UNIVERSITY OF PITESTI
FACULTY OF MECHANICS AND TECHNOLOGY
SCIENTIFIC BULLETIN
AUTOMOTIVE series, year XVI, no.20 (1)

NUMERICAL SIMULATION USING THE FINIT ELEMENT METHOD OF THE COLD PLASTIC DEFORMATION PROCESS WITH PLANETARY ROLLERS TO ACHIEVE PROFILES

Gina BOICEA[1], Ion UNGUREANU[1], Eduard NIȚU[1], Monica IORDACHE[1].
[1]University of Piteşti, Romania,

Abstract: *The main purpose of this article consists in presenting the results of researches made to achieve a type of numerical simulation of the cold plastic deformation process with planetary rollers and its analysis through the finite element method.*

Keywords: Finite-element modeling, intermittent rolling, cold plastic deformation, simulation.

INTRODUCTION

The rolling process through cold plastic deformation with planetary rollers in order to obtain profiles is made on special machines through successive incremental strain on the channels to be achieved.

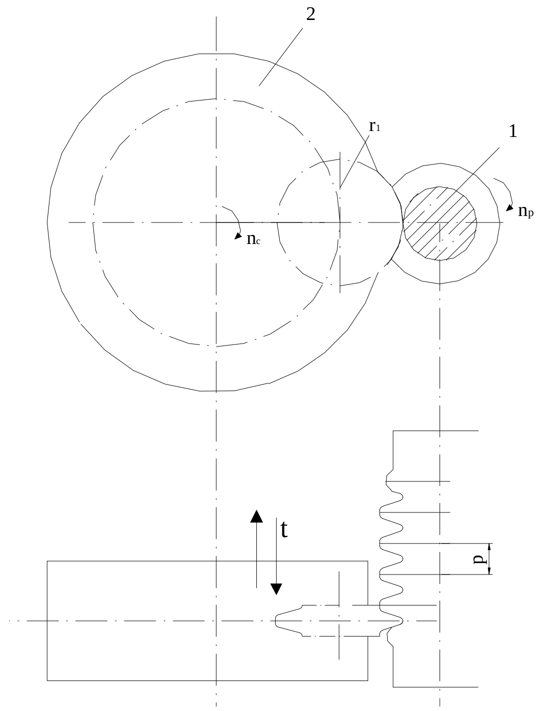


Fig.1

Thus, in order to obtain 5 profiles (annular cavities) with a rolling head 2 that has only one planetary roller r1 on a cylindrical piece 1, fig.1, we need:
- the rotational motion of the head 2, n_c ;
- the circular feed motion of the part (in this case) given by the rotation n_p ;
- the alternating indexing motion, t.

The alternating indexing motion allows for the strain to take place incrementally. For example, if we used only one roller to achieve the first channel on the entire circumference of the part, in order to achieve the second channel we would need the indexing motion of the part which requires special machines and devices.

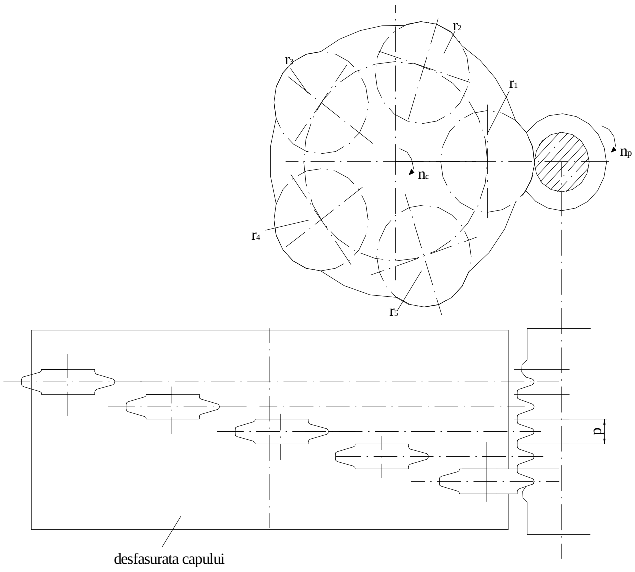


Fig.2

By assembling on the rolling head five rollers (r_1, r_2,...r_5) axially offset with a p spacing in between and placed at equal angles (72 □), fig. 2, the indexing motion is substituted. In this case in order to generate the 5 channels we only need the rotation motions and not the indexing motion.

