initial temperature (deformation start) of 600°C in the pre-heated mandrel was the mandrel temperature after 60 seconds during the first particular case (can see in fig. 6, 7, or 8).

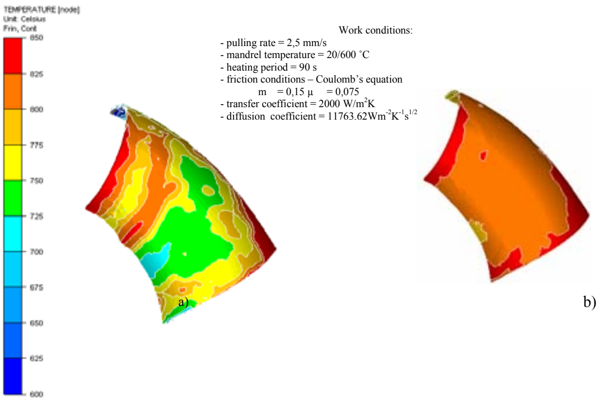

Fig. 6 Temperature gradient in processed blank after 90 s heating and pulling on mandrel with temperature of: a) 20°C; b) 600°C.

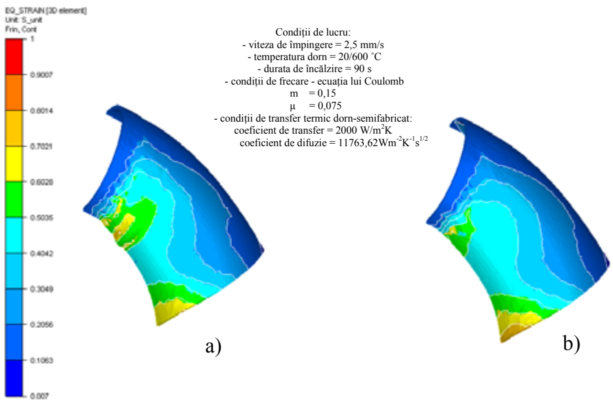

Fig. 7 Equivalent deformation distribution in the processed blank after 90s heating and pulling on mandrel with temperature of: a) 20°C; b) 600°C.

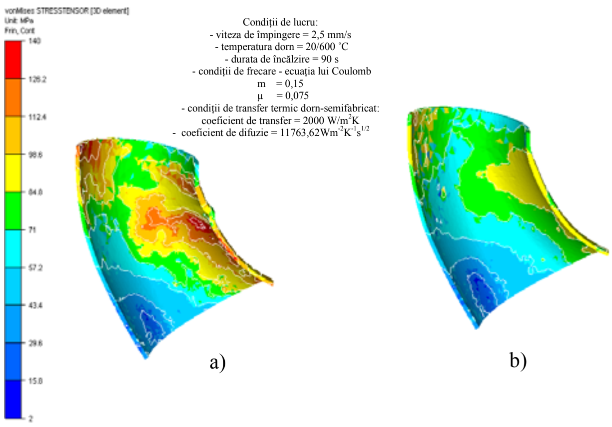

Fig. 8 Von Misses tension variation in processed blank after 90s pre-heating and pulling on mandrel with temperature of: a) 20°C; b) 600°C.

As regards the contact of blank product and mandrel surface, it can be noticed in Figure 9-b that when the mandrel is pre-heated to 600°C this will prevent the occurrence of material shaving from the mandrel.

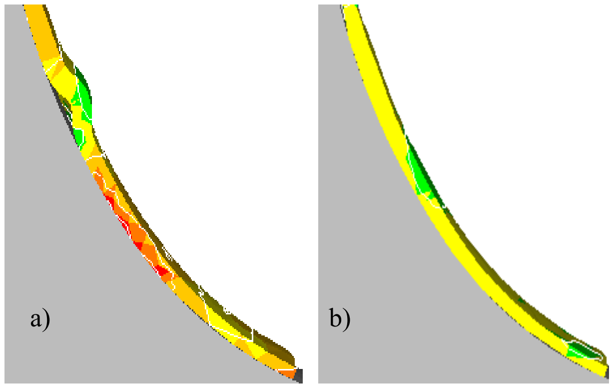

Fig. 9 Effect of mandrel temperature upon the occurrence of material shaving from the mandrel: a) 20°C; b) 600°C.

3.2 Effect of friction conditions at the contact area between mandrel and blank product

The effect of friction conditions between blank product and mandrel during the pulling process was studied by numeric analysis.

