10th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2006, Barcelona-Lloret de Mar, Spain, 11-15 September, 2006

APPLYING FINITE ELEMENT METHOD FOR ANALYSYS OF IRONING PROCESS*

Mirna Nožić
Faculty of Mechanical Engineering
University "Džemal Bijedić"
Sjeverni logor bb, 88000 Mostar
BiH

Himzo Đukić
Faculty of Mechanical Engineering
University of Mostar
Matice Hrvatske bb, 88000 Mostar
BiH

Emir Šarić
Faculty of Mechanical Engineering
University of Tuzla
M.Fizovića 6, 75000 Tuzla
BiH

Džemo Tufekčić
Faculty of Mechanical Engineering
University of Tuzla
M.Fizovića 6, 75000 Tuzla
BiH

ABSTRACT
This paper gives a picture on stress and deformation when there is concurrent load on all the rings of a multi-stage tool in the process of ironing by FEM.
By comparing FEM results with the experimental ones, a good correspondence has been observed for a final stage of the drawing process.
Key words: FEM, ironing

1. INTRODUCTION
Nature of ironing and an excepted deformation values require application of a theory of big deformations in the analysis of the process with a finite element method. Since it is an axisymmetric problem, a 2 D-r adaptive net creation was used, in which a new network is created in a wanted moment, based on the initial network and characteristic element lengths, by using a method of smallest squares. In this way, good contact preconditions are created between the working piece and the tool and „bad" elements are prevented from creation during simulation.

2. FE MODEL
To create an FE model, a 3D CAD tool model was used for simulation of ironing.
The model is consisted of a prepared part of metal and a tool for deep drawing (draw die and rings), as shown in the Figure 1. Appropriate areas needed for 2D model were transferred through IGES interface into an ANSYS pre-processor. As a result, a 2D CAD model of the prepared piece of metal and the tool were acquired, as shown in the Picture 2.
For creation of the FE model of the prepared piece and the tool, an ANSYS pre-processor was used that splits CAD model into finite elements. The resulting FE model was a discrete model of the tool and the prepared piece, consisting of 1735 finite elements and 2008 nodes. 2D axisymmetric elements were used for discretization. Since the only deformable part of the model was the prepared piece, the Material 24 wsa chosen for it (from the Library of software material models of LS-Dyn), with possible usage of so-called „piecewise" – a linear-plastic izotrope curve of hardining, incorporated by Von Mises criterion of plastic flow.

Parts of the tool (draw die and the rings) are not deformed, so their constituent elemets are considered to be „rigid“. Due to axisymmetrical quality of the model, the chosen type among all the parts was the CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE.
The Figure 3 represents the initial FE model. The Table 1. gives the data on the initial FE model.

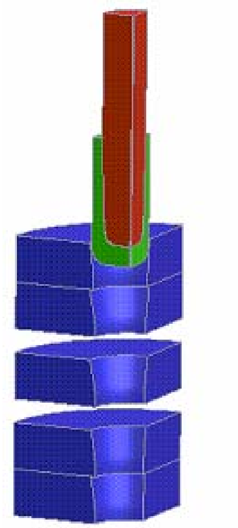

Figure 1. 3D CAD model of the prepared piece and the drawing tool

Figure 2. 2D CAD model of the prepared piece and the drawing tool

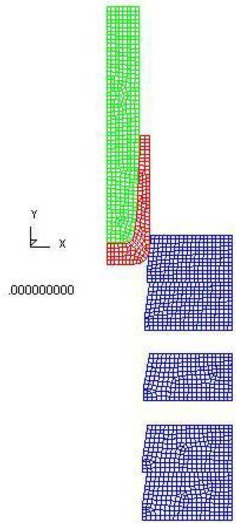

Figure 3. The initial FE model

Table 1. Data on the initial FE model

Parts	Number of elements	Type of element
Part 1 – Prepared piece	135	shell
Part 2 – Drawing device	446	shell
Part 3 – Rings	1154	shell
Σ	1735	

3. RESULTS OF STRESS-DEFORMATION ANALYSIS

Stress and deformation analysis by FEM was done for a five-stage tool of working pieces of the group 1. [1]. The Table 2 gives experimental values of stress and deformations in referent cross-sections of the researched five-stage tool, so that a comparison can be done with the results acquired by FEM. As referent cross-sections were taken those whose working piece bottom stick out of each ring of the tool.

Table 2. Values of stress and deformation in the referent cross-sections of the researched five-stage tool

Stress-deformation analysis of the production tool			
Referent cross-section	Deformation intensity ε_I	Logarithm deformation φ	Stress intensity $\left[kN/mm^2 \right]$
1	0.113	0.251	0.031
2	0.216	0.515	0.172
3	0.183	0.389	0.401
4	0.167	0.353	0.568
5	0.114	0.224	0.697

Results of the stress analysis are shown in the figures 4 and 5 for the referent cross-sections 3 and 4.