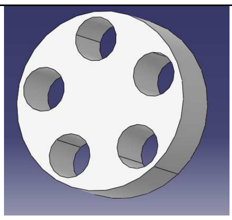
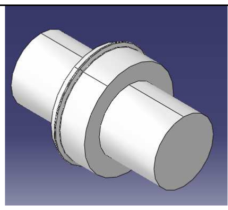
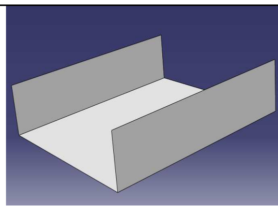
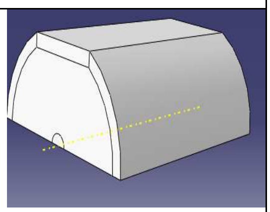
DESIGNING THE ELEMENTS OF THE MODEL AND THEIR ASSEMBLAGE

In order to obtain the numerical model was took into consideration the diagram in fig. 2 supposing that the semi-product accomplishes only the longitudinal feed motion.

Starting from the dimensional parameters of the surfaces to be obtained for profile M20, table 1, the component elements of the model are synthesised in table 2. In order to design them was used the Design module of the designing program CATIA and its compatibility with the numerical simulation program ABAQUS allowed the exchange of the numerical model into the data base of the Abaqus program.

Tab.1 Dimensional parameters of the surfaces

Profile symbol	**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
M 20 x 2 – 6h		20	0	-0.28	18.701	0	-0.16	17.835	0	-0.289

Tab.2 Elements of the model			
Rolling head	Planetary rollers	Holding prism of the semi-product	Semi-product
			

In order to obtain the numerical simulation model of the cold plastic deformation process with planetary rollers he following steps were performed: pre-processing, processing and post-processing.

During the first stage, the pre-processing, the model was prepared for the analysis. In this respect were went followed the modulus of the ABAQUS program:

- in the first module, *"Part"*, was defined the semi-product as a deformable body, and was tool design. Since it did not influence the results of the simulation and its production time was reduced, has chosen to represent only half of the semi-product.

- next, in the *"Property"* module were introduced the properties of the semi-product's material to be plastically deformed through the planetary rollers process.

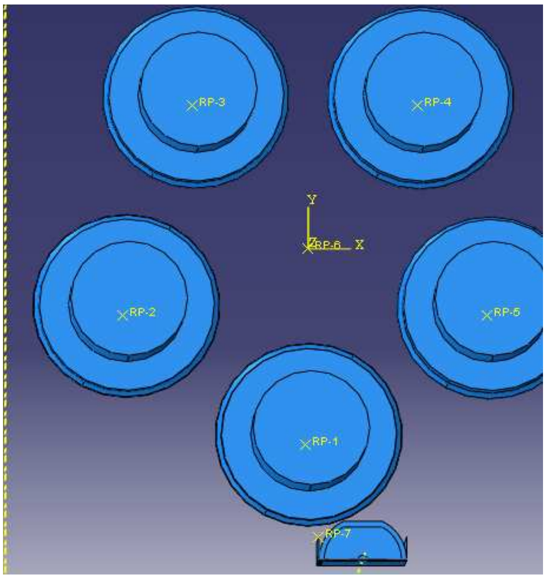


Fig. 3 Numerical model

- in the *"Assembly"* module were assembled the elements of the model, fig. 1.3. Also in this module defined reference points to the planetary rollers, the centre of the rollers and the prism, where further on applied the limit conditions and displacements.

