Composites: Part A 40 (2009) 1231–1239

Contents lists available at ScienceDirect

Composites: Part A

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

Multi-step 3-D finite element modeling of subsurface damage in machining particulate reinforced metal matrix composites

Chinmaya R. Dandekar, Yung C. Shin *

Center for Laser-based Manufacturing, School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, United States

ARTICLE INFO

Article history:
Received 21 October 2008
Received in revised form 27 April 2009
Accepted 18 May 2009

Keywords:
C. Finite element analysis
Damage prediction
E. Machining
A. Metal matrix composites

ABSTRACT

A multi-step 3-D finite element model using the commercial finite element packages Third Wave Systems AdvantEdge© and ABAQUS/Explicit© is developed for predicting the sub-surface damage after machining of particle reinforced metal matrix composites. The composite material considered for this study is an A359 aluminum matrix composite reinforced with 20 vol% fraction silicon carbide particles (A359/SiC/20p). The effect of machining conditions on the measured cutting force and damage is modeled by means of a multi-step fully-coupled thermo-mechanical model. Material properties are defined by applying the Equivalent Homogenous Material (EHM) model for the machining simulation while the damage prediction is attained by applying the resulting stress and temperature distribution to a multi-phase sub-model. In the multi-phase approach the particles and matrix are modeled as continuum elements with isotropic properties separated by a layer of cohesive zone elements representing the interfacial layer to simulate the extent of particle–matrix debonding and subsequent sub-surface damage. A random particle dispersion algorithm is applied for the random distribution of the particles in the composite. Experimental measurements of the cutting forces and the sub-surface damage are compared with simulation results, showing promising results.

1. Introduction

Metal matrix composites offer high strength to weight ratio, high stiffness and good damage resistance over a wide range of operating conditions, making them an attractive option in using these materials for structural applications. Particulate reinforced composites offer higher ductility and their isotropic nature as compared to fiber reinforced composites makes them an attractive alternative. Popular reinforcement materials for these composites are silicon carbide and alumina particles, while aluminum, titanium and magnesium constitute as the most common matrix materials. Although these composites are generally processed near net shape, subsequent machining operations are inevitable. The inherent challenge in machining of these composites is the excessive tool wear and the subsequent damage in the material sub-surface. This paper deals with a multi-step 3-D finite element modeling approach for predicting the sub-surface damage of machined particulate reinforced composites.

Machining of particulate reinforced metal matrix composites has been extensively studied experimentally [1–5] and numerically [6–10] in the past to assess the attendant tool wear, surface roughness, sub-surface damage and to predict cutting forces. Silicon carbide particle reinforced aluminum matrix composites (SiC_p/Al) have been the most popular amongst these studies, where the primary focus of experimental data was on the characterization of the excessive tool wear caused by the highly abrasive silicon carbide (SiC) particles, as severe tool wear usually resulted in the creation and progression of surface and sub-surface flaws.

In terms of modeling machining of metal matrix composites (MMC), following strategies have been employed; (a) a micromechanics based approach (b) an equivalent homogeneous material (EHM) based approach and (c) a combination of the two approaches. The micromechanics and the equivalent homogenous material (EHM) based approaches have their respective advantages and disadvantages [11,12]. The micromechanics approach describes the material behavior locally, and hence it is possible to study local defects such as debonding. However it is computationally intensive and hence cannot be used for large scale deformation simulations. On the other hand the EHM approach loses the ability to predict the local effects, namely, the damage observed at the interface separating the two phases [11], while it is more computationally efficient for machining simulations. Therefore there is a need to harness the advantages of both the continuum and micromechanics models in their capability of predicting cutting forces and sub-surface damage.

A number of attempts have been made in modeling machining of MMCs [7–10]. Although the results reported in these works seem reasonable, these studies have primarily focused on 2-D modeling of orthogonal cutting which is not realistic for actual

* Corresponding author. Tel.: +1 765 494 9775; fax: +1 765 494 0539.
E-mail address: shin@purdue.edu (Y.C. Shin).

1359-835X/$ - see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compositesa.2009.05.017

machining. The modeling work has been focused on studying the failure at particle–matrix interface [7], the tool–particle interaction [8,9] and residual stresses with sub-surface damage [7,10]. The studies focused on predicting sub-surface damage have so far lacked in their representation of the interface, since the particles are considered to be perfectly bonded to the matrix.

It is clear that the presence of reinforcement makes MMCs different from monolithic materials due to incorporation of superior physical properties into the MMC [13]. On the other hand, these reinforcement particles are responsible for very high tool wear and inferior surface finish when machining MMCs. Typical sizes of reinforcing particles are in the order of 10–30 μm diameter with reinforcements ranging from 5 to 30 vol%. Understanding the various failure mechanisms during machining of MMC's is crucial in formulating a valid and representative machining model. Li et al. [14] characterized the different failure mechanisms during dynamic loading of MMC's: (i) cracking of the reinforcing particles; (ii) partial debonding at the particle/matrix interface resulting in the nucleation of voids and (iii) the growth and coalescence of voids in the matrix. A good FEM model would be able to incorporate all the above failure modes based on the stress/strain state prevalent at the time of loading. This can be achieved by having a number of failure criteria by incorporating user defined material parameters for the different phases of the particle and matrix. The interface can be modeled using a cohesive zone model [11,15] to facilitate in capturing the interface mechanics. Prediction of damage based on either analytical or numerical studies helps in better design of tool geometry and selection of cutting parameters. In this paper, 3-D numerical simulations are conducted and compared with experimental measurements of cutting force and sub-surface damage for an A359/SiC/20p composite.

