

MULTIPHYSICS SIMULATION OF METAL SHEET ELECTROMAGNETIC FORMING

O. Fruitós*, R. Otín*, R. Méndez*, J. Llumà**

* International Center for Numerical Methods in Engineering (CIMNE)
Av. del Canal Olímpic, s/n, 08860 Castelldefels, Barcelona, Spain
e-mail: metalform@cimne.upc.edu, web page: <http://www.cimne.com>

** Dep. Ciència de Materials i Enginyeria Metal·lúrgica, EUETIB,
Universitat Politècnica de Catalunya, Comte d'Urgell 187, 08036 Barcelona, Spain
e-mail: jordi.lluma@upc.edu, web page: <http://www.upc.edu/cmemb/>

Key words: Electromagnetic forming, multiphysics simulation, frequency domain, finite elements.

Summary. Electromagnetic Forming (EMF) is a deformation process at high speed that couples electromagnetic, mechanical and thermal problems.

Due to the complexity of these couplings, the simulation of the process via finite elements (FEM) is a not a straightforward question. Nevertheless, the software must be able to simulate the electromagnetic fields and its mechanical effect on the work pieces, induced eddy currents and, at the same time, it must predict the deformed shape of the work piece and the induced thermal effects as a consequence of both electromagnetic phenomena and the heat caused by the plastic deformation of work piece.

Nowadays the few commercial software able to simulate multiphysic problems show certain limitations. That is why we propose a new simulation approach, based on the codes ERMES and Stampack®, which solves the above mentioned limitations while maintaining a good correlation with the experimental results.

1 INTRODUCTION

Electromagnetic Forming (EMF) is a metal working process that relies on the use of electromagnetic forces to deform metallic workpieces at high speeds. The EMF pieces can be used alone or combined with the usual mechanic methods. In this process, a transient electric current is produced in a coil by a capacitor bank and high-speed switches. This current generates a magnetic field that penetrates the nearby conductive workpiece where eddy currents are induced. The magnetic field, together with the eddy current, induces Lorentz forces that drive the deformation of the workpiece. In an EMF process, the material can achieve velocities in the order of 200 m/s in less that 0.1 ms. The dynamics of this event, including die impact, enhance the formability of the workpiece and reduce springback.

With the current state of the art of software, the EMF problems consist in two separate analyses. CIMNE is working into coupling the two analyses in only one process, and this paper shows the actual state of the coupling.

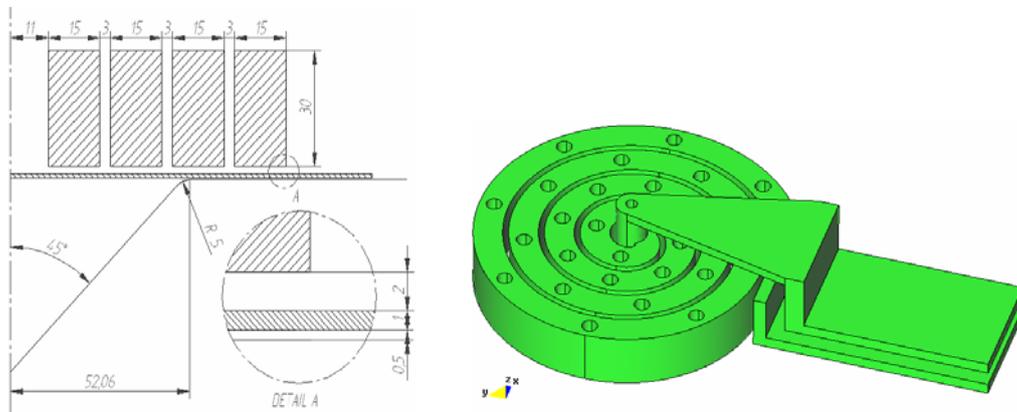


Figure 1: The LABEIN-Tecnalia's experimental device to make an electromagnetic forming of a cone ^[3].