- in the *"Step"* module were set the steps needed to analyse the model. Thus, the case of the cold plastic deformation process with planetary rollers required two steps. During the first step that took 0,3ms, the rollers had to reach a angular velocity and the semi-product had to reach the working position. During the second step the strain took place. Also in this modulus were established the output parameters (force, stresses, strains etc) every time using the option "Create field output".

- in the *"Interaction"* module was defined the type of contact between the surfaces. The contact between surfaces during cold plastic deformation with planetary rollers is a "surface-to-surface" type of contact. The option "surface-to-surface" describes the contact between a deformable surface and a rigid one of the tool.

- in the *"Load"* module were set the tests and limit conditions for every step. The boundary conditions and the tests were applied only for the rigid bodies with respect to their corresponding reference points, table 3.

Tab. 3

Tests	Reference points
Free rotation of the rollers around their own axes	RP1, RP2, RP3, RP4, RP5
Rotation of the rolling head, n=950 rot/min	RP6
Displacement of the semi-product with 5 mm	RP 7

- in the *"Mesh"* module generated the numerical model of the semi-product, a fact which influenced the results of the simulation; if the elements are very big, the forces, stresses and strains are not correctly calculated. The finite element used for the mesh was a C3D8R, hexagonal, 8 nodes, reduced integration type of element, fig.1.4. In order to obtain reliable results in an acceptable period of time, we resorted to partitioning the semi-product in specific areas function of its strain level [7].

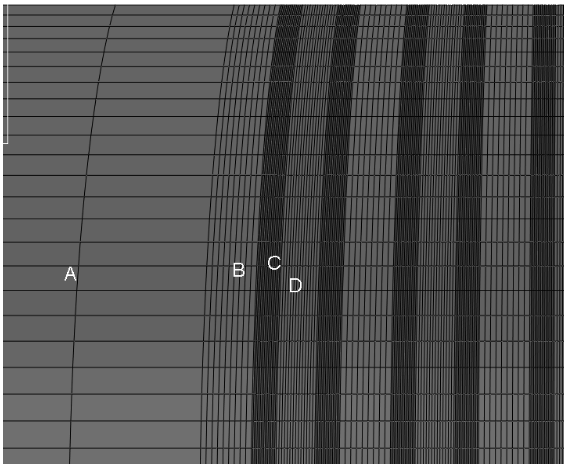


Fig.4

Thus, the semi-product was partitioned in several areas:

1- on an axial direction were achieved three areas, fig. 4.

- area A where the size of the elements is bigger because this area has very small strains;
- area B corresponds to the tip of the profile.
- area C is the one corresponding to the joining radii of the rollers;
- area D is a highly deformed area corresponding to the gap of the profile; [8]

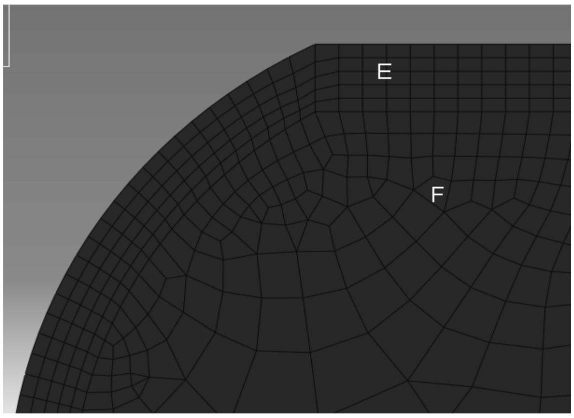


Fig.5

2- on a radial direction were achieved two areas, fig.5:

- area E (the core area) where the size of the element is smaller;
- area F (the superficial area) where the size of the element is higher. [8]

The total number of elements was 261006.

The second stage, the processing of the numerical model, simulation runtime. It is the stage that takes the longest period of time function of the program complexity.

In the third stage, the pre-processing of the model, the results obtained (for example, the variation components of the rolling force and of the total force function of the time, stress distribution etc) are presented.

MODEL VALIDATION

The validation of the numerical model is obtain by comparing the results obtained through numerical simulation and those obtained experimentally. Thus, the parameters taken into consideration are: forces (fig.1.6), micro-hardness, residual stresses and profile geometry (fig.7).