2. Modeling of 3-D machining of particulate reinforced metal matrix composites

The multi-step approach utilizes a two step approach to predicting the behavior of composites during machining. In the first step an EHM model is used for the overall prediction of cutting forces, temperature and the stress distributions in the composite undergoing machining. The second step then involves applying the predicted stress and temperature distributions to a local three phase finite element mesh. The mesh is based on distinct properties of the particulate, matrix and particulate–matrix interface. The particulate is modeled as a linearly elastic isotropic material until failure. The matrix material on the other hand is considered as thermo-elastic–plastic and isotropic in nature while the particulate–matrix interface is modeled using cohesive zone elements. The continuum elements account for the deformation present in the particulate and matrix, while the cohesive elements account for the occurrence of debonding.

2.1. Matrix material modeling

For modeling initiation and progression of damage in the A359 aluminum matrix, a thermo-elastic–plastic material model with isotropic and kinematic hardening is coded in FORTRAN in the form of a user material (VUMAT). The model is suitable for simulating processes involving high adiabatic shear localizations as observed in metal matrix composite machining.

The first step in the trial stress radial return method for thermo-elastic–plastic material implementation is calculating the trial stress [16,17]. The strain can be segregated into two parts; the elastic and the plastic part as given in Eq. (1) in indicial notation. The trial stress is then calculated from Eq. (2), where δ_{ij} is the usual notation for the Kronecker–Delta symbol, and λ and μ are the Lame's constants. K is the bulk modulus, α is the coefficient of thermal expansion, T is the material temperature and T_r is the reference temperature.

$$\varepsilon_{ij} = \varepsilon_{ij}^{e} + \varepsilon_{ij}^{p} \tag{1}$$

$$\sigma_{ij} = \lambda\varepsilon_{kk}\delta_{ij} + 2\mu\varepsilon_{ij} - beta(T - T_r)\delta_{ij} \tag{2}$$

$$\beta = 3\alpha K \tag{3}$$

On determining the trial stress from Eq. (2), back stress (x) due to kinematic hardening and the deviatoric part of the back stress $(\sigma - x)'$ are obtained. The effective stress (σ_e) as per the von Mises criterion is then calculated using Eq. (4), where ':' denotes the double contracted dot product, of two second order tensors.

$$\sigma_e = \left[\frac{3}{2}(\sigma - x)' : (\sigma - x)'\right]^{\frac{1}{2}} \tag{4}$$

The plasticity is then checked by Eq. (5), where f is the yield function.

$$f = \sigma_e - R - \sigma_y \tag{5}$$

where R is the isotropic hardening parameter and σ_y is the yield stress. Eq. (6) is then used to calculate the plastic multiplier

$$\dot{\lambda} = \frac{f}{3G + H} \tag{6}$$

and the increment in the plastic strain is given by

$$\dot{\varepsilon}_p = \frac{3(\sigma' - x)}{2\sigma_e}\frac{\dot{\lambda}}{} \tag{7}$$

At the end of the time step the elastic/plastic strains along with the stress matrix are updated. The material is considered damaged once the equivalent plastic strain exceeds the plastic strain allowable by the aluminum matrix for all material integration points.

2.2. 3-D cohesive zone modeling

Modeling of the interface between the particle and the matrix is achieved using cohesive zone elements. The cohesive zone model (CZM) has been successfully implemented in machining of ceramics [18] and predicting debonding at the fiber–matrix interface in machining of fiber reinforced composites [11]. A 3-D cohesive zone model has been developed for studying the damage during machining of the metal matrix composite. The cohesive model describes a relationship between the interfacial force and the crack opening displacement. In the CZM, the fracture process zone is simplified as being an initially zero-thickness zone, composed of two coinciding cohesive surfaces. Under loading, the two surfaces separate and the traction between them varies with separation distance according to a specified traction separation law. The cohesive element progressively degrades in stiffness as interfacial separation increases. When the opening displacement reaches the prescribed maximum, the cohesive element fails, suggesting separation and debonding of the interface. The crack propagation between the continuum elements progresses along the boundary. This feature of cohesive elements allows one to simulate debonding at the particle–matrix interface. The cohesive response addressed in the model here is based on Tvergaard's assumed traction separation law [15] and applied by Foulk et al. [19] in studying the 3-D response of a SCS-6Trimetal21S[O]$_4$ metal matrix composite to simulation fiber–matrix debonding. The cohesive equations necessary for defining the model are given below in Eqs. (8)–(14). The non-dimensional parameter (ξ) in Eq. (8) relates the normal (u_n) and tangential (u_t *and* u_s) separation to the maximum allowable normal (δ_n) and tangential (δ_t δ_s) separation of the cohesive element and hence accounts for the damage of the cohesive element. The cohesive element then fails when the value of ξ reaches 1.