A Failure Modeling for the Interfaces with Irregular Shape and its Numerical Simulation

X.R. NIU[1], Y.L. KANG[2], S.W. YU[1], X.Q. FENG[1]

Summary

Based on real microstructure geometry, a microstructure made of a metal-matrix composite (MMC) is evaluated via nonlinear finite element calculation. A user-defined element (UEL) subroutine has been implemented in the commercial finite element code ABAQUS for modeling the interfacial damage between matrix and reinforcement. An applicable non-continuum four-node interface element is proposed and formulated to investigate the interfacial failure for a typical MMC-AL/Al_2O_3 composite. The computational simulation shows that the interfacial failure in the microstructure mainly depends on the parameters controlling the interfacial behaviors, namely the interfacial strength and the interfacial stiffness. In addition, the numerical results for the microstructure with irregular shape inclusions indicate that the failure initiates at the interface along the inclusion where the edge curvature of the inclusion changes sharply.

Introduction

The rapid development of materials science promotes increasingly extensive engineering application of advanced materials such as fiber-reinforced composites, which as a rule include many interfaces. Therefore the mechanical behaviors of interfaces have a strong influence on the mechanical properties of the materials, such as their strength and toughness.

Some Experimental stress analyses have been carried out for the microstructure with the interface shown in Fig. 1 by Fischer [1,2]. In this method, modeled on the real microstructure of the MMC, the material properties are identical to the actual ones were adopted in the numerical simulation. Then the edge displacements of a typical micro-region as obtained from the experiment were used as input for the FE-calculation to predict the displacement, strain and stress fields within this region. The hybrid method has been used to discuss the deformation pattern of a typical microstructure under plane strain condition in Ref. [2]. Furthermore, Kang *et al.* [3] gave the strain and stress distributions of this microstructure under plane stress and plane strain conditions respectively by using ANSYS finite element code. However, the damage of the interface in this microstructure could not be simulated in Refs. [2] and [3] owing to the reason that a conventional FEM was used in the calculations. Soppa and Schmauder *et al.* in Ref. [4] have calculated the damage near the interface by using Rice-Tracey damage model of void growth under plan strain state.

The interfaces of the MMC are a transition region where the material behaviors change very quickly and inevitably exist some initial defects. In view of this, another special non-continuum interface element has been proposed for the simulation of the interfacial strength and its failure. Achenbach [5] and Chen [6] modeled the interfacial layer as a set of orthogonal springs which are in normal or tangential direction, thus the failure of the interface could be controlled by the maximum stress or strain of these springs. An isotropic element with zero thickness was used to simulate the mechanical behaviors of the interphase by Ye *et al.* [7] when they analyzed the stress distribution around the fiber. Allix [8] also applied the interface element similar to the above to the study of the delaminating of a laminate and an elastic damage criterion was introduced.

[1]Department of Engineering Mechanics, Tsinghua University, Beijing 100084, China
[2]Department of Mechanics, Tianjin University, Tianjin, 300072, China. *Email address*: ylkang@public.tpt.tj.cn (Y.L. Kang)

The investigations mentioned above have overcome the difficulty in determining the thickness of interfaces when meshing in FE-calculations, and some of them have given the interfacial failure criterions. Still they couldn't reflect the coupling effect between the normal direction and the tangential direction of the interfacial springs when the interfacial failure took place. The cohesive zone model was first employed in the interface analysis of materials by Needleman [9,10] who introduced an interfacial potential and described the interface constitutive relation by this potential. Tvergaard [11] gave up the interfacial potential when using the cohesive zone model, and established the relation between the interfacial adhesive traction and the interfacial displacement jump directly. A similar model has also been adopted by Zhou [12] in his research on the damage and debonding of the interface between fiber and matrix in a composite. The influence of some relevant factors on the damage of the interface was analyzed in the paper.

In this paper, a non-continuum four-node interface element is proposed and coded to investigate the deformation and damage process of a microstructure in Fig 1[1]. We make an attempt to numerically simulate the interfacial failure behaviors and its strength for a real microstructure. First several interface elements with dissimilar interfacial parameters are chosen to incorporate in the FE-analysis, and then we can investigate the damage and failure mechanism of the interface and predict a proper interface model when comparing the results of these simulations with those of the experiment [1]. This procedure can be called a semi-inverse methodology, one of the tools for the analysis and the optimum design of interfaces in composite.

