NUMERICAL MODELLING OF THE PROCESS OF COLD PLASTIC DEFORMATION WITH TOOL RACK OF THE PROFILES

Mădălin TUDOR, Ion UNGUREANU, Eduard NITU, Monica IORDACHE, Doina IACOMI
[1]University of Pitesti, Romania,

***Abstract*:** The main purpose of this article is to present the results of researches made to obtain a numerical model of the process of cold plastic deformation with tool rack, with the help of the numerical simulation program ABAQUS, to achieve a metric profile and validate through experimental dat. Since the experimental part is in progress, the comparison between the forces and the geometry of the profiles was done with the help of literary data.

Keywords: model, finite-element modelling, cold plastic deformation, rack.

INTRODUCTION

The achievement of a model for the process of volumetric cold plastic deformation with tool rack, its analysis using the method of finite element and the presentation of the results of the research, represent the aim of this article. The modelled rolling process with tool rack corresponds to the one realized on the industrial machines used to process profiles by cold deformation because the experimental stand is realized on such a machine.

The running profile is a metric one, M20STAS 6371 – 73, figure 1 with the characteristics from the table 1.

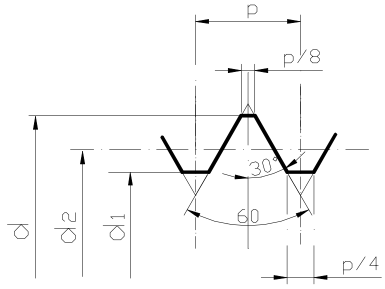

Fig.1 The form of the metric profile

Table 1 The dimensions of the metric profile

Symbol of the profile	p, [mm]	d, [mm]	Deviations, [mm]		d_2,[mm]	Deviations, [mm]		d_1, [mm]	Deviations, [mm]	
			e_s	e_i		e_s	e_i		e_s	e_i
M20x2– 6h	2	20	0	-0.28	18.701	0	-0.16	17.835	0	-0.2 89

Scheme of generating the profile is the one from figure 2.

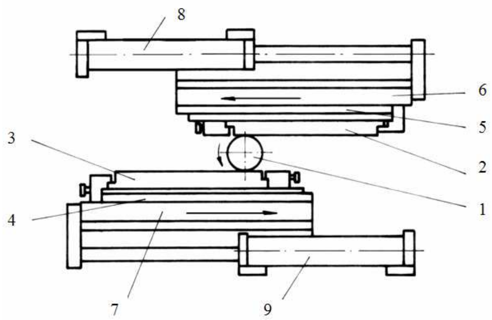

Fig. 2 Kinematical scheme of running the tool rack

The 5 circular channels generate the piece 1 which is set and fixed between the tops and trained in the rotation n_p by racks 2 and 3 which are moving simultaneously with the speeds v tangential to the piece. The two racks are fixed on the sledges 4 and 5 which are moving on the guides 6 and 7. The moving of the sledges is realized by the linear hydraulic engines 8 and 9 simultaneously to the introduction of the oil pressure with the help of a distributor. To this movement the profile on the rack penetrates the piece imprinting on it.

So, generating the profile is realized using the two notions:

- displacing the racks with the velocity v
- rotating the piece with the velocity $n_{p.}$

The rotation of the piece is made by the racks that make the part move; it is not realized by the machine tool. This is determined by the geometry of the piece and the speed v.

The profile is realized gradually by the penetration of the racks into the piece as a result of the fact that the profiles of the racks are inclined under angle α from their direction of movement.

We can conclude that to generate a profile on a piece we need the following:

- a tangential movement to the piece realized with the velocity v
- a movement of radial penetration into the piece made by the racks; this is given by the size of the slope angle α; the size of α determines a second velocity V_r which penetrates radially the piece.

The process of strain to realize a profile is described in figure 3.

