ELSEVIER

Journal of Materials Processing Technology 113 (2001) 368–372

Journal of
Materials
Processing
Technology

www.elsevier.com/locate/jmatprotec

An adaptive three-dimensional finite element algorithm for the orthogonal cutting of composite materials

Mofid Mahdi, Liangchi Zhang*

Department of Mechanical and Mechatronic Engineering, The University of Sydney, Sydney, NSW 2006, Australia

Abstract

In order to analyze the mechanism of cutting a fiber-reinforced composite using the finite element method (FEM) an appropriate meshing for the matrix, the fibers and their interfaces is essential. In a two-dimensional FEM analysis, the uniform increase of elements remains an attractive way with reasonable cost. However, this is not true in three-dimensional cases as the degrees of freedom rise dramatically. This paper aims at an efficient way to achieve smooth material removal in three-dimensional cuttings of fiber-reinforced composites. An adaptive algorithm was developed to control the solution steps and input data and to handle the remeshing of the selected elements using the criterion of the maximum shear stress. Different sets of three-dimensional elements were used to account for the fiber and matrix materials. At each solution step, the algorithm examined all sets of elements, determined those with the maximum shear stress and replaced them with refined meshes such that interconnectivity of the new elements satisfies the compatibility conditions. The solution of the previous time step was then mapped to the new mesh as the initial conditions. The chip formation was realized by the technique of element death. It was found that although the use of an algorithm increases the computational time, it offers a more cost-effective finite element mesh than the conventional method of refinement. In addition, the algorithm can accommodate the behavior of a fiber-reinforced composite under general cutting conditions. The numerical result shows a close agreement with the relevant experimental results. © 2001 Elsevier Science B.V. All rights reserved.

Keywords: Remeshing algorithm; Finite element method (FEM); Orthogonal cutting; Composite

1. Introduction

A cumbersome aspect of obtaining a finite element solution to engineering problems is the preparation of appropriate input data. Usually, the input data require a description of the element mesh topology. Bathe [1] discussed the advanced meshing issues and commented on the current state of meshing procedures. Dolsak and Jezernik [2] introduced a logic-programming algorithm to build a knowledge base for an expert system of an FEM mesh generation. Radim [3] considered an adaptive incremental FE method that makes use of composite grids. Kjell et al. [4] coupled a module for error estimation with the adaptive solution procedure in the explicit FE code LS-DYNA and thereby provided a selective refinement of the mesh of high local error. Wanga et al. [5] developed a node-based data structure for either two-dimensional (2D) or three-dimensional (3D) finite element solutions, which uses quadrilateral (2D) or hexahedral (3D) elements and is dynamically adaptive. Although, the node based approach may increase the CPU time and complexity to access an element, it reduces the complexity and time required to refine and de-refine an element. David and Stefka [6] proposed the so-called Nibble algorithm for the triangulation of a non-manifold solid boundary. The algorithm is based on an incremental boundary traversal technique. At each step of the algorithm, a surface boundary called active boundary is evaluated in such a way that it nibbles the surface to be triangulated. Lawrencea et al. [7] employed an adaptive procedure to refine a crude initial mesh on the basis of solution error indicators or other error measures. The refinement procedure strictly maintains the integrity of the geometry by placing newly generated nodes on the true boundary of the solid and also maintains the physical conditions of the problem such as loads and boundary conditions. Bruce and Karen [8] reviewed the major algorithms associated with adaptive mesh refinement, particle simulations and transient dynamics calculations in parallel computation, and proposed a number of approaches for the computational balancing problem.

Since all the above element-refining techniques will become ineffective for analyzing microscopically the cutting of fiber-reinforced composite, the present study is to develop an adaptive algorithm to overcome the numerical difficulties encountered during the cutting simulation of a composite cell.

---

[*] Corresponding author. Tel.: +61-2-9351-2835; fax: +61-2-9351-7060.
E-mail address: zhang@mech.eng.usyd.edu.au (L. Zhang).

