8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

FINITE ELEMENT MODELING OF CLINCHED CONNECTIONS

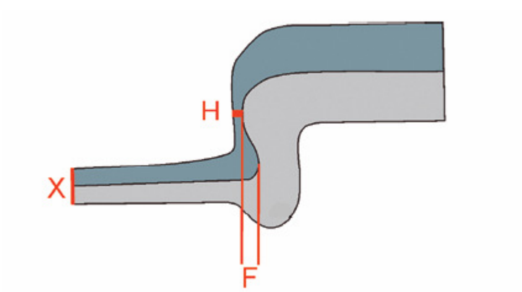
***D. Debruyne[1], S. Coppieters, S. Cooreman and M. De Strycker**

[1] Technical University KaHo St-Lieven,
Department of Mechanical Engineering,
9000 Gent, BELGIUM
dimitri.debruyne@kahosl.be
www.kahosl.be

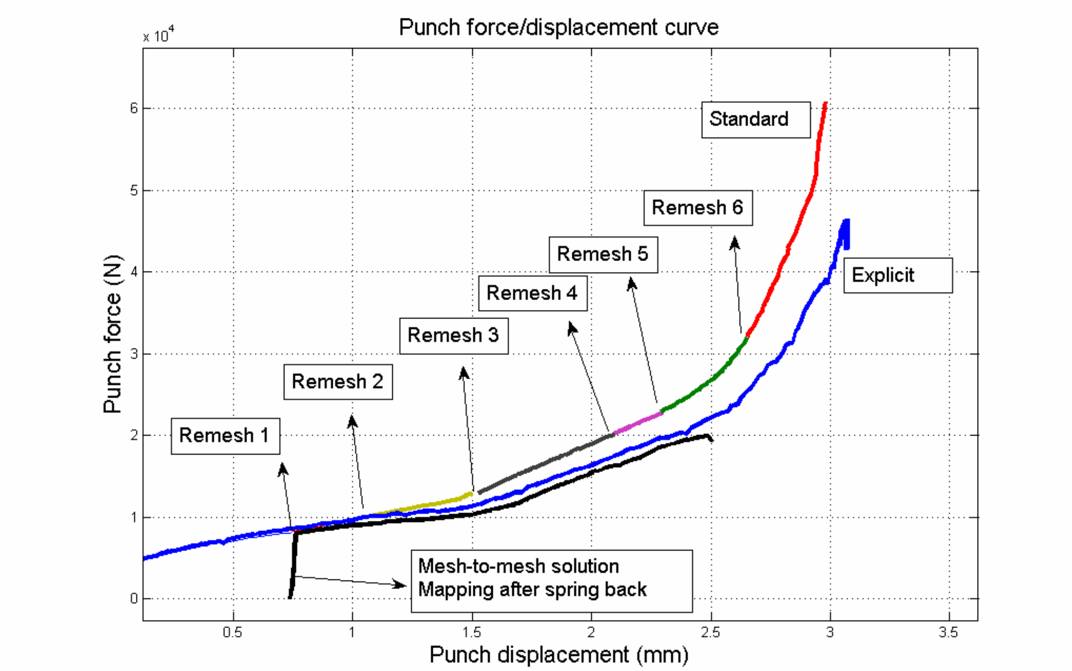
Key Words: *Finite element method, clinching, metalforming, plasticity*

ABSTRACT

The basic principle of the clinch process, also referred to as press joining, is to create an interlock F (see Figure 1) between two or more metal sheet parts through severe localized plastic deformation. This deformation is achieved with the aid of relatively simple tools: a die, a blankholder and a punch. The strength of the connection depends on the elastoplastic material properties of the sheet metal but also on the geometry of the clinch, which in turn is determined by the punch and die geometry and the displacement of the punch during the forming process. The optimum geometry for the punch and die is usually determined experimentally by trying out different combinations and dimensions. This can be a time-consuming and costly adventure. By using finite element simulations of the forming process it is now possible to determine the optimum die and punch geometry for a given application. Unfortunately, these simulations still cope with a number of technical difficulties. This contribution presents a discussion of the finite element modeling of clinched connections and the issue of severe mesh distortion due to the high plastic strains encountered.



Several authors have reported about their research of the clinching process with the aid of finite element methods and this issue with severe mesh distortions, see for example [1,2,3]. In this study we explore the use of the finite element procedures implemented in the finite element code Abaqus. The anisotropy in the plate material is ignored and the material is modeled to be isotropic. Considering the axisymmetric nature of the studied clinched joint - the present paper concentrates on the so-called 'non-cutting single stroke' clinch process - this results in an axisymmetric finite element model. As the forming of a clinched joint involves severe plastic deformation the system becomes ill-conditioned due to the excessive element distortion. Possible solution paths using different remeshing techniques to counter this problem are discussed. A comparison of the clinch forming simulation using an implicit and explicit integration scheme is presented. Both procedures are implemented in Abaqus and offer possibilities (ALE adaptive meshing and mesh-to-mesh solution mapping) to counter the excessive element distortion. The punch force versus the punch displacement resulting from both procedures and respective remeshing techniques are shown in Figure 2.



REFERENCES

[1] A.A. de Paula, M.T.P. Aguilar, A.E.M. Pertence, and P.R. Cetlin, "Finite element simulations of the clinch joining of metallic sheets", J Mat Process Tech, 182, 352-357, 2006.

[2] V. Hamel, J.M. Roelandt, J.N. and F. Schmit, "Finite element modeling of clinch forming with automatic remeshing", Comp Struct, 77, 185-200, 2000.

[3] J.P. Varis and J. Lepistö, "A simple testing-based procedure and simulation of the clinching process using finite element analysis for establishing clinching parameters", Thin Wall Struct, 41, 691-709, 2003.