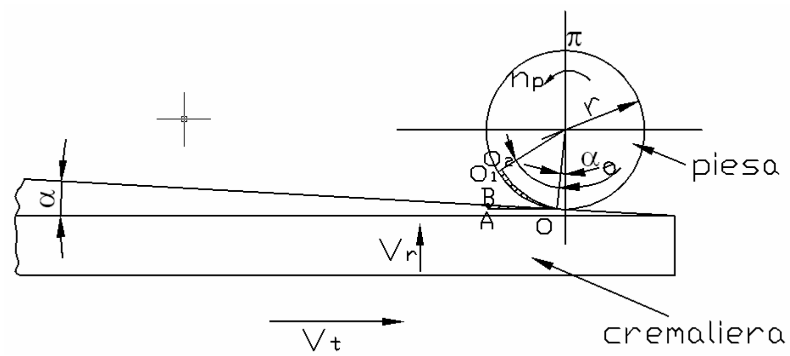

Fig. 3 Slope strain of a rack

By motion the racks tangentially to the piece the first contact appears in point O. Continuing the movement of the rack, it rotates the piece with the same speed. Considering that at contact in point O, time t=0 to time $t=t_1$ point O_1 overlaps with A and O_2 with B. The penetration of the piece is produced by a spiral with slope α.

The parameters of process are the tangential speed V_t and the slope angle α. The radial speed is determined by the two parameters. The affirmation is sustained by the fact that the volume of deformed material in time t_1 corresponds to section OAB; but the section depends on the two parameters: V_t and α.

The analysis corresponds to the rotation of the piece in angle 0-π because the surface of the piece in this angle is circular and two racks are used. For the rotation of the piece over the angle π the contact piece-rack is made on the spiral realized by the second rack. Given the fact that the slope angle is very small the observations can be considered valid for the next rotations of the piece as well.

THE DEFINITION OF THE ELEMENTS OF THE MODEL AND THEIR ASSEMBLY

In the simulation process, the input data are the following:

- shape and dimensions of the semi-product
- shape and dimensions of the tools
- behaviour law of the material
- friction coefficients
- parameters of the processing system (the moțion of the tools, their speed and the calibration time).

The semi-product used for the simulation, figure 4, has a simple cylindrical form, with the diameter defined function of the law of constant volume, d_0 = 18.8 [mm] and a length of 22 mm.

On account of the dimensional parameters of the profile the tools rack were simplified, figure 5, using the soft of design CATIA for the development of the numerical model. This software allows the saving of the files with extension "igs" which can be imported by the soft of numerical simulation ABAQUS.

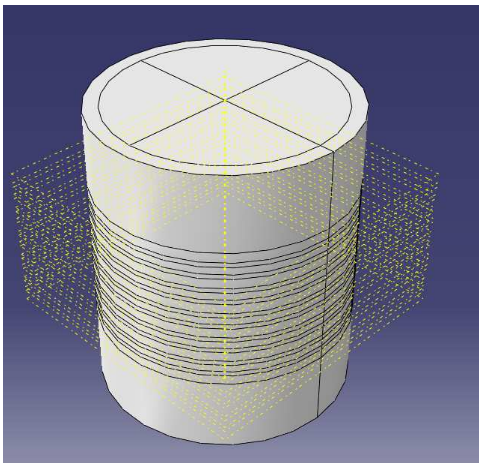

Fig. 4 Semi-product

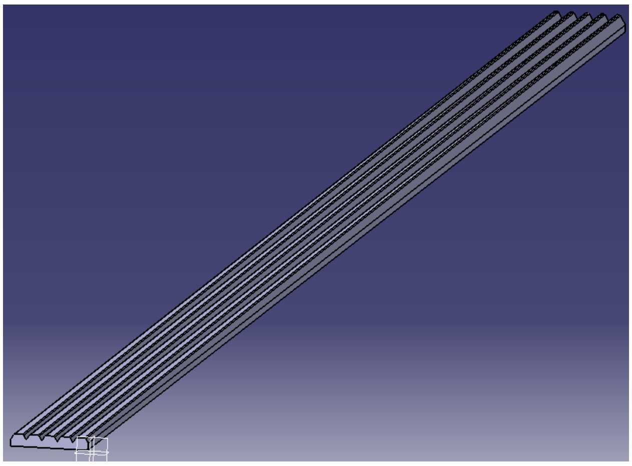

Fig. 5 Tool rack

In order to prepare the model for the analysis the following were performed, including the seven stages of the **pre-processing** of the ABAQUS program.

In the first stage, "*Part*", to drew the semi-product at real dimensions, defined it as a deformable body and imported the tools as rigid element.