Non-continuum interface damage element with four nodes

In this paper, a simple interface element is developed, which is convenient for the study of the damage behaviors of complex interfaces with irregular edges. The element is coded within the interface for the user-defined element (UEL) subroutine provided by a commercial finite element program, ABAQUS. It is a two-dimensional element with four nodes and is of zero width in the normal direction of the interface initially, as show in Fig. 2. Node 1 coincides with node 4 and node 2 coincides with node 3 before deformation. Each node has two displacement degrees of freedom, which are the translations in either x-direction or n-direction.

The springlike interaction of two plane continuum elements linked by an interface element can be simulated by this interface element. The stress–displacement jump relations for the interface element are described by the curves shown in Fig 3, where s_{t} and s_{n} are the tangential and nomal stresses respectivity. With increasing interfacial displacement jump from zero, the stress across the interface increases from zero too, and the interface is "elastic" initially. After the stress reaches a maximum, it decreases and eventually vanishes permitting a complete decohesion, when the interfacial failure occurs. Such interfacial constitutive relations enable us to simulate the interfacial deformation and failure mechanism by using this interface element. The stress–displacement jump curves characterizing the mechanical behaviors of interfaces are piecewise linear in calculations to approximately represent the really smooth curves. In this paper, 262 points are chosen from the positive part of the real stress-displacement jump curve to serve as intersections of the line segments in the piecewise linear curve. The shape of the constitutive cuves is determined by that of a typical curve, still the height and span of these curves either in tangential or normal direction can change independently. It means that the four parameters, namely d_{T}, d_{N}, s_{Tmax} and s_{Nmax} can change independently. The interfacial linking model can be imaged as nomal-tangential coupling nonlinear spring elements with a softening damag stage. The interfacial behaviors can be characterized by the four parameters as mentioned above, and the element can be used to study the deformation process of interfaces and the interfacial failure.

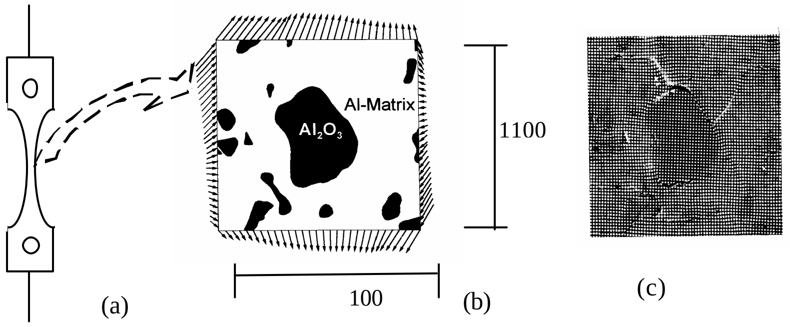


Fig. 1. Experimental specimen and a local microstructure of MMC Al-10%Al_2O_3: (a) Al-10%Al_2O_3 composite specimen, (b) microstructure with experimentally displacements, and (c) a bitmap of the deformed microstructure.

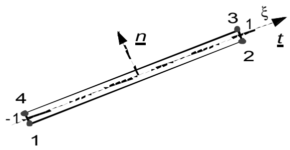


Fig.2 Local coordinates of element

The column vector of the nodal displacements of the interface element is

$$\boldsymbol{a}^{e} = [u_1, v_1, u_2, v_2, u_3, v_3, u_4, v_4]^{\mathrm{T}} \tag{1}$$

The tangential and normal displacement jumps inside the element are given by

$$\boldsymbol{\Delta}(x) = \begin{Bmatrix} \Delta_{\mathrm{t}}(x) \\ \Delta_{\mathrm{n}}(x) \end{Bmatrix} = \begin{Bmatrix} u_{\mathrm{up}}^{\mathrm{loc}} - u_{\mathrm{down}}^{\mathrm{loc}} \\ v_{\mathrm{up}}^{\mathrm{loc}} - v_{\mathrm{down}}^{\mathrm{loc}} \end{Bmatrix} = \boldsymbol{L} \cdot \begin{Bmatrix} u_{\mathrm{up}}^{\mathrm{loc}}(x) \\ u_{\mathrm{down}}^{\mathrm{loc}}(x) \\ v_{\mathrm{up}}^{\mathrm{loc}}(x) \\ v_{\mathrm{down}}^{\mathrm{loc}}(x) \end{Bmatrix} = \boldsymbol{L} \cdot \boldsymbol{N}(x) \cdot \boldsymbol{a}^{e\ \mathrm{loc}} \tag{2}$$

The detailed formula can be found in [15].