2 SIMULATION PROCESS

To simulate the problem two independent well tested codes were employed: ERMES and Stampack.

ERMES is a FEM Electromagnetic code that solves Maxwell's Equations in frequency domain ^{[1], [2]} and is used to solve the electromagnetic interaction between the coil and the metallic sheet.

Stampack is a metal forming simulation software with dynamic explicit solver for formability and dynamic implicit solver for spring-back prediction, designed for many industrial applications and simulates the sheet deformation due to the electromagnetic force.

The pre and postprocessor resources of both computational tools are based on the GID platform, allowing a relative easy way of interaction and data interchange.

With the GID pre-processor, a simplified geometry coil (see Fig. 2 left) is informed to both solvers. As Stampack could only use Cartesian planes as a symmetry planes, a quarter of the

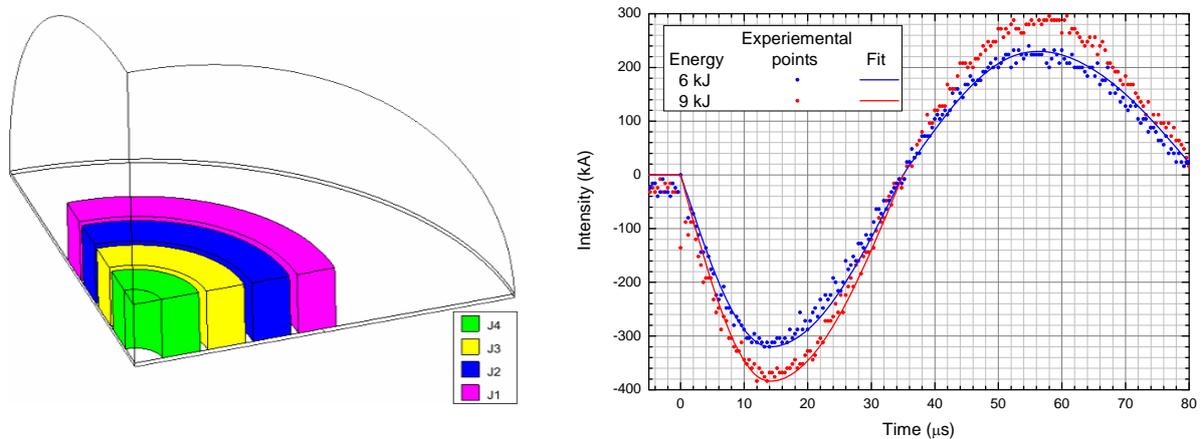


Figure 2: Simulated geometry ^[3] (left), and time evolution of the experimental current intensity ^[3] and fitted curve to compute the magnetic force (right).

axisymmetrized geometry is required to simulate the 3D problem with 2 planes of symmetry. The material properties must also be informed; in this example the sheet is made of 1050-H14 Aluminium alloy and the coil is made of electrolytic copper (99.9% Cu minimum).

Initially ERMES computes the magnetic field between the copper wires of the coil and the undeformed aluminium plate for a unitary current (1 A). The metallic sheet is modelized using the Impedance Boundary Condition (IBC).

ERMES output field is open by Stampack to obtain the unitary magnetic field module (H_1) over the nodes of the plate to compute the unitary magnetic pressure (P_1) as:

$$P_1 = \mu_0 \frac{|H_1|^2}{2} \quad (1)$$

where μ_0 is the vacuum magnetic permeability. At the actual point of the work, Stampack assumes the magnetic pressures normal to the surface (like in a hydroforming process). Although it is a formal error, the tangential component of the magnetic field is quite small and also the induced error.

Stampack simulate the time evolution of the plastic deformation of the plate due to the instantaneous magnetic pressure ($P(t)$), witch corresponds to:

$$P(t) = P_1 \cdot I(t)^2 \quad (2)$$

where $I(t)$ is the instantaneous current intensity (see Fig. 2 right).

