# A New Interface Element for Shell Structures Delamination Analysis

**Biao Li[1,*], Yazhi Li[1], Jie Su[1]**

[1] School of Aeronautics, Northwestern Polytechnical University, Xi'an 710072, China
* Corresponding author: libiao@mail.nwpu.edu.cn

**Abstract** A combined interface element consists of eight rigid beams and a zero thickness cohesive element is presented. The type of elements are used in conjunction with shell elements to constitute the three dimensional model. In the FE model, intralaminar damage takes place within the shell elements and interlaminar damage is restricted to occur at the interface elements. The eight-node interface elements are of finite thickness, with each node possessing six DOFs. No additional DOFs are required in the FE model except those of shell elements. The translational and rotational movements of the shell nodes contribute to the deformation of the interface elements. The interfacial damage accumulation and final delamination are characterized by progressive stiffness degradation of the internal cohesive element. The crack growth in double cantilever beam (DCB) was simulated with the proposed elements and corresponding finite element model. The simulation results agree well with the experimental ones and analytical solutions.

**Keywords** composite laminate, cohesive zone model, interface element, delamination

## 1. Introduction

Fiber reinforced composites are mostly used in the form of laminates, which are susceptible to interfacial delamination due to the weak interfacial bonding strength. Several numerical tools such as fracture mechanics based virtual crack closure technique (VCCT) [1, 2] and damage mechanics based cohesive zone model (CZM) method [3, 4] have been proposed to simulate the interfacial fracture process. CZM is superior to other methods in that the initiation and growth of delamination are considered within the same analysis without previous knowledge of crack position. The conventional way of applying CZM is to embed the cohesive elements among the layers of three dimensional solid elements (Fig. 1a). A softening constitutive law described by traction-displacement jump curve is introduced for the cohesive elements. The irreversible softening process is initiated when the traction attains the maximum interfacial strength and delamination is fully developed when the local energy release rates approach their critical values. However, it has been shown that highly refined mesh is required within cohesive zone which is a softening region ahead of crack tip [5]. The length of cohesive zone is usually on the order of 1 mm for typical polymeric matrix composite, which requires that much smaller elements is unacceptable for large scale application of delamination analysis using solid elements. On the other hand, more computational efficient shell elements are suitable for modeling thin walled structures than solid elements [6].

In this paper, we present a new interface element being used in conjunction with shell elements to constitute three dimensional models for laminated structures. Double cantilever beam (DCB) test are simulated using the proposed interface elements and corresponding finite element model. The simulated results are compared with experimental results and analytical solutions.

## 2. Finite element model

### 2.1. Model description

In the finite element model, laminated structures are divided into several sublaminates through the thickness. A sublaminate is a set of adjacent physical layers among which debonding is unlikely to occur. All sublaminates are modeled with four node quadrilateral shell elements on mid-planes of them. The finite thickness interface elements are used to connect shell elements belonging to adjacent sublaminates, see Fig. 1b. Thus the laminates could be considered as sandwich shells stacked by shell elements and the interface elements. In this way, the intralaminar and interlaminar damages could be considered separately with both kinds of elements.

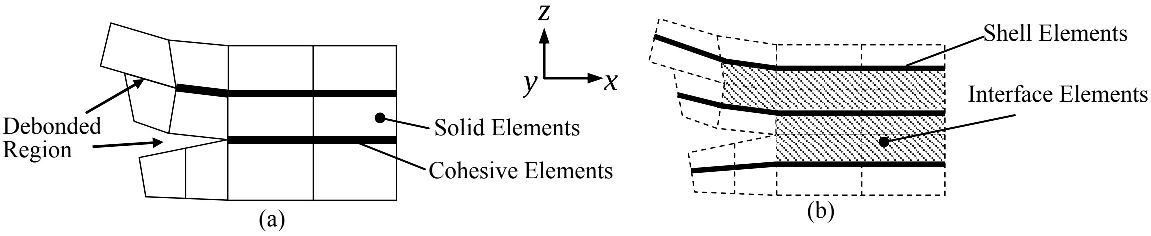


Figure 1. (a) 3D solid model (b) 3D shell model

The interface element consists of eight rigid beams and a zero thickness cohesive element. The master node of a rigid beam is used to connect external shell node, and its slave node is used to connect internal cohesive element node. The rigid beams transfer the translational and rotational movements of the shell nodes to the internal cohesive element (Fig. 2). Interlaminar fracture takes place at the internal cohesive elements. Although being composed of discrete rigid beams and cohesive element, the interface element presents itself as a solid one in which DOFs of the internal nodes have been eliminated by deduction of its kinematic formulae. There will be no additional DOFs are required in the FE models except those of shell elements. The interface element has been implemented in the commercial FE code ABAQUS [7] via its user element subroutine (UEL). Newton-Cotes full integration scheme is adopted.