PII: S0924-0136(01)00676-8

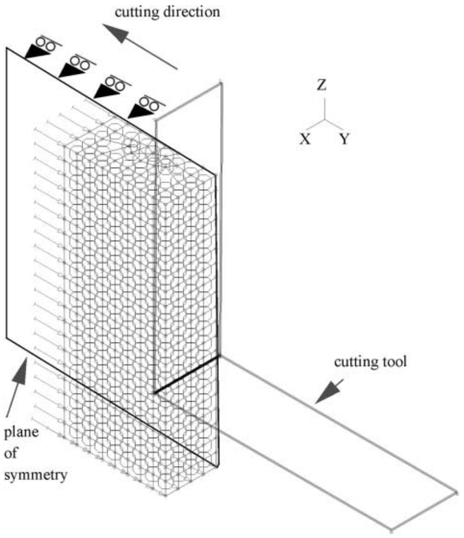

Fig. 1. Loading conditions. Mesh 1: 10 layers, mesh 2: 15 layers, mesh 3: 20 layers.

2. FEA model

A composite cell composed of matrix and a single fiber is proposed to approximate the real cutting process, as shown in Fig. 1. The problem studied is therefore symmetric about the y–z plane and only half of the cell needs to be considered in the analysis, as illustrated in Fig. 2. The matrix and fiber materials are elastic and the fiber–matrix interface is assumed to be perfect. The modulus of elasticity of the matrix and fiber are 237 GPa and 2.96 MPa, respectively and their Poisson's ratios are taken as 0.3 for fiber and 0.35 for matrix. The process of cutting is considered as quasi-static with negligible tool–workpiece friction. The selection of control volume, mesh size, material removal, contact modeling and adaptive remeshing are discussed below.

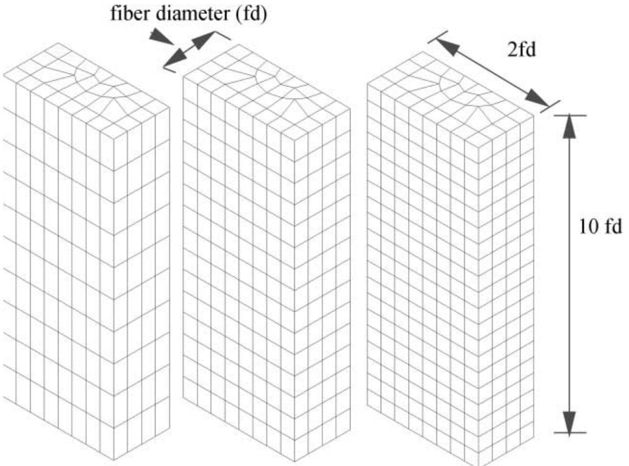

Fig. 2. The composite cell.

2.1. Control volume and initial mesh

Some simulation difficulties arise from the improper selection of mesh size and control volume. To understand the effect of mesh size on the cutting force prediction, three meshes are used, as shown in Fig. 2, which makes use of the symmetry conditions of the composite cell, i.e. the normal displacement in the x-direction vanishes. The cutting tool is stationary and the workpiece is forced to move in the positive y-direction as indicated by Fig. 1. The depth of the control volume is 1.6 times the depth of the cut, which is 50 μm. The fiber diameter is 8 μm. The control volume shown in Fig. 2 is $16\,\mu\mathrm{m}$ (y-direction) $\times$ $8\,\mu\mathrm{m}$ (x-direction) $\times$ $80\,\mu\mathrm{m}$ (z-direction). Three FEM meshes with 10, 15 and 20 layers of 8-node elements are used to study the effect of mesh size on the cutting force prediction, as shown in Fig. 2. In each layer, 8 and 24 elements are used to represent fiber and matrix, respectively. In the third mesh, the initial total number of nodal points and elements were 945 and 640, respectively. Using ADINA 7.3 FEM software [9], the analysis is carried out under Unix operating system.

2.2. Contact simulation and material removal

Physically, the cutting process involves wedging, sliding and chipping under material deformation and hence a proper treatment of the contact between the cutting tool and the workpiece is critical for a high rate of convergence of the calculations. The segment method [9] is therefore applied because this method employs an extra inner iteration for contact simulation. The cutting tool consisting of contact segments is assumed to be rigid. The workpiece, on the other hand, is considered as a contact body with contact nodes. The interaction between the rigid body and the contact body nodes will result in nodal reaction forces so that the total cutting force can be calculated by the summation of the consistent contact forces associated with cutting tool at each time step.

To imitate the chip formation, a material removal mechanism needs to be described. In contrast to the usual definition of detachment, layer of elements or lines of twin nodes in the metal cutting simulation, here a material separation is achieved through an arbitrary element removal (or rupture) criterion. The maximum shear stress with a given threshold is employed to enable element death (rupture). This is also proper for an FEM formulation which does not permit highly sheared elements (with negative or small Jacobian determinant) due to shear stress effect. Here, the elements are removed if the local maximum shear stress reaches 1.3 GPa and 30 MPa for fiber and matrix elements, respectively.