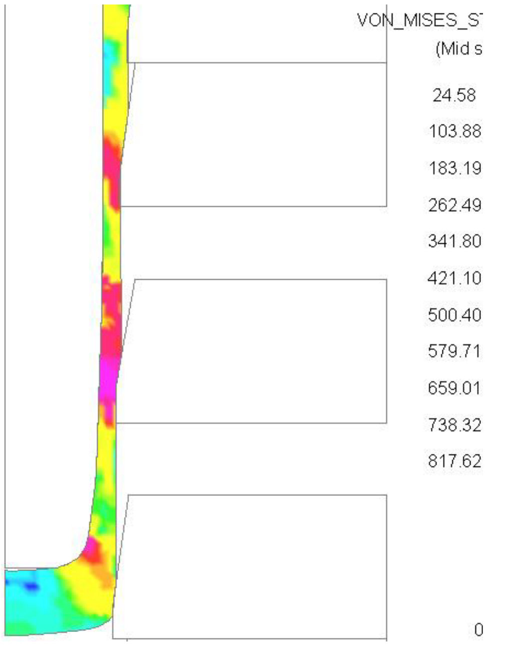

Figure 4. Stress diagram in the moment when the working piece is meeting the fourth ring

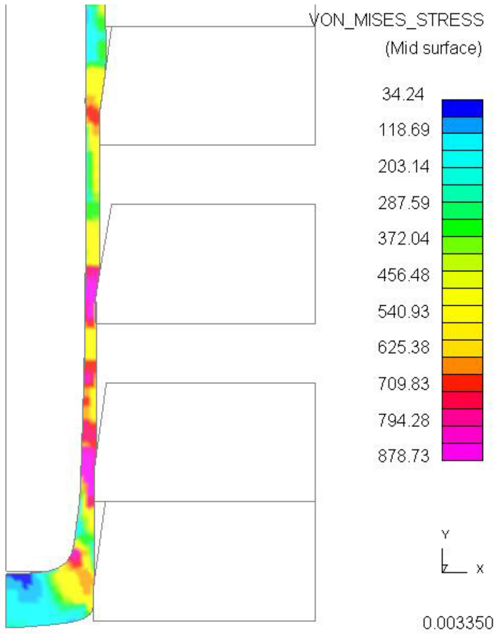

Figure 5 . Stress diagram of the moment when the working piece is in contact with all the rings

Results of the deformity analysis are given in the figures 6-9.

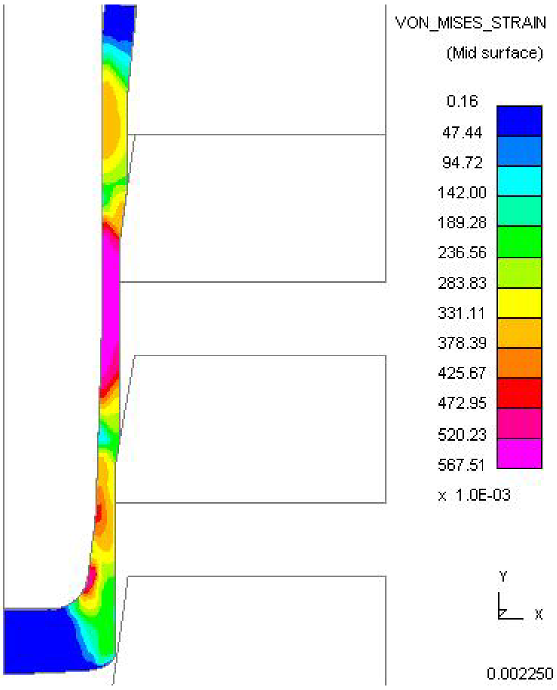

Figure 6. Intensity of deformation in the moment when the bottom of the working piece is in contact with the fourth ring

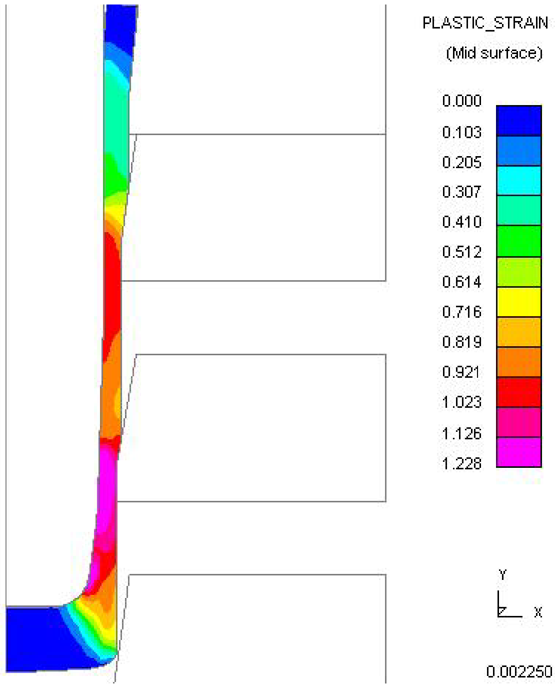

Figure 7. Logarithm deformation in the moment when the bottom of the working piece is in contact with the fourth ring