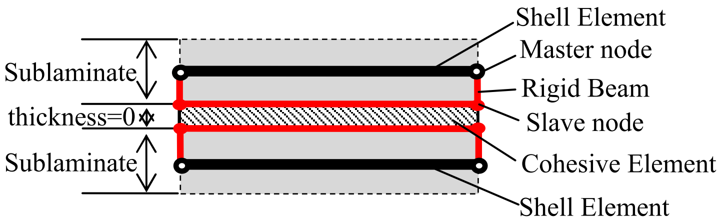


Figure 2. Schematic of the connection of shell elements and interface element

## 2.2. Interface element formulation

In the interface element, the relation of the nodal displacements between master and slave nodes of the $i$th ($i$=1,…,8) rigid beam can be given as:

$$\boldsymbol{u}_{\mathrm{S}}^{i}=\boldsymbol{t}_{\delta\mathrm{i}}\boldsymbol{u}_{\mathrm{M}}^{i}\quad(\mathrm{i}=1,\ldots,8)\tag{1}$$

where $\boldsymbol{u}_{\mathrm{M}}^{i}=\left(u_{\mathrm{M}}^{i},v_{\mathrm{M}}^{i},w_{\mathrm{M}}^{i},\theta_{\mathrm{Mx}}^{i},\theta_{\mathrm{My}}^{i},\theta_{\mathrm{Mz}}^{i}\right)^{\mathrm{T}}$ and $\boldsymbol{u}_{\mathrm{S}}^{i}=\left(u_{\mathrm{S}}^{i},v_{\mathrm{S}}^{i},w_{\mathrm{S}}^{i},\theta_{\mathrm{Sx}}^{i},\theta_{\mathrm{Sy}}^{i},\theta_{\mathrm{Sz}}^{i}\right)^{\mathrm{T}}$ are the displacement vectors at master and slave nodes of $i$th rigid beam respectively. $\boldsymbol{t}_{\delta\mathrm{i}}$ is the displacement transformation matrix of the two nodes:

$$\boldsymbol{t}_{\delta\mathrm{i}}=\begin{bmatrix}1&0&0&0&z_S^i-z_M^i&-\left(y_S^i-y_M^i\right)\\0&1&0&-\left(z_S^i-z_M^i\right)&0&x_S^i-x_M^i\\0&0&1&y_S^i-y_M^i&-\left(x_S^i-x_M^i\right)&0\\0&0&0&1&0&0\\0&0&0&0&1&0\\0&0&0&0&0&1\end{bmatrix}\tag{2}$$

where $x$, $y$, $z$ are coordinates, subscript S and M denote the slave and master nodes respectively, superscript $i$ is the number of rigid beams. Similarly, the relation of nodal forces between master and slave nodes can be expressed as:

$$\boldsymbol{F}_{\mathrm{S}}^{i} = \boldsymbol{t}_{\mathrm{Fi}}\boldsymbol{F}_{\mathrm{M}}^{i} \tag{3}$$

where $\boldsymbol{F}_{\mathrm{M}}^{i}$ and $\boldsymbol{F}_{\mathrm{S}}^{i}$ is the master and slave nodal force of $i$th rigid beam. $\boldsymbol{t}_{\mathrm{Fi}}$ is the nodal force transformation matrix. Furthermore, the relation of the two transformation matrices can be obtained according to the principal of virtual work:

$$\boldsymbol{t}_{\delta\mathrm{i}}^{\mathrm{T}} = -\boldsymbol{t}_{\mathrm{Fi}}^{-1} \tag{4}$$