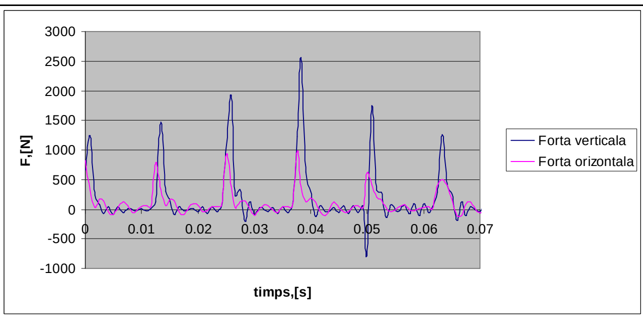
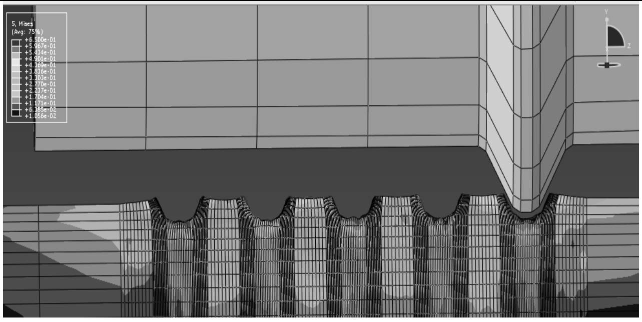
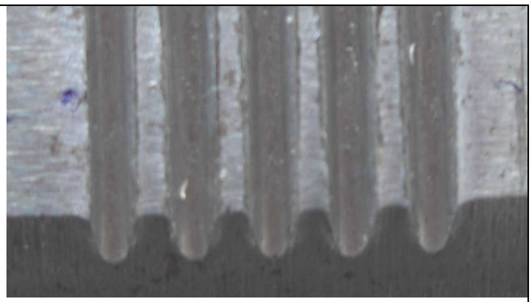


Fig. 6 Partial results obtained by experimental research on rolling force

From the graph we can notice that the value of the vertical force is approximately 3 times bigger than the horizontal force.



a) Partial results obtained through numerical simulation on the profile geometry



a) Partial results obtained by experimental research on the profile geometry

Fig.7 Partial results on the profile geometry

CONCLUSIONS

Using a numerical model of a process it can obtain its results even from the designing stage and it can correspondingly establish the processing parameters in order to reach certain objectives.

Through simulation using the FEM were obtained the values of the force required to achieve five channels, micro-hardness, residual stresses and profile geometry function of the characteristics of the part’s material and the input parameters of the process.

In this case the model was not entirely validated; the experimental part is still in progress.

AKNOWLEDGEMENTS

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 K., Diagnosis of involutometric issues in flat rolling of external helical gears through the use of finite-element models, The University of Michigan, 2006.

[2] Blumenfeld M., Introducere în metoda elementelor finite, Editura Tehnică, Bucureşti, 1995.

[3] Craveur C., Modélisation des structures. Calcul par éléments finis, Editura Masson, Paris, 1996.

[4] Daridon L., Curs de element finit, Universitatea din Strasbourg, 2000.

[5] Gârbea D., Analiză cu elemente finite, Editura Tehnică, Bucureşti, 1990.

[6] Marinescu G., Ivan C., Metoda elementului finit. Analiză numerică şi aplicații în termoelasticitate, Editura CIA, Bucureşti, 1996.

[7] Menezes F., Teodosiu C., Three-dimensional numerical simulation of the deep-drawing process using solid finite elements, Journal of Materials Processing Technology, pag. 100-106 Vol. 97, 2000.

[8] Nițu E., Ungureanu I., Iacomi D., Proiect de cercetare exploratorie, Modelarea analitică şi numerică a proceselor de prelucrare prin deformare plastică volumică la rece a profilelor complexe, ID_711 / 2008, pag. 150-165, Universitatea din Piteşti, 2009.