2.3. Adaptive remeshing

In a cutting operation, the large shear deformation of the material results in the distortion of the elements. This can

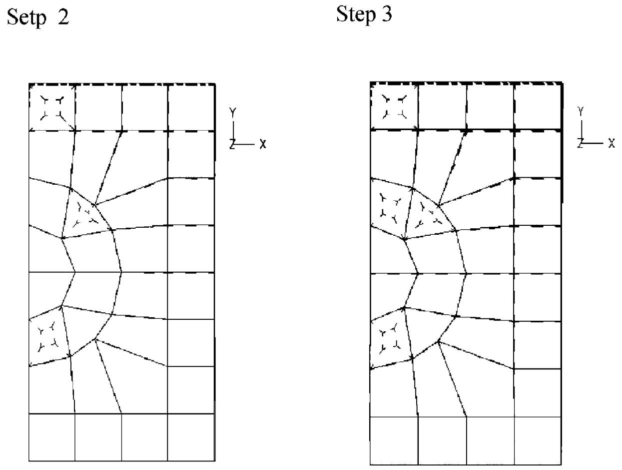

Fig. 3. Remeshing in the composite cell.

cause a deterioration of the FEM simulation in terms of convergence rate and numerical errors. To overcome this dilemma, an adaptive remeshing procedure for selected elements with a prescribed criterion is employed. The remeshing is characterized by: (i) the remeshing criterion using the maximum shear stress; (ii) a mesh generator; (iii) a data transfer operator that carries out a mapping of history dependent variables from the old mesh to the new mesh. Fig. 3 is an example that demonstrates a typical remeshing process in two solution steps. When the maximum shear stress reaches the pre-set threshold, an element refining takes place, i.e., smaller size elements are substituted in the element. For this typical study, each local refinement results in an addition of six 8-node elements and one node. The progress of element addition during mesh refinement is illustrated in Fig. 4. The addition of six elements is accompanied by the removal of the parent element, thus the net of five are increased. Since any refinement increases computational time in the assembly of the global stiffness matrix and the solution of FEM equations, the above method of local mesh refinement is a less expensive procedure for adaptive remeshing. After the refinement, strains and displacements are mapped to the new mesh as the initial conditions for the next calculation. Fig. 5 outlines the general feature of the above adaptive technique.

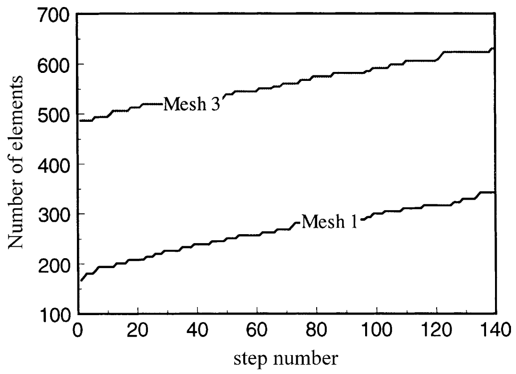

Fig. 4. Total number of elements and remeshing.