The relations (1) and (3) for all rigid beams in an interface element are brought together as:

$$\boldsymbol{u}_{\mathrm{S}}^{\mathrm{N}} = \boldsymbol{T}_{\delta}\boldsymbol{u}_{\mathrm{M}}^{\mathrm{N}} \text{ and } \boldsymbol{F}_{\mathrm{S}}^{\mathrm{N}}=\boldsymbol{T}_{\mathrm{F}}\boldsymbol{F}_{\mathrm{M}}^{\mathrm{N}} \tag{5}$$

where

$$\boldsymbol{T}_{\delta} = \begin{bmatrix} \boldsymbol{t}_{\delta 1} & & & 0 \\ & \boldsymbol{t}_{\delta 2} & & \\ & & \cdots & \\ 0 & & & \boldsymbol{t}_{\delta 8} \end{bmatrix} \quad \text{and} \quad -\boldsymbol{T}_{\mathrm{F}}^{-1}=\boldsymbol{T}_{\delta}^{\mathrm{T}} \tag{6}$$

The displacement jumps of internal cohesive element, $\boldsymbol{\delta}$, can be obtained as:

$$\boldsymbol{\delta} = \boldsymbol{B}\boldsymbol{\psi}\boldsymbol{u}_{\mathrm{S}}^{\mathrm{N}} \tag{7}$$

where

$$\boldsymbol{\psi} = \begin{bmatrix} \boldsymbol{\gamma} & & & \\ & \boldsymbol{\gamma} & & \\ & & \cdots & \\ & & & \boldsymbol{\gamma} \end{bmatrix}_{24\times 48}, \quad \boldsymbol{\gamma} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \end{bmatrix}$$

is used to pass translational movements of the slave nodes to cohesive element. $\boldsymbol{B}$ transforms the nodal displacements of the cohesive element to local displacement jumps.
The constitutive law for internal cohesive element can be defined as:

$$\boldsymbol{\tau} = \boldsymbol{D}\boldsymbol{\delta} \tag{8}$$

where $\boldsymbol{\tau}$ is stress vector, $\boldsymbol{D}$ is damaged elasticity matrix. With the use of virtual work principle, we get the stiffness matrix of the interface element:

$$\boldsymbol{K} = \boldsymbol{T}_{\delta}^{\mathrm{T}}\boldsymbol{\psi}^{\mathrm{T}}\int_{\Gamma}\boldsymbol{B}^{\mathrm{T}}\boldsymbol{D}\boldsymbol{B}d\Gamma\boldsymbol{\psi}\boldsymbol{T}_{\delta} \tag{9}$$

The nodal force vector of the element is:

$$\boldsymbol{F}=\boldsymbol{T}_{\delta}^{\mathrm{T}}\boldsymbol{\psi}^{\mathrm{T}}\int_{\Gamma}\boldsymbol{B}^{\mathrm{T}}\boldsymbol{\tau}d\Gamma \tag{10}$$

It is seen that the nodal force vector and stiffness matrix of the new interface element can be simply obtained from cohesive element as follows:

$$\boldsymbol{K} = (\boldsymbol{\psi}\boldsymbol{T}_{\delta})^{\mathrm{T}}\boldsymbol{K}_{\mathrm{cohesive}}(\boldsymbol{\psi}\boldsymbol{T}_{\delta}) \quad \text{and} \quad \boldsymbol{F}=(\boldsymbol{\psi}\boldsymbol{T}_{\delta})^{\mathrm{T}}\boldsymbol{F}_{\mathrm{cohesive}} \tag{11}$$

### 2.3. Bilinear constitutive law

A bilinear constitutive damage model under pure Mode I loading is adopted for the new interface element, see Fig.3. The interfacial damage is initiated after the normal traction attains the interfacial tensile strength. After that, the stiffness is gradually reduced to zero. The onset displacement is obtained as: $\delta_3^{\mathrm{o}}=N/K$, where $N$ is the interfacial tensile strength and $K$ is the interfacial stiffness. The area under the traction-displacement jump curve is the Mode I fracture toughness $G_{\mathrm{IC}}$:

$$\int_0^{\delta_3^f} \tau_3 d\delta_3 = G_{\rm IC} \tag{12}$$

where $\tau_3$ is normal traction. The final displacements $\delta_3^f$ can be obtained as: $\delta_3^f = 2G_{\rm IC} / N$ .

