**ANNALS of the ORADEA UNIVERSITY.**

**Fascicle of Management and Technological Engineering, Volume VII (XVII), 2008**

# ON SIMULATION OF METAL SHEET HIGH SPEED DEFORMATION

**Tiberiu VESSELENYI**[1]**, Pavel MUDURA**[1]**, Ioan MOGA**[1]
[1] University of Oradea
e-mail: vtiberiu@uoradea.ro

**Keywords** : finite element analysis, electromagnetic forming, aluminum deformation.

**ABSTRACT:** The paper presents the structural analysis part of a method for simulation of electromagnetic forming of an aluminum sheet, based on numerical model of transient electromagnetic field in computation domain and on transient analysis of visco-plastic forming of sheet under action of electromagnetic forces, resulting at each time step from magnetic computation.

## 1. INTRODUCTION

Electromagnetic forming of metals is a relatively new technology used to process complex shaped sheet metal parts and pipes mostly for automotive and aeronautical industries. Modeling of the electromagnetic forming implies multiphysiscs transient analysis capabilities [1], which are only met in finite element analysis programs. There are two main options to model coupled electromagnetic and structural processes. One of them is to model the electromagnetic physics in one environment, obtain the electromagnetic forces which are acting on the structural part and then import them into the structural analysis program which has viscoplastic deformation modeling capabilities. The other option would be to model the electromagnetic field and its interaction to the structure in the same environment.

This paper describes the structural physics of the analysis in the first approach. The electromagnetic analysis had been done in the FLUX 2D environment using the axial symmetry option. Then the mesh was exported to a text file which was decoded with the help of a MATLAB module written by the authors. The MATLAB module also accomplishes the task of decoding the force files and from the mesh point and force load information it generates an ANSYS input file (.inp). This file serves then as a command listing for the ANSYS structural environment. In the structural analysis the authors had chosen the viscoplastic model as ANSYS documentation is describing it that is capable to model large, rate – dependent deformations (element type PLANE183).

## 2. APPLICATION SETUP

In case of metals with high deformation ratios, the resistance to deformation depends on the speed of the deformation process [2]. Conventionally it is considered that between stress and deformation rate the following relation exists:

$$\sigma = k \cdot \dot{\varepsilon}^{m} \qquad (1)$$

The parameter “m” is a coefficient which characterizes the sensibility of the resistance to deformation as a function of deformation speed:

$$m = \frac{d(\log \sigma)}{d(\log \dot{\varepsilon})} \qquad (2)$$

Relation (2) is only a general characterization of the deformation rate dependence.

In order to obtain accurate descriptions of the phenomena a large number of theoretical and experimental studies were made, obtaining different rate – dependence laws. Among these there are two models which are largely used, laws which are also implemented the ANSYS finite element analysis environment.

These models are shortly defined in the followings.

The Perzyna model – has the following expression:

$$\sigma = \left[ 1 + \left( \frac{\dot{\varepsilon}^{pl}}{\gamma} \right)^m \right] \sigma_0 \qquad (3)$$

Where: σ is the yield stress of the material;

$\dot{\varepsilon}^{pl}$ - is the equivalent plastic strain rate;

m – rate –dependence sensibility parameter;

γ- viscosity parameter;

$\sigma_0$ – static yield stress of material.

According to ANSYS reference [3], $\sigma_0$ is a function of hardening parameter. When $\gamma \to \infty$ or $m \to 0$ or $\dot{\varepsilon}^{pl} \to 0$ solution converge to the static (rate - independent) solution. When m is very small (<0,1) the solution presents convergence difficulties.

The Peirce model – has the following expression:

$$\sigma = \left[ 1 + \left( \frac{\dot{\varepsilon}^{pl}}{\gamma} \right) \right]^m \sigma_0 \qquad (4)$$

Similar to the Perzyna model, solution converges to the static case as $\gamma \to \infty$ or $m \to 0$, $\dot{\varepsilon}^{pl} \to 0$. This model presents a better convergence than the Perzyna model, for small values of m.

For the present analysis the Perzyna model had been chosen. Other settings of simulation parameters are as follows:

- $\sigma_{pl} = 70*10^7$ [Pa];
- $G = 10^7$ [Pa]
- m = 0.5;
- γ = 1.

