European Journal of Mechanics A/Solids 49 (2015) 548–560

Contents lists available at ScienceDirect

European Journal of Mechanics A/Solids

journal homepage: www.elsevier.com/locate/ejmsol

Non-local damage models in manufacturing simulations

Olufunminiyi Abiri*, Lars-Erik Lindgren

Luleå University of Technology, 97187 Luleå, Sweden

ARTICLE INFO

Article history:
Received 13 June 2014
Accepted 29 August 2014
Available online 13 October 2014

Keywords:
Finite element method
Non-local damage
Plasticity

ABSTRACT

Localisation of deformation is a problem in several manufacturing processes. Machining is an exception where it is a wanted feature. However, it is always a problem in finite element modelling of these processes due to mesh sensitivity of the computed results. The remedy is to incorporate a length scale into the numerical formulations in order to achieve convergent solutions. Different simplifications in the implementation of a non-local damage model are evaluated with respect to temporal and spatial discretisation to show the effect of different approximations on accuracy and convergence.

1. Introduction

Integrated microstructure and constitutive models are used in thermo-mechanical simulations of individual, e.g. (Börjesson and Lindgren, 2001; Svoboda et al., 2010) as well as chains of manufacturing processes (Lindgren et al., 2011a,b; Tersing et al., 2012). There are modelling challenges with respect to the material behaviour as well as friction conditions in many cases. However, there are also numerical problems requiring special precautions. The latter are mainly the need for handling extremely large deformation as well as localised deformations. Both issues occur in machining simulations (Svoboda et al., 2010). The focus of the current study is on the localisation problem.

There are two basic approaches to reduce the extreme mesh sensitivity when modelling localisation problems. In both cases, a length scale is introduced that enables the convergence of the solution by limiting the localisation of the deformation. This length scale can have a connection to the physics of the material behaviour but can also be seen as a numerical, regularisation parameter (Al-Rub and Voyiadjis, 2004; Al-Rub and Voyiadjis, 2006; Bazant and Jirásek, 2002; Enakoutsa et al., 2007; Geers et al., 2003). The two variants of including this length scale can be related either to non-local formulations or higher order continuum theory. An example of higher order continuum theory is the multiresolution continuum theory (MRCT) introduced by W.K Liu and co-workers (Lindgren et al., 2011a,b; Liu et al., 2009; McVeigh et al., 2006). It includes the Cosserat continuum, polar and micromorphic formulations (de Borst, 1991; Eringen and Suhubi, 1964; Forest and Sievert, 2006) as special cases. The current focus is on a simplified non-local formulation of damage models. The plastic behaviour is based on a standard plasticity model. The damage evolution is coupled to the plastic straining of the material.

The aim of the current work is to evaluate how different levels of implementation simplifications affect accuracy and efficiency. This is of particular concern when using implicit finite element formulations where one wants to take as large time steps as possible. This evaluation has been possible by using an in-house code. One point is to investigate the effect on accuracy of the non-local damage model by limiting use of non-local data relevant for a certain integration point to the data at the beginning of an increment, called an explicit non-local update (Cesar de Sa et al., 2010). This data are readily available during the iterative incremental solution of the finite element equations. It simplifies the implementation of the model via user routines in commercial finite element codes as well as reduces the nonlinearity in the solution process. The effects of various approximations in the consistent tangent matrix on convergence are also investigated in our paper. (Leblond et al., 1994; Tvergaard and Needleman, 1995) used a simplified local counterpart of the constitutive tangent matrix in their extension of Gurson's plasticity model (Gurson, 1977). They found that they had to take extremely small time steps. Neither did they include any comparison with using the exact tangent operator or the effect of longer time steps.

Furthermore, the non-local damage model is compared with a local damage model in order to evaluate the mesh sensitivity. Numerical results for tensile and shear deformations examples are used for this evaluation.

* Corresponding author.
E-mail addresses: oluabi@ltu.se (O. Abiri), lars-erik.lindgren@ltu.se (L.-E. Lindgren).

http://dx.doi.org/10.1016/j.euromechsol.2014.08.012
0997-7538/

Table 1
Investigation of effect of θ in Equation (15) for damage.

Length of time step [secs]	Average strain increment[a]	Peak force [kN]		Maximum ε_e^p before failure		Maximum ω before failure	
		$\theta = 0$	$\theta = 1$	$\theta = 0$	$\theta = 1$	$\theta = 0$	$\theta = 1$
1	0.085	8.87	8.77	0.07	0.07	0.09	0.10
0.5	0.021	8.77	8.67	0.08	0.10	0.15	0.21
0.25	0.005	8.68	8.61	0.11	0.10	0.26	0.21
0.125	0.001	8.61	8.58	0.11	0.10	0.24	0.22
0.0625	3.33e-4	8.58	8.55	0.10	0.10	0.22	0.21
0.03125	8.322-5	8.55	8.54	0.10	0.10	0.23	0.22
0.015625	2.08e-5	8.54	8.53	0.10	0.10	0.21	0.21
0.5-> 0.05	2.13e-4	8.58	8.55	0.10	0.10	0.22	0.22

[a] This is based on length of time steps combined with Equation (29) and assuming homogeneous deformation.

Table 2
Investigation of discretisation effect of using variable time stepping 0.5–0.05 secs.

