Available online at www.sciencedirect.com
ScienceDirect
Computers and Structures 84 (2006) 2016–2032
Computers & Structures
www.elsevier.com/locate/compstruc

Evaluation of a finite element modelling approach for welded aluminium structures

T. Wang [a,*], O.S. Hopperstad [a], P.K. Larsen [a], O.-G. Lademo [b]

[a] *Structural Impact Laboratory (SIMLab), Department of Structural Engineering, Norwegian University of Science and Technology (NTNU), 7491 Trondheim, Norway*
[b] *SINTEF Materials and Chemistry, Applied Mechanics and Corrosion, 7465 Trondheim, Norway*

Received 15 July 2005; accepted 7 August 2006

Abstract

Experimental and numerical studies were carried out to investigate the behaviour of quasi-statically loaded fillet-welded connections in aluminium alloy EN AW 6082. The components were designed so that fracture should occur in the heat-affected-zone (HAZ), which was also the case in the experiments. In the numerical study, the components were modelled in LS-DYNA using shell elements. A user-defined material model comprising the Barlat and Lian anisotropic yield criterion and a ductile fracture criterion was adopted. The strength and hardening data for the HAZ, weld and base material were taken from existing experimental data in the literature. The constants for the yield criterion were identified from uniaxial tensile tests by various methods, and the set of constants that best represents the measurements was adopted in the numerical analysis. Reasonable estimates on ductility were obtained by the simulations when a refined mesh was used, while the strength was somewhat over-predicted.

Keywords: Aluminium alloy; Fillet welds; Heat affected zone; Plastic anisotropy; Failure; Finite element analysis

1. Introduction

Multi-stiffened aluminium panels made from welded extrusions are used in several applications where there is a demand for reduced structural weight, increased payload, and, in the case of transportation applications, higher speed and reduced fuel consumption. Some applications are high speed ferries and ship superstructures, living quarters on offshore installations, containers, plate girders and bridge decks. The competitiveness of such structures is primarily due to modern extrusion technology, new joining technologies and efficient manufacturing processes. Further market penetration depends on more efficient designs that fully utilize the structural capacity of such panels. However, at present this is impeded by the lack of suitable design rules in the ultimate limit state and in accidental load situations. The localization of deformation in poorly designed welded structural details may cause significant problems and a loss of structural integrity, particularly when subjected to tensile forces.

At present the welds are designed according to interaction formulas given in design codes. However, there is no unified approach to the problem of material failure in stiffened panels, which considers the mechanical properties of the welds and the heat-affected zones (HAZ), as well the stress concentrations caused by the extruded stiffeners. Sufficiently accurate and computationally feasible models are currently not available for the prediction of ductile failure.

In this study, the problem to be considered is an efficient, robust and sufficiently accurate finite element model applicable to welded multi-stiffened aluminium panels in the ultimate limit state and in accidental load situations. The objective of the present study is to carry out an investigation of the accuracy, efficiency and robustness of shell

* Corresponding author. Tel.: +47 92 64 77 76.
E-mail address: ting.wang@raufossneuman.com (T. Wang).

0045-7949/$ - see front matter
doi:10.1016/j.compstruc.2006.08.011

element simulations of a range of fillet-welded aluminium connections, including failure predictions, and eventually provide a validated FEM-based procedure for the analysis of welded multi-stiffened aluminium panels, which retains the safety level given by the design codes.

Experimental and numerical studies were carried out to investigate the quasi-static behaviour of some generic fillet-welded connections in the aluminium alloy EN AW 6082 T6 when loaded in tension. The components consist of two plates of different thickness and width, which are connected using double fillet welds. The welds are oriented at angles 0°, 30° and 45° relative to the extrusion and load directions. In order to characterize the plastic anisotropy of the base material, tension tests in three directions with respect to the extrusion direction were carried out. These tests were used to calibrate the constants of the Barlat and Lian anisotropic yield criterion [1] and to determine the strain hardening of the base material. The strength and hardening data of the weld and HAZ materials were obtained by utilizing data from a previous study by Matusiak [2]. A user-defined material model, called the Weak Texture Model, comprising the Barlat and Lian anisotropic yield criterion, has been developed and implemented in the finite element programme LS-DYNA [3,4]. Numerical analyses were carried out with LS-DYNA using shell elements and the Weak Texture Model, and the results were compared with the experimental data.

2. Constitutive model

The Weak Texture Model has previously been implemented in LS-DYNA by Lademo et al. [3,4] as a user-defined material model, and was validated as a suitable model for aluminium extrusions. The model adopts isotropic elasticity, the anisotropic yield criterion of Barlat and Lian, associated flow rule and nonlinear isotropic hardening. The constitutive equations of the model are presented in the following. Small strains and rotations are assumed in the presentation, while in the numerical implementation large rotations are accounted for in the co-rotational shell elements [5].

