Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 207 (2017) 437–441

Procedia Engineering

www.elsevier.com/locate/procedia

International Conference on the Technology of Plasticity, ICTP 2017, 17-22 September 2017, Cambridge, United Kingdom

Damage dependent material properties in a Finite Element Simulation of a hybrid forward extrusion process

Stephan Hojda[a)]*, Karl J.X. Sturm[a)], Michael Terhorst[b)], Fritz Klocke[c)], Gerhard Hirt[a)]

[a]Institute of Metal Forming (IBF), RWTH Aachen University, Intzestr. 10, 52056 Aachen, Germany
[b]Carcoustics International, Neuenkamp 8, 51381 Leverkusen, Germany
[c]Laboratory for Machine Tools and Production Engineering (WZL), RWTH Aachen University, Steinbachstr. 19, 52074 Aachen, Germany

---

Abstract

The occurrence of damage and fracture limits modern forming processes. An example for such a process is the full forward extrusion of steel, where accumulating damage results in characteristic chevron cracks. To prevent chevron cracks, in conventional extrusion, the billet is pre-heated before forming. As an alternative development in a hybrid approach the workpiece is heated resistively, by applying an electric current during extrusion. This results in an inhomogeneous temperature distribution in the workpiece, influenced by the local resistivity of the material, which in turn is dependent both on temperature and damage. While the press force is measured and a cut through the final part gives information on the occurrence of failure, it is not possible to measure local temperatures within the workpiece during forming. Since experimental results show that press force does not significantly change with resistive heating, numerical methods have to be used to investigate the influence of property changes of the material on the local temperature in the workpiece. In this work a 3D Finite Element Method model using Abaqus/Standard is presented that is capable of simulating the highly nonlinear hybrid forward extrusion process (thermomechanical and electric field) incorporating a remeshing routine. Subsequently, this model is extended to couple a strain based damage with electrical resistivity. Validation is performed by comparisons with experimental results for the conventional cold, pre-heated and hybrid approach. Notably, the results show a significant influence on the temperature distribution in the workpiece for simulations with strain coupling compared to those with unaffected material properties. This in conclusion shows how the presented method of coupling material properties to strain values enables new possibilities to design and optimize complex forming processes.

Peer-review under responsibility of the scientific committee of the International Conference on the Technology of Plasticity.

Keywords: Damage, Hybrid Extrusion, Finite Element Method (FEM)

1. Introduction

Metal forming relies on materials which exert a certain amount of ductility. However, at some point, critical failure occurs and thus limits forming capabilities. A major driver in the damage evolution of steel is the increasing number of dislocations with increasing plastic strain. The dislocations aggregate at inclusions and grain boundaries. In the first phase, this process manifests itself as work hardening on the macroscopic scale. But when a critical stress at those accumulations is exceeded, small voids form. With increasing strain, the small voids grow and coalescence with neighbouring ones, putting even more stress on the yet intact material in-between. Said sections are then prone to necking, ultimately leading to failure [1].

Deformation and resulting fracture are known to alter the electrical resistance of metals. Contributing to this effect is the mentioned growth of dislocation density with plastic strain. This has been quantified for multiple grades of steel [2,3]. With the

---

* Corresponding author. Tel.: +49-241-80-95946; fax: +49-241-80-92234.
E-mail address: hojda@ibf.rwth-aachen.de

1877-7058
Peer-review under responsibility of the scientific committee of the International Conference on the Technology of Plasticity.
10.1016/j.proeng.2017.10.801

formation of micro fractures and pores, the local resistivity of the material is further increased [4]. Additionally, a direct influence of grain size on resistivity during the process of recrystallization is known [5].

So far, the mentioned observations are applied to monitor wear and fatigue of components in a laboratory environment [6]. Similarly, industrial implementations exist, such as for supervision of oil pipelines [7].

