中國機械工程學刊第四十四卷第四期第 371~377 頁(民國一百一十二年)
Journal of the Chinese Society of Mechanical Engineers, Vol.44, No.4, pp371~377 (2023)

Study on Fracture Simulation of Composite Material Structures by Extended Finite Element Method

Chia-kan Chang*and Javid Bayandor**

Keywords：Composite material, Finite element method, fracture.

ABSTRACT

In the simulation of structural strength, the finite element method has always been an important means. In this paper, the extended finite element method (XFEM) was used to simulate the three-point bending process of carbon fiber composite laminates. The load-displacement curves obtained by simulation were compared with the experimental results, and it was found that the extended finite element method could accurately simulate the fracture propagation process of carbon fiber composite laminates during three-point bending. The findings of this paper can reduce part of the destructive experiments, so as to save the related costs.

INTRODUCTION

Composite materials are widely used in automobile, aerospace and other fields due to their outstanding mechanical properties according to the study made by Libing Zhao et al.(2019) Carbon fiber laminates are prone to large deformation and bending under impact load in practical use, lead to local stress concentration and the strain that cause material damage, local stress concentration and strain will result in material damage, such as matrix fracturing, fiber fracture or interlayer delamination, etc. The extension of material damage will further lead to the reduction of mechanical properties, resulting in material failure and ultimately structural failure. By introducing enrichment function, extended finite element method modified approximate displacement function of traditional finite element method to describe discontinuity interface, which made discontinuity description independent of finite element mesh and avoided mesh reconstruction in the calculation process.

Paper Received January, 2023. Revised May, 2023. Accepted May, 2023. Author for Correspondence: Chia-kan Chang.

* *Graduate Student, Department of Mechanical and Aerospace Engineering, University at Buffalo, State University of New York, Buffalo, New York 14260, United States.*

** *Associate Professor, Department of Mechanical and Aerospace Engineering, University at Buffalo, State University of New York, Buffalo, New York 14260, United States.*

EXTENDED FINITE ELEMENT METHOD

Extended finite element method is a numerical method for solving discontinuity problems proposed by the research group of Prof. Belytschko et al.(1999) at Northwestern University, which can effectively solve strong and weak discontinuity problems listed by Lilun Guo et al.(2011) The basic principle of extended finite element method is to add a special function (strengthening function) into the traditional finite element displacement mode based on the unit decomposition method, so as to reflect the existence of discontinuity. By the study made by Zhuo Zhuang et al.(2012), different types of discontinuity problems can be regarded as different reinforcing functions.

Unit Decomposition Method

The unit decomposition method was proposed by Melenk and Bubska.(1996), for solving region Ω, the unit decomposition method is covered by overlapping subfields Ω_I, every subfield is connected with a function $\varphi_I(x)$. $\varphi_I(x)$ is a non-zero function only in Ω_I, and satisfy the condition of unit decomposition

$$\sum_I \varphi_I(x) = 1, \tag{1}$$

Duarte and Oden.(1996) use K order moving least squares approximation functions to construct the unit decomposition, i.e:

$$u^h(x) = \sum_I \varphi_I^k(x)[u_I + \sum_{i=1}^{m} b_{iI} q_i(x)], \tag{2}$$

In the equation above, $q_i(x)$ could be the monomial base, the coefficients are unknown quantities and can be solved by Galerkin method or collocation method. In order to improve approximation accuracy, or to satisfy special approximation requirements for undetermined problems, other forms of functions (enhanced basis functions) may also be included.

Another extended form of the unit decomposition method is to mix different unit factorizations, typically：

$$u^h(x)=\sum_{I=1}^{n_1}\varphi_I^k(x)u_1+\sum_{I=1}^{n_2}\varphi_I^0(x)\sum_i b_{iI}q_i(x)\,, \qquad (3)$$

In the equation above, $q_i(x)$ are strengthening basis functions, such as fracture-tip dependent singular functions.

In general, $n_2<<n_1$, in that case, the effect of the reinforcement base of the model (3) is limited to a small area, and its increased computation amount because of it is not large. The extended finite element method adopts the form of Equation (3).

Melenk and Bubska.(1996) have proved the convergence of the unit decomposition method, so the convergence can be guaranteed as long as the numerical method is constructed based on the unit decomposition method.

The Displacement Mode

The extended finite element model is based on the traditional finite element model, adding the displacement field function reflecting the fracture surface and fracture tip. For example, the displacement function u(x) of object Ω with an internal fracture can be divided into continuous part and discontinuous part, i.e:

$$u=u_{cont}+u_{disc}, \qquad (4)$$

the u_{cont} is continuous in Ω and u_{disc} are not.