The strain tensor $\boldsymbol{\varepsilon}$ is decomposed into elastic and plastic parts [6]:

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}^{\mathrm{e}} + \boldsymbol{\varepsilon}^{\mathrm{p}} \tag{1}$$

where $\boldsymbol{\varepsilon}^{\mathrm{e}}$ and $\boldsymbol{\varepsilon}^{\mathrm{p}}$ are the elastic and plastic tensors, respectively. The relation between the stress tensor $\boldsymbol{\sigma}$ and the elastic strain tensor $\boldsymbol{\varepsilon}^{\mathrm{e}}$ is defined as

$$\boldsymbol{\sigma} = \mathbf{C} : \boldsymbol{\varepsilon}^{\mathrm{e}} = \mathbf{C} : (\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^{\mathrm{p}}) \tag{2}$$

where $\mathbf{C}$ is the tensor of elastic moduli.

The yield function f, which defines the elastic domain in stress space, is expressed in the form

$$f(\boldsymbol{\sigma}, \bar{\varepsilon}) = \bar{\sigma}(\boldsymbol{\sigma}) - \sigma_{\mathrm{Y}}(\bar{\varepsilon}) \leqslant 0 \tag{3}$$

where isotropic hardening has been assumed, $\bar{\sigma}$ is the effective stress, and σ_{Y} is the flow stress. Assuming plane stress state, the yield function of Barlat and Lian [1] can be written in the form:

$$\begin{aligned} \bar{\sigma} &= \sqrt[M]{\frac{1}{2}\left(a|K_1 + K_2|^M + a|K_1 - K_2|^M + c|2K_2|^M\right)} \\ K_1 &= \frac{\sigma_{xx} + h\sigma_{yy}}{2} \\ K_2 &= \sqrt{\left(\frac{\sigma_{xx} - h\sigma_{yy}}{2}\right)^2 + p^2\sigma_{xy}^2} \end{aligned} \tag{4}$$

where M is a material constant, and the coordinate axes are aligned with the principal axes of anisotropy. The weighting constants a, c, h, and p have been introduced to capture the anisotropic properties of the material, and are based on analyses by polycrystalline plasticity theory [1]. When M equals 2, and a, c, h, and p all equal unity, the criterion is identical to the von Mises criterion.

In the present study, the effective stress–strain relation is defined by an extended Voce law, i.e.,

$$\sigma_{\mathrm{Y}} = Y_0 + Q_1(1 - \exp(-C_1 \cdot \bar{\varepsilon})) + Q_2(1 - \exp(-C_2 \cdot \bar{\varepsilon})) \tag{5}$$

where Y_0, C_i and Q_i are material constants, and $\bar{\varepsilon}$ is the accumulated plastic strain.

The associated flow rule defines the evolution of the plastic strain as [6]

$$\dot{\boldsymbol{\varepsilon}}^{\mathrm{p}} = \dot{\lambda}\frac{\partial f}{\partial \boldsymbol{\sigma}} \tag{6}$$

where $\dot{\lambda} \geqslant 0$ is the plastic multiplier. The effective strain rate is defined as

$$\dot{\bar{\varepsilon}} = \frac{\boldsymbol{\sigma} : \dot{\boldsymbol{\varepsilon}}^{\mathrm{p}}}{\bar{\sigma}} = \dot{\lambda} \tag{7}$$

where the latter equality follows from Eq. (6) and that f is a positive homogeneous function of order one.

The loading/unloading conditions are written in the Kuhn–Tucker form [5]

$$f \leqslant 0; \quad \dot{\lambda} \geqslant 0; \quad \dot{\lambda} f = 0 \tag{8}$$

Eq. (8) is used to define plastic loading and elastic unloading, while the consistency condition, $\dot{f} = 0$, is utilized to determine the plastic multiplier $\dot{\lambda}$ during a plastic process.

Fracture is modelled by eroding elements when a fracture criterion is satisfied for an optional number of integration points within the element. For this purpose, a critical thickness-strain criterion was introduced in the material model. The criterion assumes that fracture occurs as the thickness strain reaches a critical value $\varepsilon_{3\mathrm{f}}$, i.e.,

$$\varepsilon_3 = \varepsilon_{3\mathrm{f}} \tag{9}$$

where it is noted that $\varepsilon_{3\mathrm{f}}$ is a negative number. Owing to plastic incompressibility, this criterion can in the case of negligible elastic strains be expressed as

$$\varepsilon_1 + \varepsilon_2 = -\varepsilon_{3\mathrm{f}} \tag{10}$$