The stresses in the element are linked with the displacement jumps by the material matrix $[D]$

$$s = \begin{Bmatrix} s_{\mathrm{t}} \\ s_{\mathrm{n}} \end{Bmatrix} = [D] \begin{Bmatrix} \Delta_{\mathrm{t}} \\ \Delta_{\mathrm{n}} \end{Bmatrix}, \tag{3}$$

where the material matrix $[D]$ is determined by the constitutive curves of the interface as shown in Fig. 3.

The stiffness matrix of the element is calculated by

$$[K^{e}] = \int_{-L}^{L} [B]^{T} [D][B] dl = \int_{-1}^{1} [B]^{T} [D][B] L dx \,. \tag{4}$$

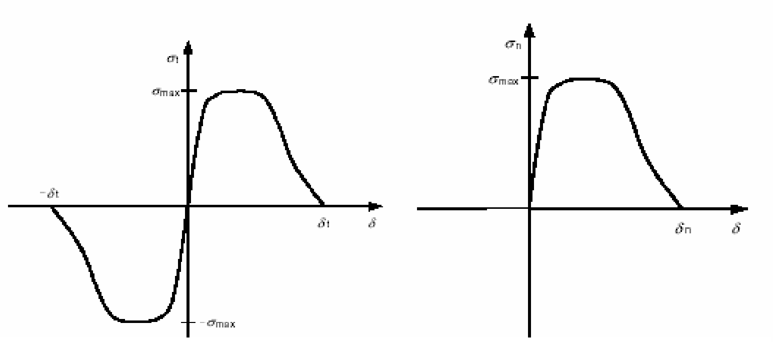
The nodal force vector of the element is given by

$$\{F^{e}\} = \int_{-L}^{L} [B]^{T} \begin{Bmatrix} s_{\mathrm{t}} \\ s_{\mathrm{n}} \end{Bmatrix} dl = \int_{-1}^{1} [B]^{T} \begin{Bmatrix} s_{\mathrm{t}} \\ s_{\mathrm{n}} \end{Bmatrix} L dx \,. \tag{5}$$

The integrals above are calculated numerically by using 4-point Gauss integration in the FE-calculations

$$\int_{-L}^{L} A dl = \int_{-1}^{1} A L dx = \sum_{i=1}^{4} A_i H_i L \,, \tag{6}$$

where H_i is the Gauss weigh associated with the integration point i. The four Gauss integration points are located at $x =$ -0.861136311594053, -0.339981043584856, 0.339981043584856, 0.861136311594053 respectively, and the corresponding weighs are 0.347854845137454, 0.652145154862546, 0.652145154862546, 0.347854845137454 respectively.



(a) Tangential direction (b) normal direction

Fig. 3 Stress-displacement relation forspecial

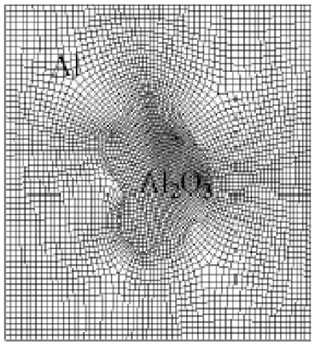


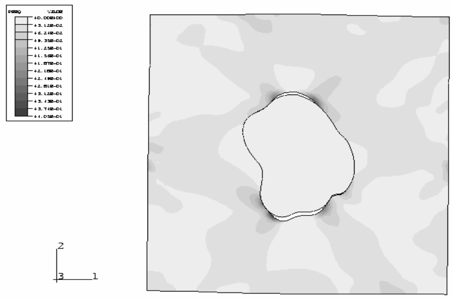
Fig. 4 Computational mesh interfacial element