In the next stage, "*Property*", there was inserted data about the mechanical properties of the material of the semi-product meant to be deformed as follows:

- into the elastic domain through Young's module E = 210GPa and Poissons's coefficient v=0,3
- into the plastic domain of behaviour through Voce's law [7].

The coefficients of this law were set experimentally using torsional mechanical testing.

In the stage "*Assembly*", figure 6, the assembly of the elements of the model was realized: the two racks were positioned in contact with the semi-product, their position being symmetrical to the axis of the semi-product and opposite to each other.

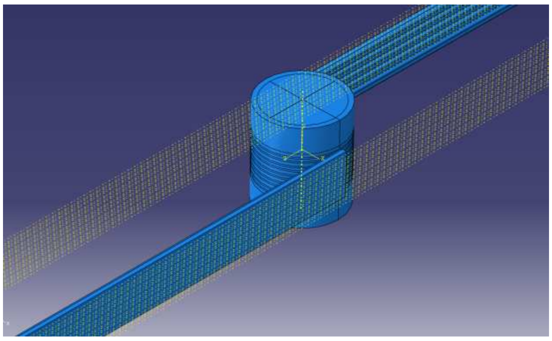

Fig. 6 Numerical module

In the stage "*Step*", figure 7, the steps required for the analysis of the model are defined. In this stage we established the output parameters (force, stresses, strains, etc) for each step using the option "Create field output". Also in this stage was assigned points to the tools rack in order to put conditions into the next stages. The output parameters are: force, strains and stresses.

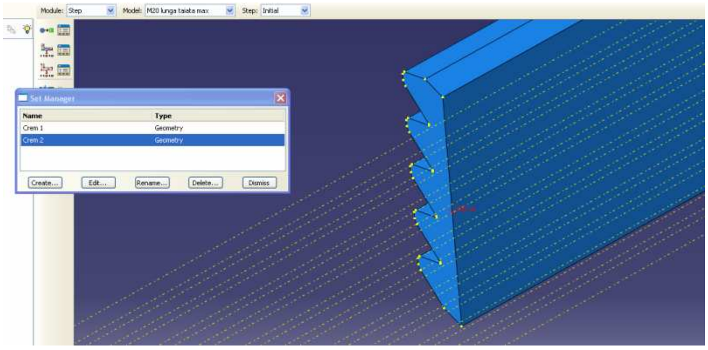

Fig. 7 Assigning the reference points

In the stage "*Interaction*", figure 8, we defined the type of contact between the surfaces. The contact between the surfaces is "surface to surface", the deformed surface of the semi-product and the surface of the tools that interact with the semi-product, the friction coefficient between the tools and the semi-product is 0.3.

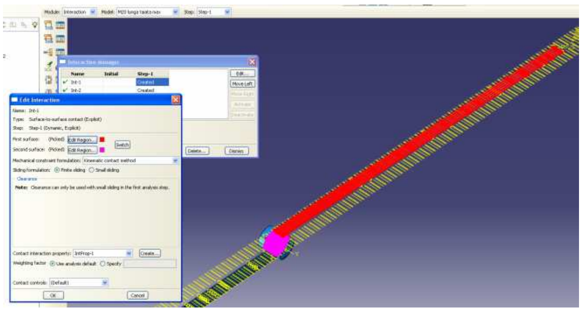

Fig. "Surface to surface" type of contact

In the stage "*Load"*, figure 9, or established the loadings and the limit conditions for each step:

- the tools rack have one degree of freedom: the displacement on the 3^{rd} axis
- the semi-product is free between the two racks.

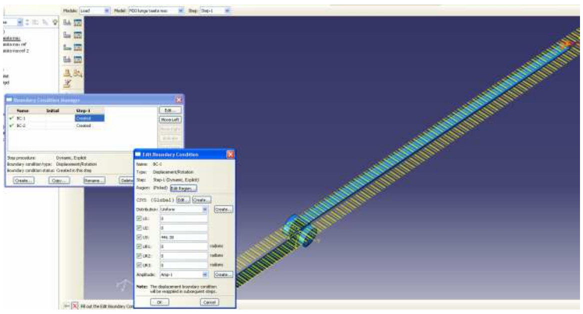

Fig. 9 Limit conditions

In the stage "Mesh" was realized the discretization of the semi-product which influences the results of the simulation if the elements are too big and the strains are not correctly calculated, thus, making the strains and stresses have the same values on a wide surface (the one of the element mesh); while in practice the strains and stresses are different on the same surface. The discretization of the semi-product has to be developed in a reasonable period of time and the number of elements has to be big enough to reproduce the real state of stresses and strains of the part. In order to obtain some satisfactory results the semi-product was partitioned in specific zones depending on its degree of strain, as follows:

- on the axial direction four zones were realized, figure 10:
    - area A, with very small strains where the size of the elements can be big

- zone B, corresponding to the top of the profile, with low atrains where the size of the elements is medium
- zone C, corresponds to the flanks of the profile, very strong strained where the size of the elements must be as small as possible
- zone D, corresponding to the gap of the profile, very strong strained where the size of the elements must be as small as possible

- on the radial direction two zones were realized, figure 11:
  - zone E, associated to the superficial strained layer where the size of the element must be small
  - zone F, corresponding to the middle, with very small strains where the size of the element can be bigger.

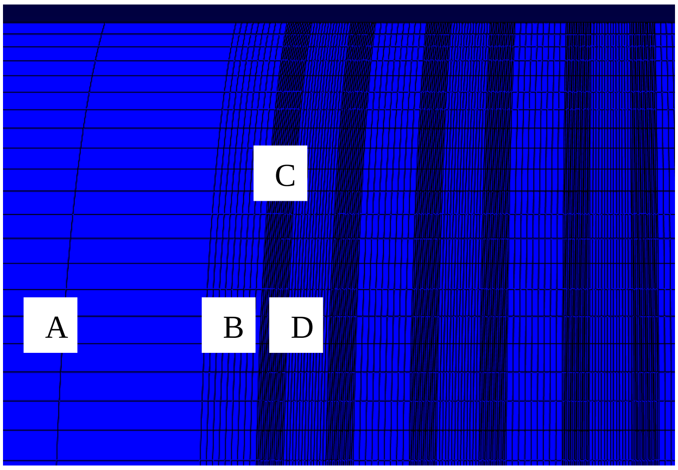

Fig. 10 Discretization of the semi-product on axial direction

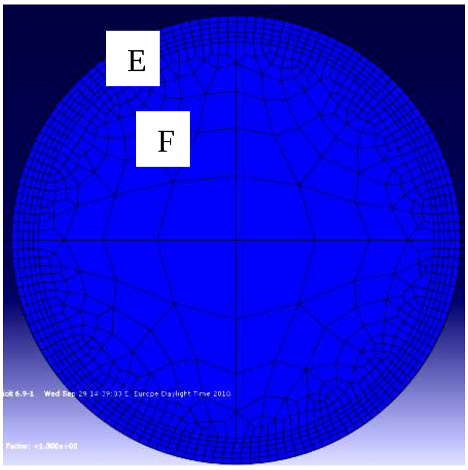

Fig. 11 Discretization of the semi-product on radial direction

The second phase of **processing,** the model is analysed and its aim is running the program, figure 12. This is the longest phase depending on the complexity of the program and the resources of the computer on which the simulation is done.

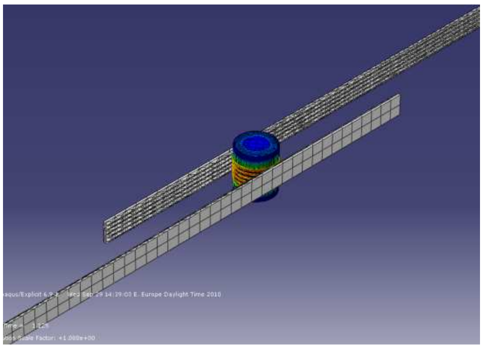

Fig.12 Running the program

The third phase is **post-processing,** the results and charts obtained after the simulation are extracted.

THE VALIDATION OF THE MODEL

The validation of the model is realized by comparing the geometry of the profiles, the remanent streses - figure 13 and the forces – figure 14, obtained through model simulation, to the data obtained experimentally. The values of the obtained forces are the following:

-for the tangential force 6 KN
-for the radial force 20 KN.

The radial force is much bigger than the tangential force, and the entire value of the force for a single rack can be calculated with the Pythagorean Theorem.