Also with regard to forming processes, the described connection between deformation, damage and electrical properties are of interest. Conductive heating of sheet metal before deep drawing is known to reduce springback and increase formability [8], and the method was further developed to use with spline forming by heating the side walls of a cup conductively before forming [9]. Also, a hybrid incremental forming process was demonstrated, where an electric current was passed through the tool tip into the metal sheet to heat it locally [10].

It becomes apparent that damage, even without macroscopic failure, and resulting effects need to be incorporated into simulations of forming processes to properly optimize them. This study will exert said adaption for a new variant of forward extrusion that has been proposed by Klocke et al. [11]. Here, resistive heating during the extrusion process alters the temperature distribution and material flow in the workpiece. To reproduce this forming process numerically using the Finite Element Method (FEM), the following requirements for the simulation model must be met:

- Solver: coupling of plastic and thermal effects with electrical field equations,
- Material: material properties, especially resistivity, with its dependence on a damage value,
- Discretization (spatial): remeshing because of large deformations.

The purpose of this study is the implementation and validation of a model that fulfils these requirements. The feedback of damage to electro-mechanical material characteristics for use of electro-thermal-mechanical FE analyses has not yet been implemented on large deformation forming processes.

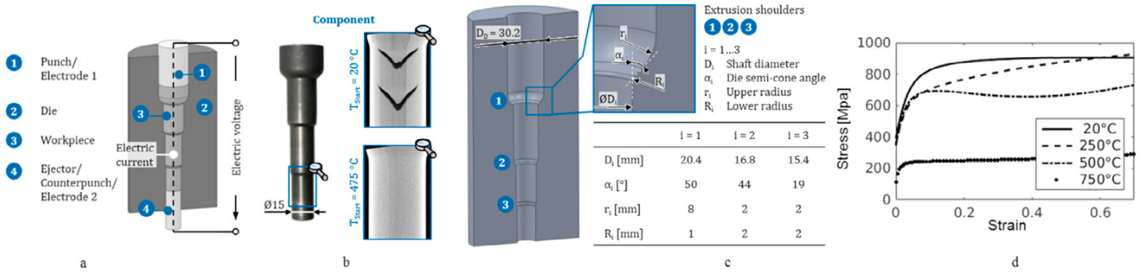

Fig. 1: a) solid forward extrusion process extended with resistive heating b) shape of the formed part and computer tomographic image of cold and warm formed part in the region of extrusion shoulder three c) geometry of the extrusion die (based on [11,15]) d) flow curves for different temperatures at a constant strain rate of 1 s^{-1}

2. Hybridized solid forward extrusion

Hybridized solid forward extrusion is an adaption of conventional solid forward extrusion where a punch presses a metal cylinder through the die. In order to make use of work hardening, conventional solid forward extrusion is often realized as cold forming. Doing so leads to the formation of chevron cracks once the workpiece formability is exceeded. This is due to tensile stresses and occurs on the rotational workpiece axis [12]. The cracks can, for instance, be avoided by increased process temperatures [11]. Figure 1 shows an extruded component made of Cf53 (AISI 1055) where extrusion has to be performed at initial temperatures T_{Start} > 475 °C to thermally avoid chevron cracks [11].

When industrial extrusion is executed at the process limits, chevron cracks usually occur below the last extrusion shoulder, as this is where highest tensile stresses arise. In Fig. 1 b computer tomographic images of a sample with failure and one without are given. This can be counteracted by applying direct current (DC) resistance heating within the forming tool during extrusion. It effectively provides local heating in the area that is prone to develop cracks. Figure 1 a shows the concept of this hybridized process. Punch and ejector are in contact with the workpiece throughout the process and at the same time function as electrodes. The local heating in the area of interest is achieved by passing the electric current through the reduced cross-section, compared with the rest of the workpiece. It also has to be assumed that change in electric resistivity due to a high dislocation density is largest in the region of the third shoulder.

3. FE model

The FE model of hybridized solid forward extrusion, as described in the previous section, is based on a cold solid forward extrusion process and being partially extended. Due to the higher flexibility in terms of adapting the numerical code, the FE software Abaqus is chosen. The implicit solver is used because of the suitability for quasi-static problems such as extrusion.