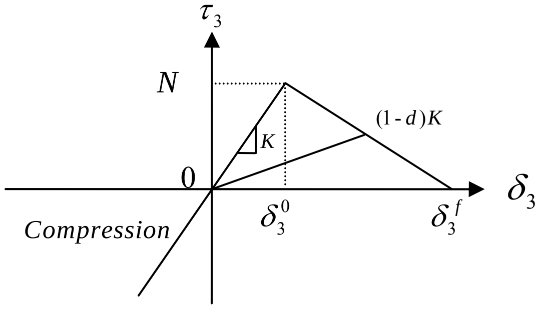


Figure 3. Pure Mode I bilinear constitutive law

## 3. Numerical simulation of DCB specimen

The double cantilever beam (DCB) is a standard specimen for mode I fracture and has been widely investigated. In this section, we simulated the DCB test [8] using the proposed interface element and relevant FE model. The specimen is made up of T300/977-2 unidirectional laminates $[0]_{24}$. The geometry and boundary conditions are shown in Fig. 4 and material properties are listed in Table 1.

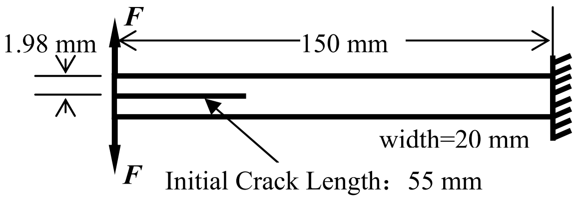


Figure 4. Geometry and boundary condition for DCB test

Table 1. Properties for T300/977-2[8]

| $E_{11}$ | $E_{22}$=$E_{33}$ | $G_{12}$=$G_{13}$ | $G_{23}$ |
|---|---|---|---|
| 150.0 GPa | 11.0 GPa | 6.0 GPa | 3.7 GPa |
| $\nu_{12}$=$\nu_{13}$ | $\nu_{23}$ | $G_{IC}$ | $N$ |
| 0.25 | 0.45 | 0.352 kJ/m$^2$ | 60 MPa |

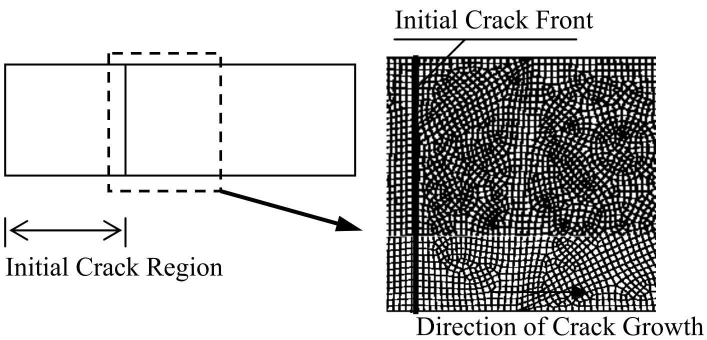


Figure 5. Irregular mesh for DCB model

The FE model for DCB specimen was built with two layers of shell elements at the mid-planes of both beam arms. The interface elements were employed to connect the upper and lower shells except where of pre-crack. The interfacial stiffness was chosen as $K$=10$^6$ N/mm$^3$ . To demonstrate the applicability of the interface element, we established two models with regular and irregular

meshes respectively. The regular mesh contains 24000 square shell elements, 7600 interface elements and 24682 nodes. The element size of the mesh is 0.5 mm×0.5 mm. The irregular mesh shown in Fig. 5 contains 29320 shell elements, 9296 interface elements and 30006 nodes and the nominal element size of which is 0.5 mm. The curves of load vs. deflection by FE analysis, experiment and analytical solutions were presented in Fig. 6, in which the analytical solution is obtained using a corrected beam theory for mode I case.
It can be seen that the coincidence of the simulation results between regular and irregular meshes is very well. The predicted strength is closed to the experimental one. The simulated result of load vs. deflection curve achieves good agreement with experimental and analytical results. The mesh with the new interface elements is easy to be built for the same geometric form as brick elements. The longitudinal stress contours are exhibited in Fig. 7 on the deformed meshes at different moments of loading. The scaling factor of the deformation display is 4.0.