Type of model	Peak force [kN]	Width of damage band[a]	Maximum ε_e^p before failure	Maximum ω before failure
20×80 local model	8.66	0.90 mm	0.18	0.46
20×80 explicit non-local update	8.67	2.10 mm	0.12	0.14
40×160 local	8.60	0.50 mm	0.22	0.51
40×160 explicit non-local update	8.56	1.90 mm	0.13	0.13

[a] Defined as region with effective plastic strain above 0.08.

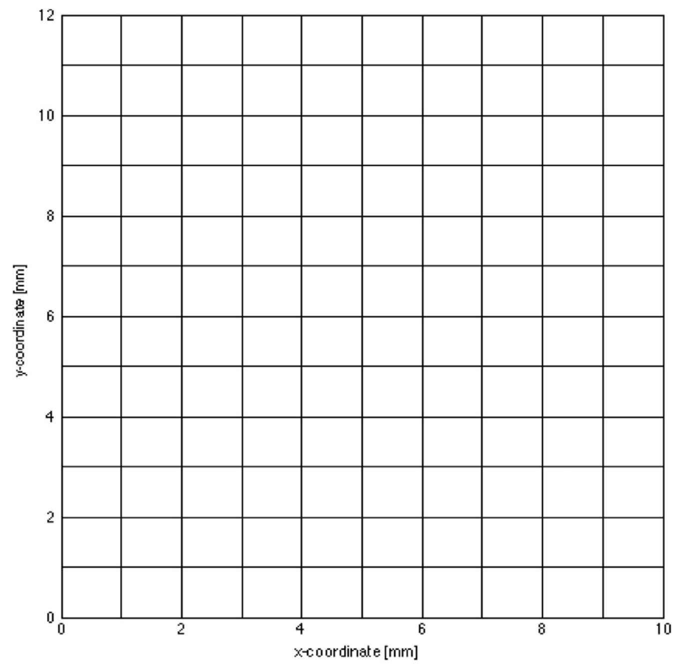

Fig. 2. The most coarse finite element mesh used in the shear simulations. It consists of 12 rows and 10 columns of four node elements, denoted 12x10 in the text. A quarter of the specimen is modelled. Symmetry conditions are applied to the left ($x = 0$) and bottom ($y = 0$). The centre of the specimen is made 2% thinner.

2. Non-local damage formulation

2.1. Coupled damage and plasticity

Ductile fracture is preceded by large-scale plastic yielding. Simulation of ductile fracture requires models that account for the simultaneous occurrence of plastic deformation coupled with damage. The use of a continuum mechanics approach can be based on the hypothesis strain equivalence (Simo and Ju, 1987) leading to a definition of effective stress $\bar{\sigma}$ as a transformation of the Cauchy stress σ as

$$\bar{\sigma} := M^{-1} : \sigma \tag{1}$$

where $\boldsymbol{M}$ is a fourth-order tensor, which characterize the damage state. The equation reduces to

$$\bar{\sigma} = \frac{\sigma}{1 - \omega} \tag{2}$$

for isotropic damage. Usually the material is assumed to have failed when it reaches a critical damage ω_c. Physically, the damage parameter ω can be interpreted as the ratio of damaged surface area or volume over the total surface area or volume at a local material point. The strain equivalence formulation model is combined with the assumption that damage affects elasticity, plasticity or visco-plasticity in the same way. This simplifies the modelling as all deformation of a damaged material is represented in the constitutive law of the virgin material by replacing the stress by effective stress.

2.2. Constitutive model

The strain-based approach is used and the change in effective Cauchy stress tensor $\bar{\sigma}$ is calculated as

$$\sigma^{\nabla} = (1 - \omega) C^e : d^e \tag{3}$$

where σ is the Cauchy stress tensor, C^e is the elastic material fourth order tensor, d^e is the elastic spatial velocity gradient and the right superscript ∇ denotes any objective stress rate. The effective stress used in the plasticity calculations is given by Equation (2). It is the used in the yield criterion

$$F = \bar{\sigma}_e - \sigma_y \tag{4}$$

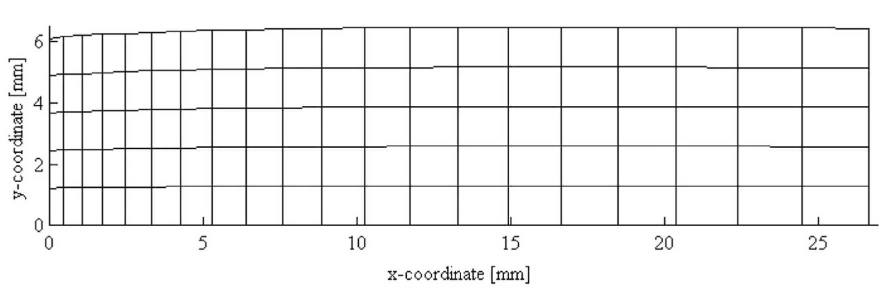

Fig. 1. The most coarse finite element mesh used in the tensile necking simulations. It consists of 5 rows and 20 columns of four node elements, denoted 5 × 20 in the text. A quarter of the specimen is modelled. Symmetry conditions are applied to the left ($x = 0$) and bottom ($y = 0$). The centre of the specimen is made 1.8% thinner.