Each time that the change in the shape of the sheet is high enough to have a significant effect on the value of the unitary magnetic pressure, ERMES solver is called again with the actual geometry to recompute the distribution of magnetic field.

The iteration process goes on until the deformation speed is only due to the vibration.

11 RESULTS

The final shape and the relative thickness results are shown in figs 3 and 4. The dispersion in the experimental vertical displacement is due to nonsymmetric design of the coil, which

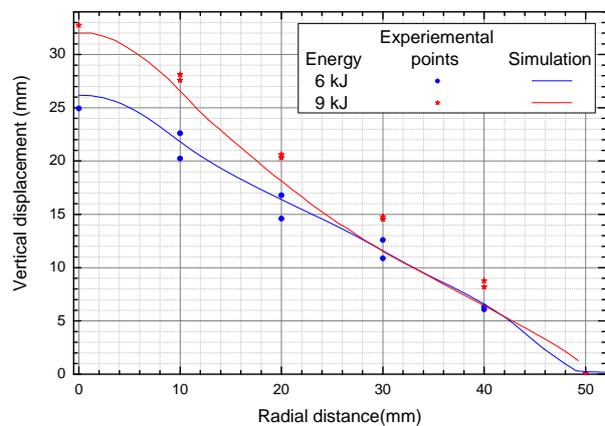


Figure 3: Formed sheet at 6 kJ^[3] (left), and vertical displacement simulated and measured at 6 and 9 kJ (right).

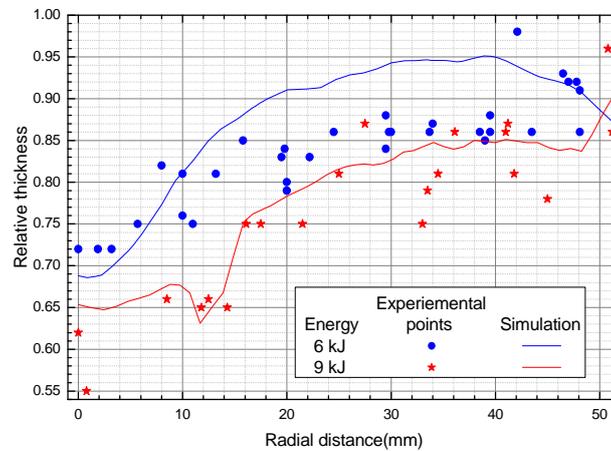


Figure 4: Relative thickness simulated and measured at 6 and 9 kJ.

induces a slightly horizontal displacement of the vertex position. Other factor to consider can be the rate strain dependence and the tangential component of the magnetic pressure.

12 CONCLUSIONS

- The simulation of the final shape is in good arrangement with the experimental data.
- The simulation of the thickness is in good arrangement with the experimental data.
- The aspects mentioned in the chapter of the results are being studied at the present.

ACKNOWLEDGEMENTS

The authors thank the Spanish “Ministerio de Educación y Ciencia” their funding in the SICEM project (Simulación multifísica para el diseño de conformado electromagnético) (REF.: DPI2006-15677-C02-01). It is coordinated by CIMNE and with the participation of LBEIN, Tecnalia.

REFERENCES

- [1] R. Otin, “A High-order finite element formulation for microwave engineering”, 24th Annual Review of Progress in Applied Computational Electromagnetics (ACES 2008), Canada.
- [2] R. Otin, “Regularized Maxwell equations with nodal elements as an alternative approach to the edge-based FEM formulations”, 9th International Workshop on Finite Elements for Microwave Engineering, 8-9 Mayo 2008, Bonn (Alemania) .
- [3] I. Aranguren, I. Pérez, B. González, I. Eguia, “A New Multiphysics Simulation Approach for Electromagnetic Forming”, 8th. World Congress on Computational Mechanics, a791 (2008).