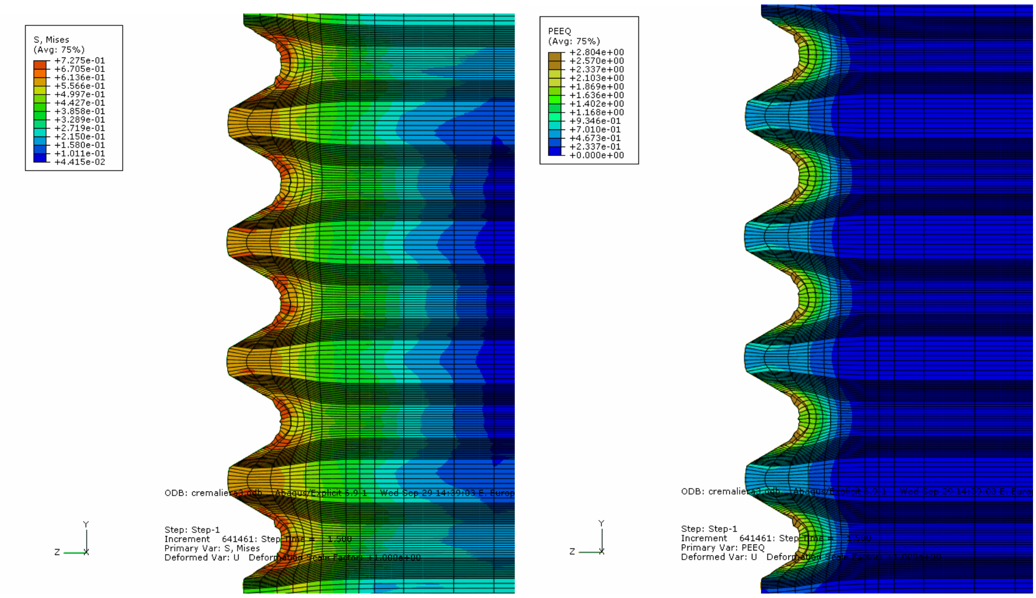

Fig.13 Streses and strains obtained through simulation

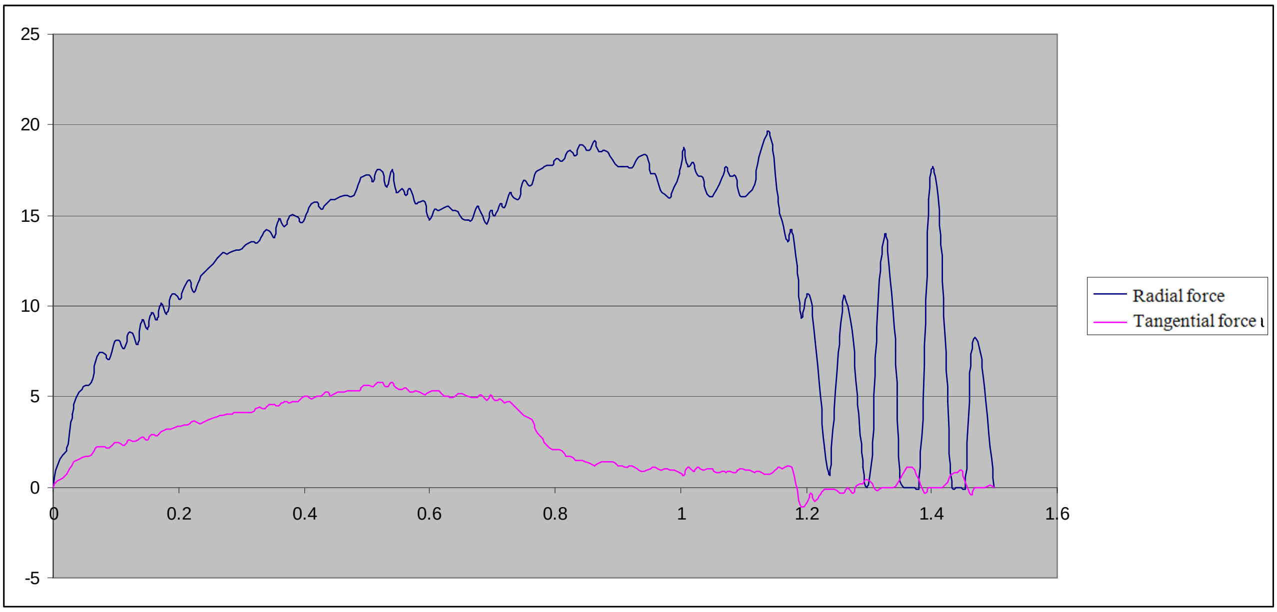

Fig.14 Presentation of the forces obtained through simulation

An experimental stand on a special machine of running, ROTO-FLO model 3235, was conceived and realized for these experiments. The fundamental scheme of this stand is represented in figure 15.

