Available online at www.sciencedirect.com

ScienceDirect

Procedia CIRP 58 (2017) 104 – 109

www.elsevier.com/locate/procedia

16th CIRP Conference on Modelling of Machining Operations

Kinematic hardening of AISI 5120 during machining operations

Andreas Fellmeth[a]*, Frederik Zanger[a], Volker Schulze[a]

[a]*wbk – Institute of Production Science, Karlsruhe Institute of Technology (KIT), Engelbert-Arnold-Straße 8, 76131 Karlsruhe, Germany*

* Corresponding author. Tel.: +49 721 608-46316; fax: +49 721 608-45004. *E-mail address:* andreas.fellmeth@kit.edu

Abstract

Metal manufacturing processes like machining include complicated load cases and significant plastic deformation inside the manufactured component. The Finite-Element-Method (FEM) has been successfully applied to analyze machining processes.
The plastic deformations during machining operations, especially of ductile materials, are a major part of the total deformation. If the deformation incorporates a large plastic deformation part with changing spatial directions, kinematic hardening should be considered, additionally to isotropic hardening. Previous work on the kinematic hardening of ARMCO iron revealed an almost near constant ratio of isotropic and kinematic hardening. The constant kinematic hardening ratio is revised and analyzed in tensile-compression tests with normalized AISI 5120. The FEM simulation results using the new material model of the kinematically hardening AISI 5120 are validated with experimental force measurement during orthogonal machining. The influence of kinematic hardening during machining operations is not the major influence, but still substantial.

Keywords: Finite element method (FEM), Machining, Kinematic hardening

1. Introduction

Machining operations include complicated load cases inside the manufactured component. The FEM has been applied to analyze the machining process, and characteristics like temperature, residual stresses and process forces.

Machining operations are very hard to simplify, considering the large amount of interdependencies between the thermo-mechanical variables calculated within the analysis. Usually, plastic deformation heats up the material, which results in changing temperatures, which themselves change the material properties. Most engineering simulations only incorporate isotropic hardening, because it is both easier to implement and to test for, and usually the more dominant material behavior compared to kinematic hardening. Kinematic hardening is however an important effect [1,2], depending on the degree of simplification and abstraction.

Fig. 1 shows the basic concept of kinematic hardening and its effect on the movement of the yield surface. In contrast to isotropic hardening, the yield surface keeps its initial size

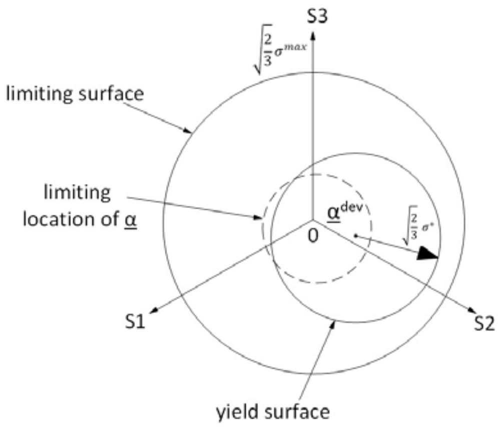

Fig. 1 Kinematic hardening in the stress space [3]

doi:10.1016/j.procir.2017.03.201

during pure kinematic hardening. Fig. 1 also shows a bounding limit to the amount of kinematic hardening, which is contained within a cylinder, due to the Mises yield stress definition, as described in the ABAQUS documentation [3].

Groundwork in the field of kinematic hardening was published by Armstrong and Frederik [4]. The application to viscoplasticity was done by Malinin and Khadjinsky [2,5]. To apply this work to modern FEM, additional problems had to be solved.

Notably Simo and Taylor published about the importance of consistent tangent operators when using a nonlinear 'incremental' model [6]. This tangent operator makes the quadratic convergence of solutions possible, gained by the iterative Newton method [7]. The use of the 'normal' elastic tangent modulus would not result in optimal convergence, but still give the same result. The constitutive equations derived describe J_2 plasticity in this work.