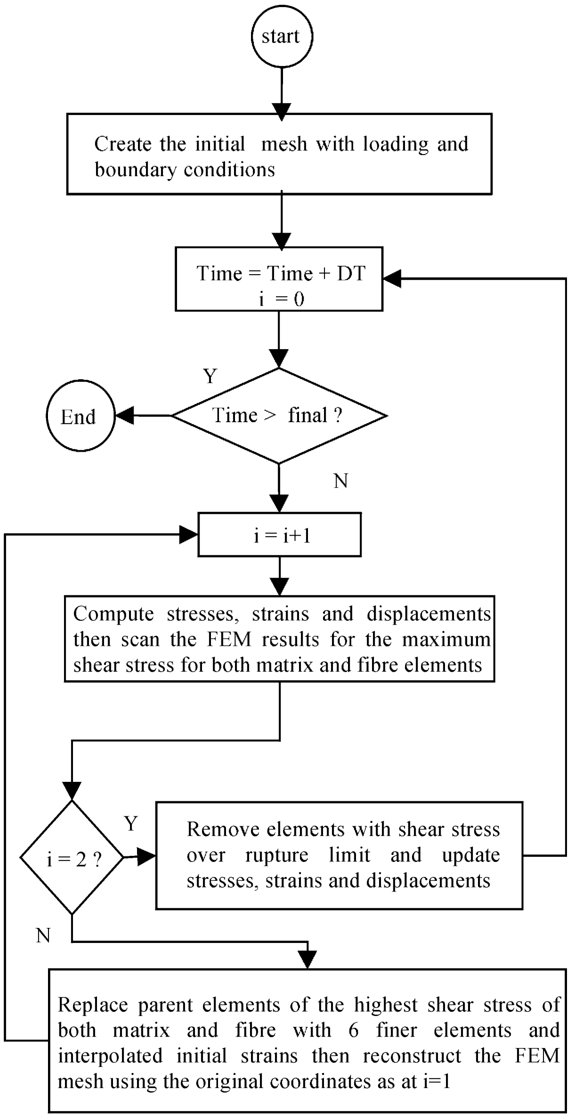

Fig. 5. The adaptive algorithm of remeshing for cutting simulation.

3. Results and discussion

The method introduced above has been used to simulate a three-dimensional cutting process. Fig. 6 illustrates the history of deformation at the given solution steps. It shows clearly that the material removal mechanism in cutting is complex. The elements below the depth of cut are often intact, while those within the cutting depth that are subjected to high pressure and shearing are highly distorted or removed. It is also clear that some elements are refined before removal. This indicates that the time step of mesh refining should be reduced to achieve a smoother element removal process. Fig. 7 shows the effect of adaptive mesh intensity on the accuracy of the horizontal cutting force prediction. It is understandable that

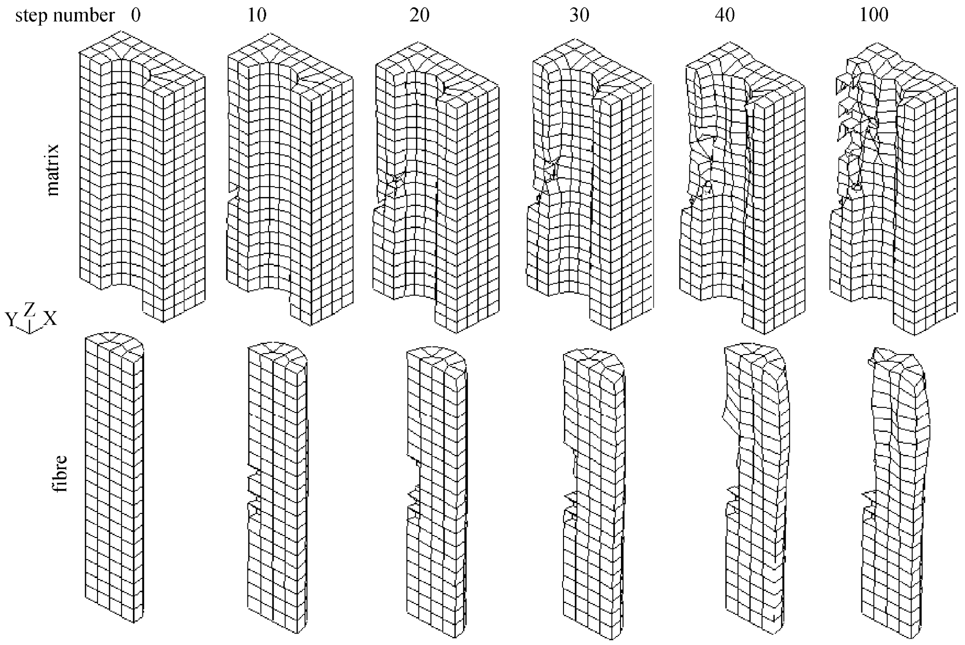

Fig. 6. The progress of deformation in the composite cell.

the cutting force increases when the cutting tool starts to interact with the workpiece and then decreases as some elements in the cutting zone are removed. The mesh intensity plays a central role in the prediction of cutting forces. The finer mesh results in a better agreement with corresponding measurement, as shown in Fig. 8, the experimental details of which were described and discussed by Wang and Zhang [10]. On the other hand, the effect of adaptive and non-adaptive algorithms on cutting force prediction is also clear. As shown in Fig. 9, the FEM model without remeshing results in much poorer cutting forces, since the forces decrease rather than increase to approach to the expected value. Hence, the