Table 1 Material Properties parameters

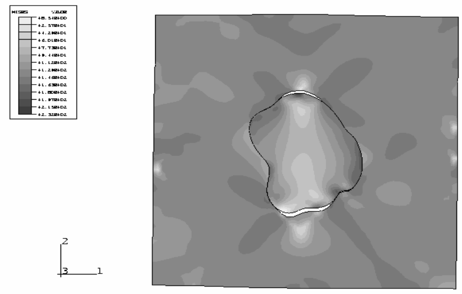
	E (GPa)	n	s_0 (MPa)	s_8 (MPa)
Al	68.3	0.33	105.0	170.0
Al_2O_3	380.0	0.22	---	---

Table 2 Interface elements with dissimilar failure

Interface element	1	2	3	4	5	6	7
s_{max} (MPa)	170	120	85	170	170	85	85
d_c (μm)	2.0	2.0	2.0	1.0	0.5	1.0	0.5

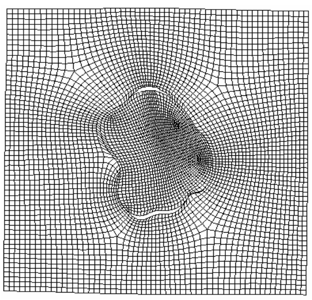


(a)Effective plastic strain（PEEQ）

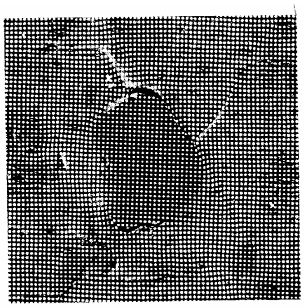


(b) Mises effective stress

Fig.5 Computational results for 7th interfacial element under real displacement loading



(a) Computational results in this paper



(b) Observation by Fischer

Fig.6 Comparison the computational results and experimental results by Fischer [2,3]

Numerical simulation of the strength of an Al/Al_2O_3 MMC microstructure

After establishing a proper element for studying interfacial failure, now we attempt to apply it to the numerical simulation of the strength of a MMC microstructure with irregular shape inclusions. Modeling on the real microstructure subjected to the experimental displacement loading as measured by Fischer *et al.*[1] and as a first approximation, we investigate a cell with only the biggest inclusion neglecting the influence of other small inclusions and the interaction between them. The geometry of this cell is shown in Fig. 4 and the material properties are listed in Table 1. The non-continuum interface element described in Section 2 is used here and we suppose that the tensile behavior of the element in the x-direction and the behavior in the n-direction are the same. Seven interface elements with dissimilar failure parameters, namely seven interfacial failure modes with different stress-displacement jump relations are designed as shown in Table 2 and Fig. 3, where δ_c and s_{max} are the displacement jump limit and the strength limit respectively, which are the parameters set for the interface element.

The geometry of a cell simply approximate to a real microstructural area, which was made of the composite material Al6061-10%Al_2O_3 and measured in the experiment in Refs. [1,2,3]. The edge displacements of the real microstructure shown in Fig. 1 have been measured by using fine grating method in Ref. [2]. Such edge displacements of the real microstructure obtained from the experiment are applied to the simplified model shown in Fig. 4. Incorporating the seven non-continuum interfacial failure elements with dissimilar stress-displacement jump relations listed in Table 2, we carry out the numerical simulation respectively. The contour plots of equivalent plastic strain (PEEQ) and Mises equivalent stress for the microstructure subjected to experimentally obtained edge displacements when the 7th interface element used in the calculation are shown in Fig.5. However some differences in the distributions of the plastic strain or stress bands and in the peak value of stress or strain in the matrix can be found between the seven cases. It is indicated that the influence of the interfacial parameters (s_{max} and d_c) on the deformation and the interfacial failure of the microstructure is weakened when the microstructure is embedded in the composite and under the real complex loading condition The binary image of the deformed real microstructure experimentally obtained by Fischer *et al.*[2,3] and the computational result of the microstructure for the 7th interface element are shown in Fig. 6, from which we can find that the tendency for the real microstructure to fail at the interface is reflected in the numerical result neglecting small inclusions in this paper. So we can say that the numerical result agrees basically with the experimental result.