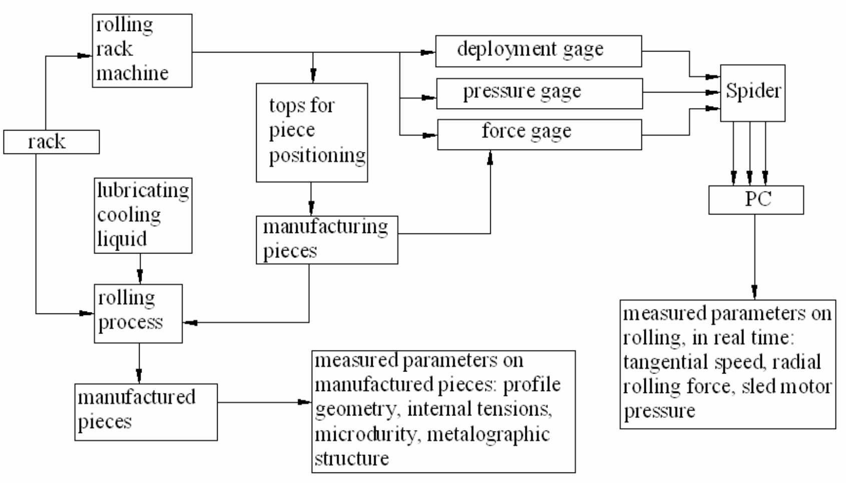

Fig. 15 Stand of experiments

Because the experimental part is in progress the validation of the model is partial. For the validation the values of the forces obtained through simulation were compared with those from literature which can be found in the mentioned interval [6].

CONCLUSIONS

With the help of numerical simulation results can be obtained from the phase of design and we can interfere on the model to establish properly the parameters of the process so the desired objectives can be fulfilled.

Through this simulation we established the forces necessary to the strain and the achievement of five circular grooves, the remanent streses of the deformed layer and the profile geometry function of the output parameters and the characteristics of the material.

For the moment the validation of the model is partial and was based on comparing the sizes of the forces obtained through simulation with those from literature. The final validation of the model will be made after realizing all the experiments.

ACKNOWLEDGEMENTS

This article was produced under the project "Supporting young Ph.D students with frequency by providing doctoral fellowships", co-financed from the EUROPEAN SOCIAL FUND through the Sectoral Operational Program Development of Human Resources. This work was supported by CNCSIS – UEFISCSU, project number PN II – IDEI 711 – 2008, "Analytical and numerical modelling of the processes of cold plastic processing of complex profiles”. The numerical simulation were made in laboratory of University of Metz.

REFERENCES

[1] Amir A. Kamouneh, *Diagnosis of involutometric issues in flat rolling of external helical gears through the use of finite-element models*, The University of Michigan, 2006.
[2] Craveur, J. C. *Modélisation des structures. Calcul par éléments finis*. Editura Masson, Paris, 1996
[3] Daridon, L., *Curs de element finit.*Universitatea din Strasbourg, 2000
[4] Gârbea, D. *Analiză cu elemente finite*. EdituraTehnică, Bucureşti, 1990
[5] Menezes, L. F., Teodosiu, C. *Three-dimensional numerical simulation of the deep-drawing process using solid finite elements*. Journal of Materials Processing Technology, Vol. 97, 2000, pp. 100-106
[6] Neagu C., Vlase A., Marinescu N.I., *Presarea volumică la rece a pieselor cu filet şi dantură*, Editura Didactică şi Pedagogică Bucureşti, 1994.
[7] Niţu E., Ungureanu I., Iacomi D., *Modelarea analitică şi numerică a proceselor de prelucrare prin deformare plastică volumică la rece a profilelor complexe*, Proiect de Cercetare Exploratorie, Cod proiect ID_711/2008, Universitatea din Piteşti, 2009.