The electromagnetic forces, imported from results of FLUX2D program in 100 steps, were applied as loads in the transient structural model. A diagram of force vector distribution on the analyzed sheet is given in figure 1. The sheet has a disk shape with a radius of 70 mm and 1 mm thick. The structural model was defined in an axial symmetric mode.

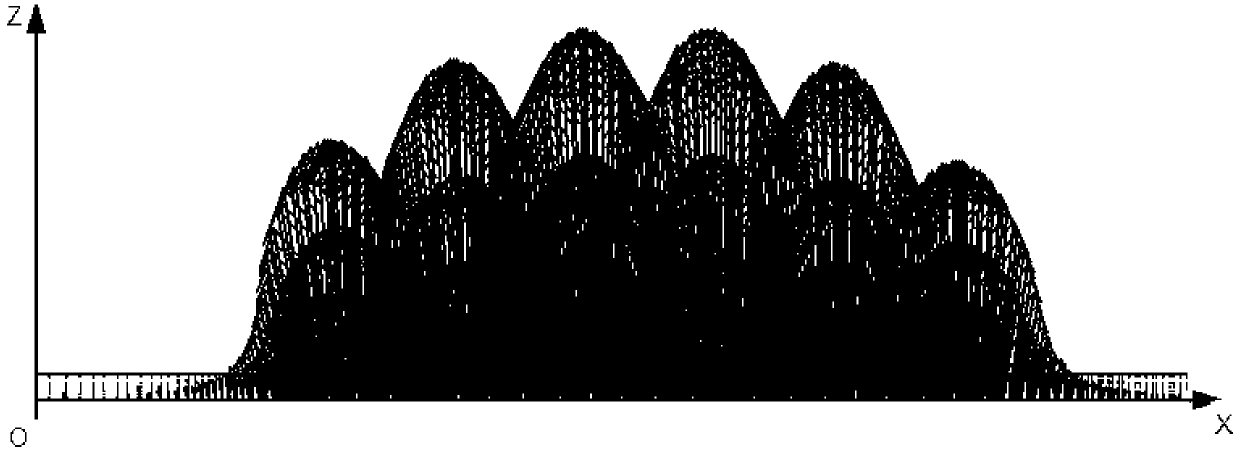

*Fig. 1.Electromagnetic forces applied on the part.*

## 3. SIMULATION RESULTS

Simulations were made for different values of the presented parameters ($\sigma_{pl}$,G,m,γ). There are some significant qualitative differences for different parameter settings. Figures 2 to 7 presents a study of the influence of $\sigma_{pl}$, on the deformational behavior.