Conclusions

Summarizing the work in this paper, we can draw several conclusions as follows:

1. The results of the numerical simulation for a microstructure made of a metal matrix composite with an irregular shape inclusion indicate that the failure of the microstructure initiates at the interface along the inclusion where the curvature of its edge changes sharply which are in agreement with the experimental observations.
2. The interfacial failure of the microstructure mainly depends on the parameters controlling the interfacial behaviors such as the cohesive strength s_{max} and the stiffness of the interfacial spring d_c; and the loading transfer between microstructures is another considerable factor of the interfacial failure in the microstructure as well.
3. An applicable non-continuum interface element has been formulated. This interface element has advantages over traditional spring elements, in that it is not required to determine the normal and tangential directions of the interface in advance.

4. A hybrid method for the investigation of the mechanical behaviors of microstructures has been proposed, which combines the experimental measurements with the numerical simulations. The discussions are centered on the deformation, slip and damage processes near the interfacial region in a microstructure of a metal matrix composite with an irregular shape inclusion.

Acknowledgements

The authors wish to acknowledge the financial supports by National Natural Science Foundation of China (No. 10232030,10172050) and key International co-operation project for NNSF of China.

Reference

[1] Liu YL, Fischer G. "Local strain fields in particulate metal matrix composites characterized by the object grating technique". *Proc. ICCM-11*, M.L.Scott ed., Australian: Composite Structure Society, 1997(3). p.1-9.
[2] Fischer G, Soppa E *et al.* "Modelling of strain localization in real microstructural areas of the particle reinforced metal-matrix composite Al6061-10%Al_2O_3". *Proc. of the 19th Riso Int. Symp. on Material Science*, Eds by J.V. Christensen *et al*. Denmark: 1998. p.261~266.
[3] Kang YL, Soppa E, "Schmauder S. Analysis of the modeling of stress and strain fields in a real microstructure using a hybrid method". *ACTA Mechanic Solid Sinica* 2002; 15(3):277-285.
[4] Soppa,E. Schmauder,S,Fischer,G,*et al.*, "Deformation and damage in AL/AL_2O_3", *Comp. Materials Science*, 2003,28:,574-586.
[5] Achenbach JD, "Zhu H. Effect of interfacial zone on mechanical behavior and failure of fiber-reinforced composites". *J Mech Phys Solids* 1989; 37:381-393.
[6] Chen LP, Pan JZ, "Qian LX. Parameter-variables FEM numerical simulation for the interface failure of an interface of fiber-matrix composite". *J of Composites* 1993; 3:71-75. (In Chinese)
[7] Ye BQ *et al.*, "An analysis of interfacial behavior of composite by using interfacial element". *Appl. Math. And Mechanics* 1996; 17:343-346. (In Chinese)
[8] Allix O, Ladeveze P., "Interlaminar interface modeling for the prediction of delamination". *J Composite Struct.* 1992; 22:235~242.
[9] Needleman A., " A continuum model for void nucleation by inclusion debonding". *J Appl Mech* 1987; 54:525-531.
[10]Needleman A., "An analysis of tensile decohesion along an interface". *J.Mech.Phys.Solids*, 1990; 38:289-324.
[11] Tvergaaed V., "Effect of fiber debonding in a whisker-reinforced Metal". *Mater Sci Eng* A 1990; 125: 202-213.
[12] Zhou ZW., "Multi-damage of metal matrix materials", *Ph D Thesis*, Tsinghua University, Beijing China, 1999. (In Chinese)
[13] Xu X P, Needleman A., "Void nucleation by inclusion debonding in a crystal matrix". *Model Simul. Mater. Sci. Eng.,* 1993; 1:111-132.
[14] Xu X P, Needleman A., "Numerical simulations of dynamic crack growth along an interface". *Int. J. Fracture,* 1996; 74:289~324.
[15] Niu XR, YU S W and Feng X Q., " Finite element simulation of thermal mismatch of interfaces of dissimilar materials", *Journal of Tsinghua University*,2001, 40(4/5):104-107.(In Chinese)
[16] Hibbitt, Karlsson & Sorensen., "ABAQUS/Standard User Manual II and III", *Version 5.8. HKS Inc*. 1998