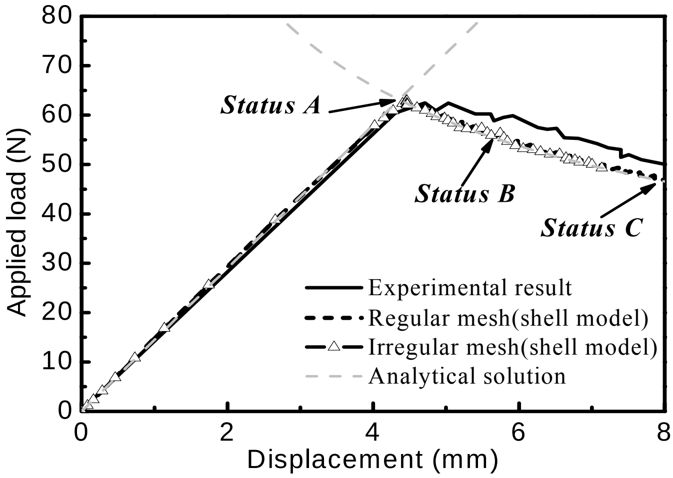


Figure 6. Results comparison for DCB

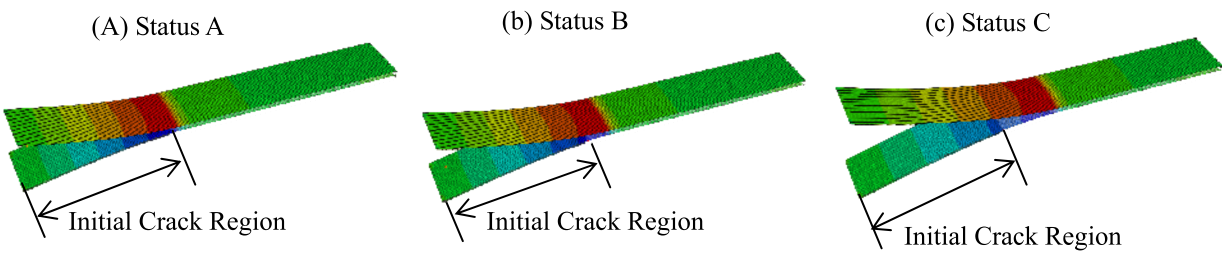


Figure 7. $S_{11}$ stress contour of DCB specimen at different status (regular mesh)

## 4. Conclusions

A combined interface element is proposed to be used together with shell element to establish finite element model of the shell structures and conduct interfacial fracture analysis. The new interface element comprised of a zero-thickness cohesive zone element and a number of rigid beam elements. The shell thickness offset and nodal translational and rotational degrees of freedom are considered by the use of rigid beams. A bilinear constitutive law is applied to the new interface element. The modeling technique greatly reduces the model scale as compared with the model built with solid elements. By the simulations of double cantilever beam test, good agreements are achieved among the results of simulation, experiment and analytical solution.

**References**

[1] E.F. Rybicki, M.F. Kanninen. A finite element calculation of stress intensity factors by a modified crack closure integral. Engineering Fracture Mechanics, 1977, 9: 931-938.

[2] I.S. Raju. Calculation of strain-energy release rates with higher order and singular elements. Engineering Fracture Mechanics, 1987, 28:251-274.
[3] A. Hillerborg, M. Modeer, P.E. Petersson. Analysis of crack formation and crack growth in concrete by means of fracture mechanics and finite elements. Cem Concr Res, 1976,6:773-782.
[4] V.K. Goyal, E.R. Johnson, C.G. Davila. Irreversible constitutive law for modeling the delamination process using interfacial surface discontinuities. Composite Structures, 2004,65:289-305.
[5] P.W. Harper, S.R. Hallett. Cohesive zone length in numerical simulations of composite delamination. Engineering Fracture Mechanics, 2008,75:4774-4792.
[6] P.J. Gray, C.T. McCarthy. A global bolted joint model for finite element analysis of load distributions in multi-bolt composite joints. Composite: Part B, 2010, 41: 317-325.
[7] ABAQUS 6.11 User's Manual, ABAQUS Inc, Province, RI, USA, 2011.
[8] A. Turon, C.G. Davila, P.P. Camanho, et al. An engineering solution for mesh size effects in the simulation of delamination using cohesive zone models. Engineering Fracture Mechanics, 2007,74:1665-1682.