Because of the details of the implementation, a strain rate and temperature dependent kinematic hardening model is not stable with a fully thermo-mechanically coupled FEM simulation subjected to large nonlinear material deformations. For highly non-linear problems, as in thermo-mechanical numerical calculations, the combined isotropic-kinematic-hardening routine may trigger convergence problems [3, 12]. These might arise due to convergence issues during the Newton algorithm, which is implemented to solve the material equations during FE-analysis. This would eliminate the possibility to find a solution using an acceptable time increment. Small models, with small deformations, subjected to moderate temperature changes can be calculated using a custom UMAT subroutine, following the implementation introduced by Simo and Taylor [6]. This UMAT allows complete freedom in material model implementation, and allows for the implementation of a consistent tangent modulus. As for the intrinsic software possibilities, the calculation of kinematic hardening combined with any changing thermal simulation component using the proprietary functions within ABAQUS is prohibited by the software itself.

The kinematic hardening model is therefore implemented in the ABAQUS 'combined kinematic hardening'-model, which can include thermal dependencies (at different isothermal states) and strain rate dependencies (the strain rate dependencies can be used directly within the model, if the deformation produces different strain rates within the geometry).

With those limitations at hand, the influence of kinematic hardening on the cutting simulation is analyzed, the question being: How large is the influence of kinematic hardening if used in a simulation model on the simulation results, compared to a simulation model with isotropic hardening only, and compared to a simulation model with different constant friction parameters.

Nomenclature

γ	Scalar value (zeroth order tensor, small letters)
$\boldsymbol{a}$	First order tensor (bold small latin letters)
$\boldsymbol{\sigma}$	Second order tensor (bold small greek letters)
$\boldsymbol{C}$	Fourth order tensor (bold capital letters)

1.1. Preliminary remarks

Zanger et al. [8] used a constant linear kinematic hardening ratio, called Bauschinger-effect-parameter, which was originally mentioned in the work of Ibrahim and Embury [9] and which is constant in the range of the total equivalent plastic strain.

The ratio used to describe the experimental findings in this work is different compared to Zanger et al. [8]. A more detailed material model of normalized AISI 5120 was used, compared to the normalized ARMCO iron in the previous work.

In this paper, first, the hardening data and methods are established while explaining the choices made. Then the experimental results are being described and compared to the simulation. A closing discussion follows. Closing remarks are then made on further need for research on this topic.

2. Experiments and simulation methods

2.1. Kinematic hardening characterization and experimental results

To determine material parameters related to kinematic hardening in the specific material AISI 5120 (normalized), experimental work was necessary.

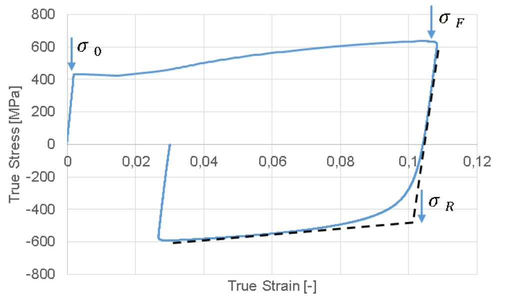

Fig. 2 True strain - true stress measurement, tensile-compression, at room temperature (approx. 293 K), force regulated, low strain rate. Material AISI 5120.

The results to implement the movement of the yield surface in the stress space were quantified by dividing the difference of the maximum stress in the primary load direction σ_F and the second opposite load direction σ_R by σ_F, see equation (1). The values used are visualized in Fig. 2. As the movement of the yield surface consists of only half the ratio, the ratio needs to be divided by 2. The resulting ratio is called r_{imp} ("ratio for implementation").

$$r_{\text{imp}} = \frac{\sigma_F - \sigma_R}{2 \cdot \sigma_F} \quad (1)$$

This quotient r_{imp} will directly be used to accordingly modify the isotropic hardening material model data.