Apply the conventional finite element approximation to solve u_{cont}, i.e:

$$u_{cont}=\sum_{i\in N^s}N_i(x)u_i\ , \qquad (5)$$

Where: N^s is all node sets in the discrete domain, N_i is node shape function, U_i is node displacement.

The discontinuous part of the displacement u(x) can be expressed as:

$$u_{disc}=\sum_{i\in N^{cut}}\overline{N}_i(x)H(x)a_i+\sum_{i\in N^{tip}}\overline{N}_i(x)\sum_j^4\Phi_j(x)b_i^j\ , \qquad (6)$$

Where, N^{cut} is the node set which is completely cut by fracture in the support domain of shape function, and N^{tip} is the node set with fracture tip in the support domain of shape function. N_i is the corresponding nodal shape function; ai and b_i^j are strengthening variables of corresponding nodes, respectively. H(x) is a generalized step function, equal to 1 on the fracture surface, equal to -1 under the fracture surface; $\Phi_j(x)$ is a displacement field function which can reflect the singularity of fracture tip, its expression is as follows:

$$\Phi_j(x)=[\sqrt{r}\sin\frac{\theta}{2},\sqrt{r}\cos\frac{\theta}{2},\sqrt{r}\sin\frac{\theta}{2}\sin\theta,\sqrt{r}\cos\frac{\theta}{2}\sin\theta] \qquad ,(7)$$

In the formula above, the r and θ is the polar coordinates of the crack tip.

FINITE ELEMENT EXPERIMENTS

Establishment of Numerical Model

A three-point bending finite element model of carbon fiber composite laminates was established by finite element software ANSYS. Its dimensions, boundary conditions and stress conditions are illustrated in fig. 1. The laminates and supports are constrained by hinges, with fixed hinges on the left and sliding hinges on the right. The load is applied at the midpoint of the beam by way of displacement loading. The laminate is divided into units, the unit type is PLANE182, and the total number of units for single layered model is 2773. Its numerical model is presented in fig. 2. Material parameters of the model are shown in Table 1:

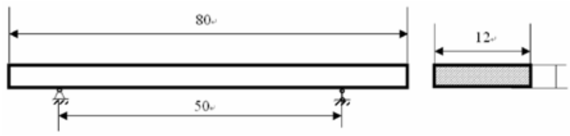

Fig. 1. Three point bending model dimensions.

Fig. 2. Three point bending ANSYS model of laminates.

Table 1. Model material parameters

PARAMETERS	VALUES
Densityρ(Kg/m^3)	1600
Transverse Young's modulus E_{11}=E_{22}(MPa)	15500
Longitudinal Young's modulus, E_{33}(MPa)	5000
Shear Modulus G_{12}(MPa)	8770
Shear Modulus G_{13}=G_{23}(MPa)	6200
Poisson's ratio ν_{12}	0.26
Critical energy release rate G_{1C}(kJ/m^2)	0.125
Critical energy release rate G_{2C}(kJ/m^2)	0.175

Damage Process Analysis

The load-displacement curve was obtained through simulation calculation, and the simulation results were compared with experimental results made by Weiming Zhuang's team.(2019), as showed in fig. 3.

As can be seen from the curve comparison in fig. 3, the maximum load in simulation calculation is 266.5N, and the maximum load in the experiment is 280.6N, with an error of 3.7%. The simulated maximum displacement is 2.893mm, the experimental displacement is 2.794mm, and the error is 3.5%, which proves that the extended finite element method can better simulate the fracture growth process of carbon fiber laminates and meet the accuracy requirements.

According to the simulation calculation, the equivalent stress cloud diagram of the laminated plate is obtained, as showed in fig. 4. The fracturing process of the laminated plate is successively from top to bottom:

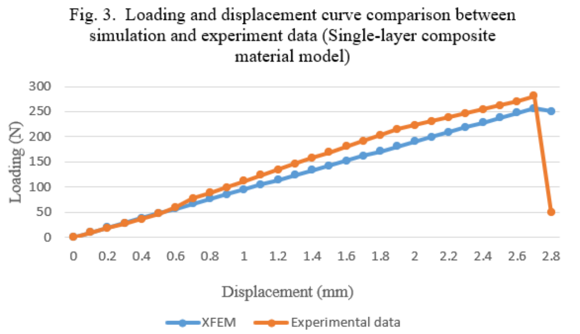

Fig. 3. Loading and displacement curve comparison between simulation and experiment data (Single-layer composite material model).