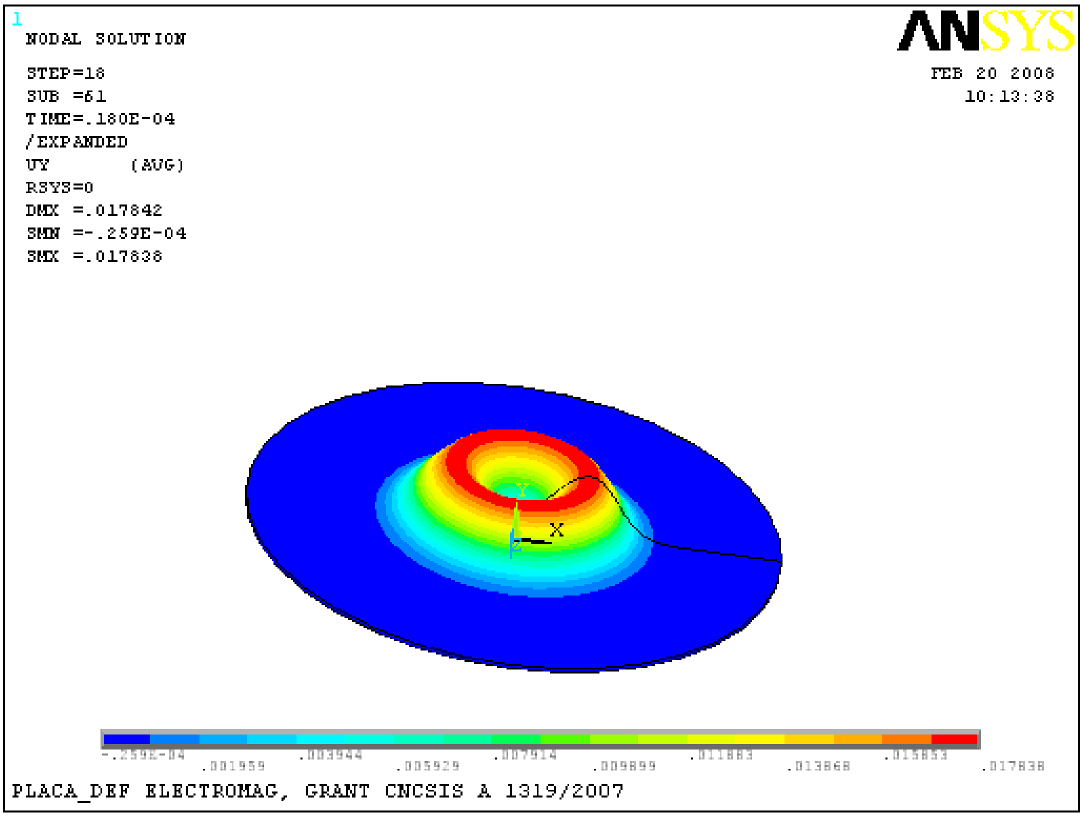

*Fig. 2. Displacements diagram at step 18. ($\sigma_{pl}$ = 70*$10^7$ [Pa]).*

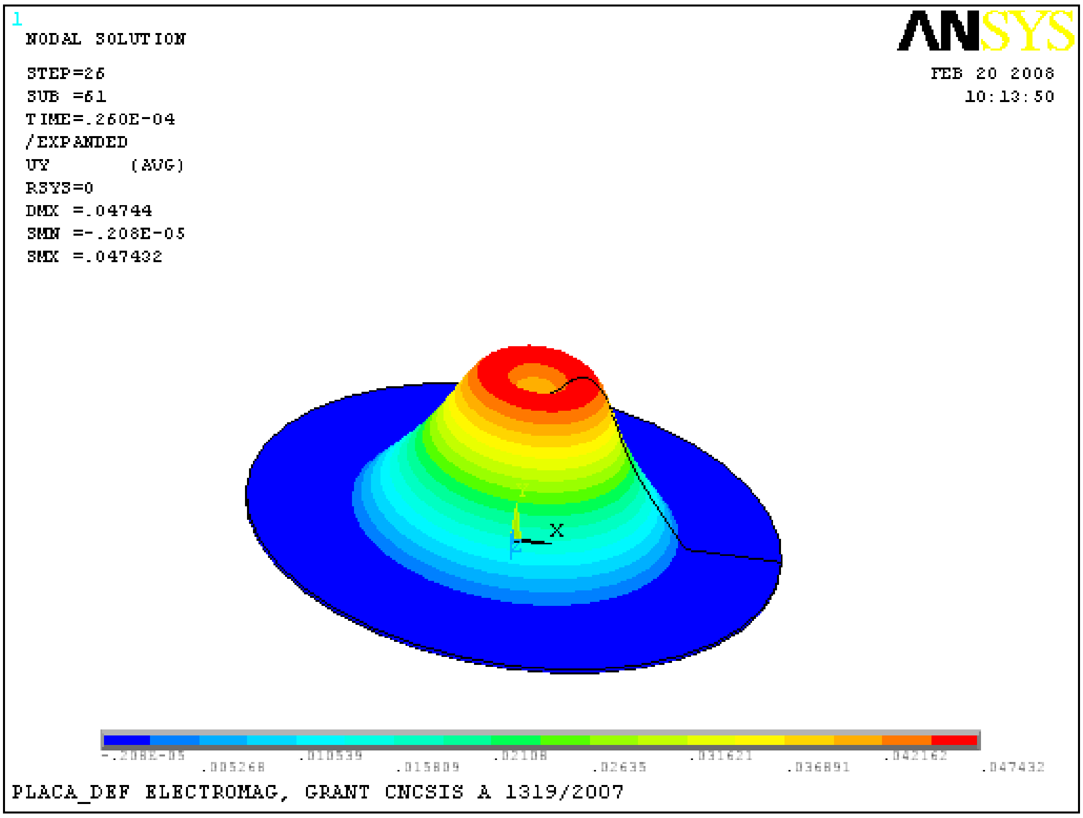

*Fig. 3. Displacements diagram at step 26. ($\sigma_{pl} = 70*10^7$ [Pa]).*

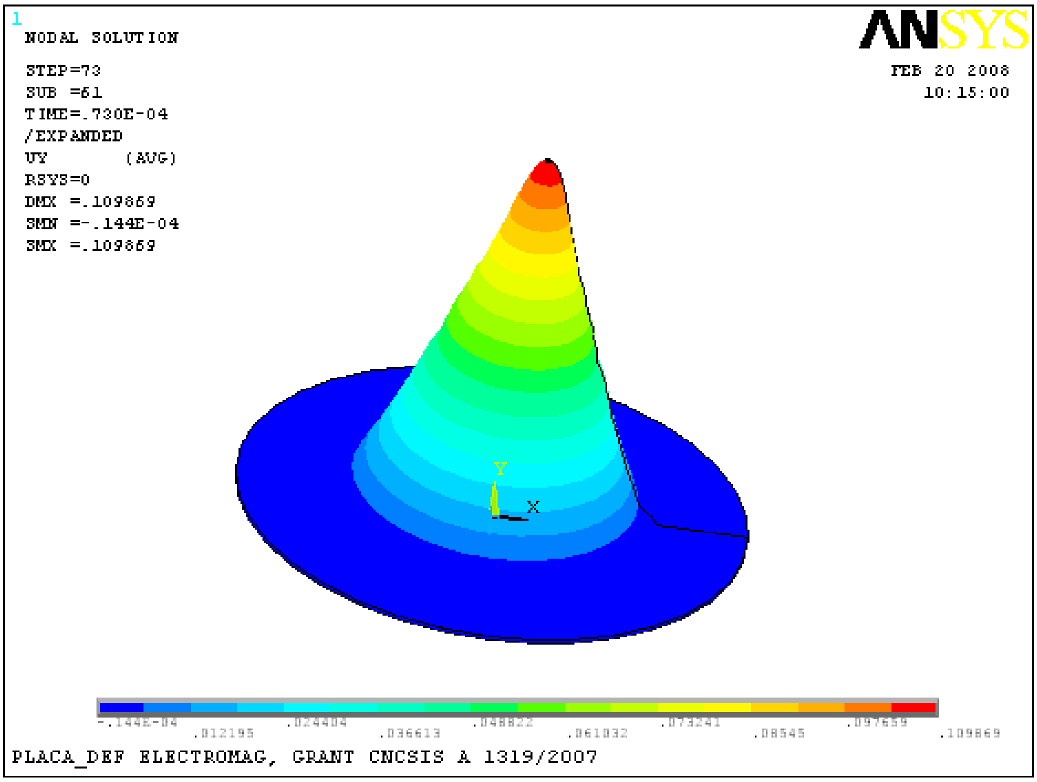

*Fig. 4. Displacements diagram at step 73. ($\sigma_{pl} = 90*10^7$ [Pa]).*

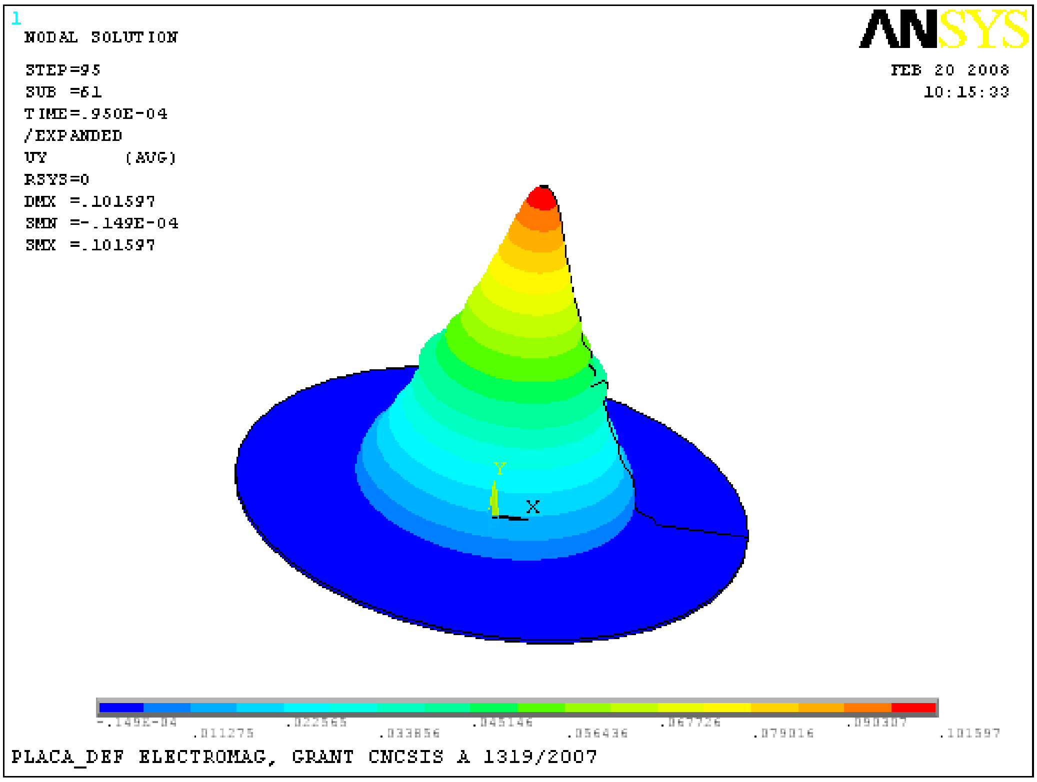

*Fig.5. Displacements diagram at step 95. ($\sigma_{pl}$ = 90*$10^7$ [Pa]).*

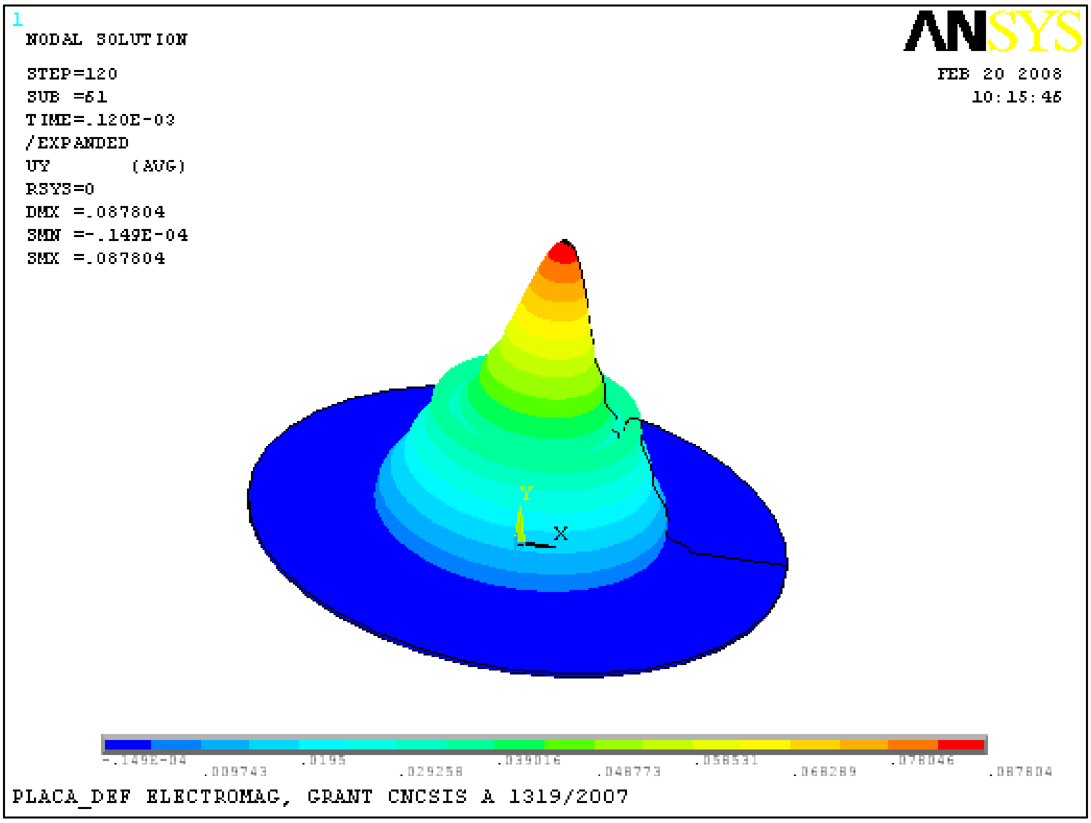