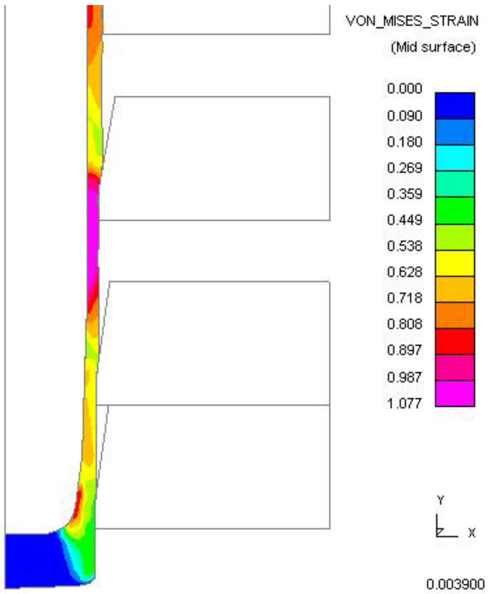

Figure 8. Intensity of deformation in the moment when the working piece bottom sticks out of the fifth ring

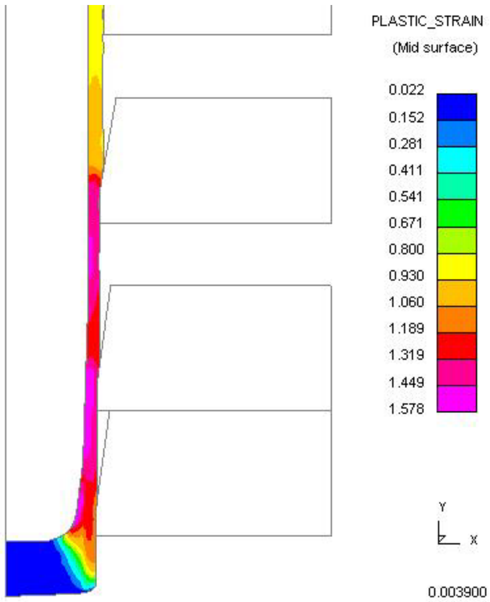

Figure 9. Logarithm deformation in the moment when deformacija u trenutku the working piece bottom sticks out of the fifth ring

Results of the stress analysis are given for the moment when the deformation of the working piece bottom starts on the fourth ring (Figure 4) and in the moment when the working piece bottom deformation starts on the fifth ring. (Figure 5).
The deformation diagrams show that maximum deformations occur on the shell, at the very place of contact of the working piece with the third and forth ring of the tool.
To compare results acquired by FEM with the experimental ones, it is important to mention that results of stress deformation analysis by FEM for the referent cross-section 2 equal the sum of the stresses (deformations) in the cross-sections 1 and 2 (Table 2), while the results for the cross-section 3 are equal to the sum of the stresses (deformations) in the first three cross-sections etc.
The given results of the stress analysis prove a good approximation with experimental results.
Deformation diagrams show that maximum deformations appear on the shell. Comparing the specific deformation values with the experimental ones for the moment when deformation begins in the forth ring of the working piece is 9.8% (Figure 6) and 5.94% (Figure 7).
In the final stage of the bottom deformation (Figure 9), deviation of the acquired results as compared with those experimental is 8.8 %.

4. CONCLUSION

Stress-deformation state of all rings in the multi-stage tool is very complex so the diagram of real load can be reached with the finite elements method. Based on the acquired results, some conclusions can be reached about individual load for each tool ring.
All the shown position acquired satisfying approximations with experimental results, with the best correspondence in the final stage of the drawing process.

5. REFERENCES

[1] Nožić M. : Prilog istraživanju pojedinačnog i ukupnog opterećenja višestepenih alata za izvlačenje sa redukcijom debljine zida, Doktorska disertacija, Mostar, 2005.
[2] Meinders T.: Developments in numerical simulations of real-life deep drawing process, Universiteit Twente, februari, 2000.
[3] Kobayashi S., Soo-Ik Oh, Taylan A.: Metal forming and the finite-element metod, Oxford University, 1989.

**** The paper is part of the research project „Contribution to research of single and total load in multi-stage drawing tools with wall thickness reduction“.***