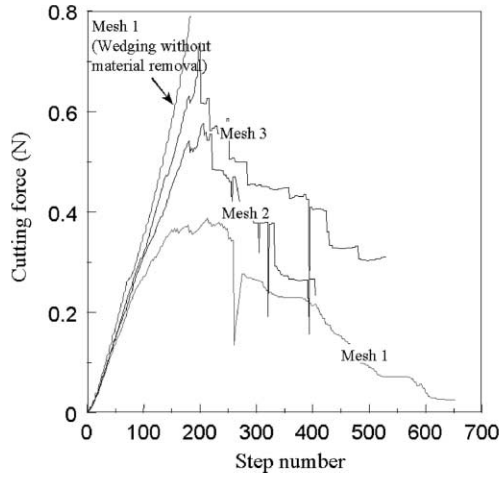

Fig. 7. The effect of mesh intensity on the cutting force prediction.

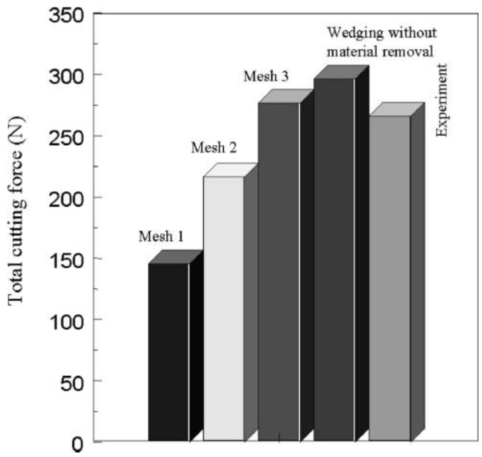

Fig. 8. The effect of remeshing.

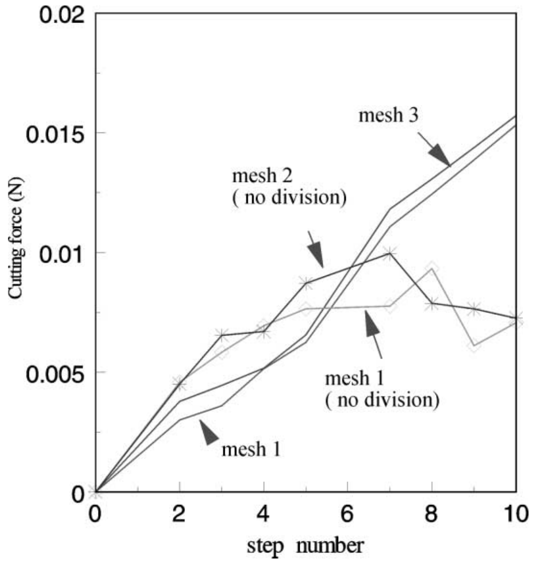

Fig. 9. The effect of the adaptive algorithm.

level of the cutting force predicted by the adaptive remeshing technique shows increasing accuracy, whereas those with the non-adaptive mesh cannot do so.

4. Conclusions

An adaptive finite element algorithm for local remeshing was proposed for simulating the cutting of fiber-reinforced composite. It is found that the algorithm is efficient and that the maximum shear stress criterion is appropriate for cutting simulations.

Acknowledgements

The financial support from ARC to the present study is appreciated. ADINA code was used for all the calculations.

References

[1] K.-J. Bathe, Mech. Eng. 120 (7) (1998) 70.
[2] B. Dolsak, A. Jezernik, Comput. Ind. 17 (2–3) (1991) 309.
[3] B. Radim, Math. Comput. Simulat. 50 (1–4) (1999) 123.
[4] M.M. Kjell, S.H. Odd, M.O. Knut, B. Torodd, Comput. Struct. 72 (4–5) (1999) 627.
[5] G.H. Wanga, J.M. Tylerb, J.S. Weltmanc, J.D. Callahand, Adv. Eng. Software 30 (1) (1999) 31.
[6] M. David, G. Stefka, Comput. Graphics 22 (2–3) (1999) 181.
[7] K.L. Lawrencea, S.N. Muthukrishnana, R.V. Nambiarb, Computer-aided Des. 27 (8) (1995) 637.
[8] H. Bruce, D. Karen, Comput. Meth. Appl. Mech. Eng. 184 (2–4) (2000) 485.
[9] Theory and modeling guide, ADINA Report ARD97, Vol. 9, ADINA, 1997
[10] X. Wang, L.C. Zhang, in: J. Wang, W. Scott, L.C. Zhang (Eds.), Abrasive Technology: Current Development and Applications I, World Scientific, Singapore, 1999, p. 429.