*Fig. 6. . Displacements diagram at step 120. ($\sigma_{pl}$ =90*$10^7$ [Pa]).*

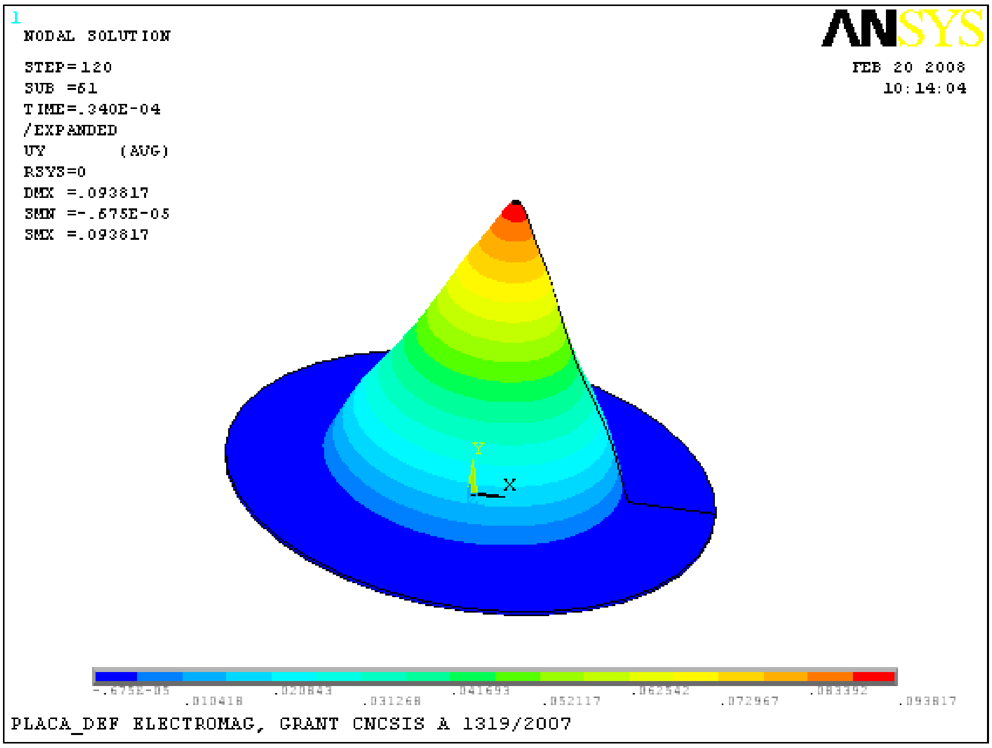

***Fig. 7. Displacements diagram at step 120. ($\sigma_{pl} = 70*10^7$ [Pa]).***

In the first steps the sheet is deformed in the same manner, for different values of the yield stress (figures 2 and 3). For the $\sigma_{pl}$ = $90*10^7$ Pa case the material is too elastic and a revenue from the maximal deformation state is observed (figures 4,5 and 6). For yield stress values around $70*10_7$ Pa, the deformation stops at the maximal position (figure 7). For values smaller than $65*10^7$ Pa the material wil have very large uncontrollable deformations resulting in simulation failure.

## 4. CONCLUSIONS.

Results obtained in deformation simulation will help in the design of electromagnetic circuits and its components, by establishing the right values of forces which can be applied to a certain material with given geometric configuration.

The next step in our study will by the development of a fully coupled electromagnetic – structural model.

## 5. ACKNOWLEDGEMENTS

This work was supported by Romanian Ministry of Education and Research, Grant CNCSIS A 1319/2007, 34GR/21.05

## REFERENCES

[1] S. Pasca, V. Fireteanu, “Transient Magnetic Finite Element Model of Magnetoforming”, *Rev. Acta Electrotehnica*, Vol. 45, No. 6/2004, pp. 575-580
[2] Shiro Kobayashi, Soo-ik Oh, Taylan Altan, Metal forming and the finite element method, New York, Oxford, Oxford University Press, 1989.
[3] *** Ansys, Inc. Theory Reference, Ansys release 10.0, 002184, 2005.