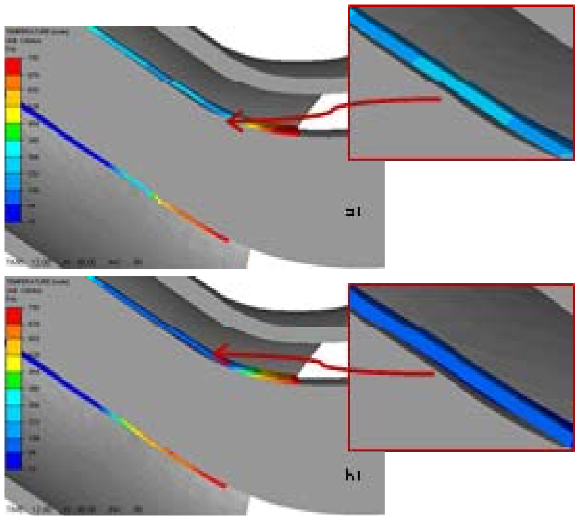

Fig. 10 Effect of friction between blank and mandrel upon material flow: a) μ=0,15; b) μ=0,30.

The geometry study reflects the experimental dimensions. Friction coefficient and Tresca's friction factor were given values specific for hot deformation. Studies were performed considering the mandrel as pre-heated to a temperature of 750°C, for which the inductor's heating temperature was of 850°C.

As presented in figure 10, a reduction of friction between the mandrel and blank product contact areas will slightly reduce the material shaving occurring just before it reaches the mandrel's interior side.

Experimental tests used as lubrication agent a graphite and water emulsion for which the numeric analysis considered a friction coefficient = 0,15. In order to assess the effect of lubrication conditions, tests were performed with lubricant of graphite, water and frit (glass), for which the friction coefficient used in numeric analysis was = 0,30.

4. CONCLUSION

Numeric analysis of material deformation behaviour in mandrel pulling was performed with FORGE® application.

The geometric model created and imported in the simulation application complied with the dimensions of the experimental deformation assembly/device. Furthermore, the assessment and quantification of experimental process conditions which could not be measured (friction coefficient, heat transfer coefficient) was applied. Using the actual experiment conditions in the numeric analysis enables the comparison of results. A Norton-Hoff mathematical model was selected for studying the material bending behaviour with mandrel pulling.

ACKNOWLEDGMENT:

This paper is supported by the Sectorial Operational Programme Human Resources Development POSDRU/159/1.5/S/137516 financed from the European Social Fund and by the Romanian Government.

REFERENCES

[1] Avitzur T. Handbook of metal-forming processes, John Wiley & Sons , Inc., NY.1983

[2] Forge (R) v.2005, Genaral Data File, Transvalor S.A.

[3] Grecu, H.: Tehnologia îndoirii țevilor şi profilurilor. Editura Tehnică, Bucureşti, 1977.

[4] Maksay Şt. I., Bistrian D. A.: Introducere în metoda elementelor finite, Editura Cerni, Iaşi, 2008.

[5] Neag, A.V.: Cercetări privind obținerea şi comportarea la deformare a aliajelor tixotrope de aluminiu, Teza de Doctorat, 2008.

[6] Practica modelării cu elemente finite, Curs, Departamentul de Rezistenta Materialelor, Universitatea Politehnică Bucureşti, 2000.

SIMULAREA NUMERICĂ A COMPORTĂRII MATERIALULUI LA ÎNDOIREA PE DORN

Rezumat: Metoda elementului finit (MEF) este cea mai eficientă metodă de analiză numerică în inginerie. Simplitatea principiilor şi conceptelor de bază ale metodei elementelor finite creează un important avantaj pentru cei care folosesc această metodă. MEF s-a dezvoltat continuu ca metodă de calcul numeric într-o multitudine de domenii şi aplicații din mecanica corpului deformabil prin metoda deplasărilor, demarând cu necesitatea obținerii unor instrumente care să permită prin calcul riguros caracterizării unor sisteme mecanice complexe, ajungând astăzi la o dezvoltare care permite rezolvarea unor sisteme de ecuații diferențiale cu derivate complexe. Printre principalele avantaje ale MEF se numără: flexibilitatea, posibilitatea de modelare a unor corpuri care nu sunt omogene punct de vedere fizic, precum şi implementarea facilă în programe generale de calcul. Obiectivul articolului de fa□ă este de a extinde în□elegerea fenomenelor care au loc în țevi de diametru mare la îndoire, folosind dornul.

Cristel MUREŞAN, Phd. Student Eng., Technical University of Cluj-Napoca, Department of Manufacturing Engineering, Muncii Boulevard 103-105, Cluj-Napoca, ROMANIA, e-mail: cristelmuresan@yahoo.com; Gherla 405300, Str. Mihai Viteazu no. 25, Județ Cluj, 0264 243129, 0744 604794.

Gheorghe ACHIMAŞ, Prof. Dr. Eng., Technical University of Cluj-Napoca, Department of Manufacturing Engineering, Muncii Boulevard 103-105, Cluj-Napoca, ROMANIA, e-mail: Gheorghe.Achimas@tcm.utcluj.ro; Cluj-Napoca 400537, St. Clăbucet no. 1/38, Județ Cluj 0720 054863.