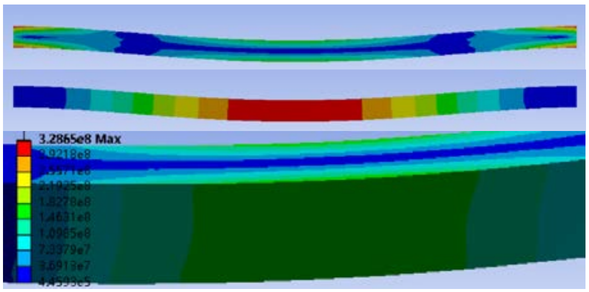

Fig. 4. The stress cloud, the deformation state cloud, and the part where the fracture will start of the first status (From the top downward).

At the first status of deformation and fracture, the middle part of the composite plate is deformed due to pressure, but there is no fracture. The above figure is the stress cloud, and the second one is the deformation state cloud, the third figure is the part where the fracture will start, the left bar is the stress scale (Pa).

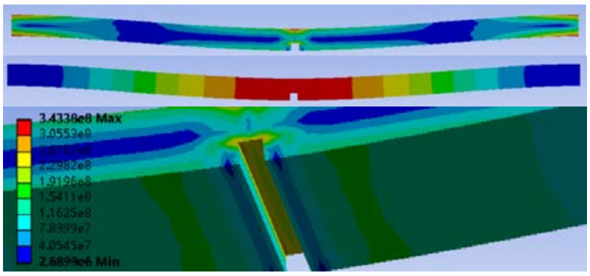

Fig. 5. The stress cloud, the deformation state cloud, and the part of fracture of the second status (From the top downward).

As figure 5, this is the status when the fracture just appeared. The middle part of the composite plate was subjected to greater pressure, resulting in a fracture. Due to the calculation speed, the simulation here is not very precise, the mesh is rough. As the stress increases further, the fracture will continue to grow. The figures are the stress cloud diagram, the deformation state cloud, and the details of the crack. From the detailed figure, the fracture process started and the first layer of mesh was deleted, and we can find that the stress at fracture is obviously greater than that at the first status.

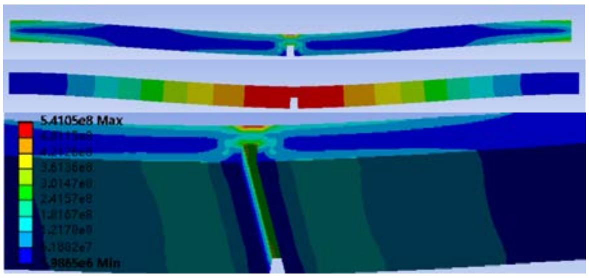

Fig. 6. The stress cloud, the deformation state cloud, and the part of fracture of the third status (From the top downward).

As figure 6, in the third status, with the further increase of stress, the fracture in the middle part continues to expand. It can be seen here that the deformation outside the fracture has basically stopped, which is consistent with the second state. Deformation occurs mainly in the part where fractures are about to grow. The figures are the stress cloud diagram, the deformation state cloud, and the details of the crack. From the detailed figure, the fracture process continued and the second layer of mesh was deleted, and the stress at fracture continued increasing, however, because the actual time between the second and third state is very small, so the stress change is not dramatic. And due to the stress values corresponding to colors are different in different status, the color of the stress cloud at the third status is lighter than that at second status.

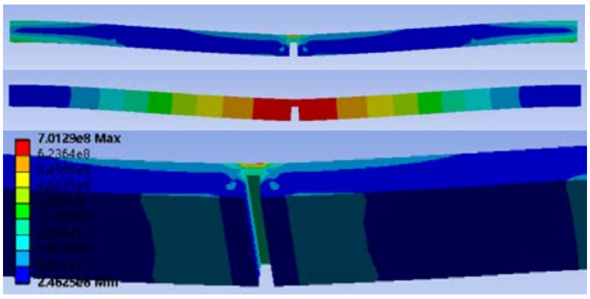

Fig. 7. The stress cloud, the deformation state cloud, and the part of fracture of the fourth status (From the top downward).

As figure 7, due to the stress continues to increase, fractures continue to grow, which is the final status of the simulation. The next state is the complete fracture of the structure, namely the complete fracture of the grid. Since smart fracture growth is not used in the simulation here, the complete fracture of the grid will lead to errors in the simulation. Therefore, this status can be used as the final state of the simulation. The figures are the stress cloud diagram, the deformation state cloud, and the details of the crack. With crack growth, the third layer of mesh was deleted, and the increase trend of stress is stopped, that status is the status just before the structure completely splintering into two parts.

The above figure is stress cloud figure at different time during the fracturing process of laminates, because of the stress is mainly concentrated in the fracture tip region, so the rest of the stress value is small. The extended finite element model is extremely non-convergent when the fracture is completely broken at the last step, but it can accurately simulate the fracture process at the moment before the fracture is completely broken.

In general, there is a linear relationship between the change of pressure and the shape of composite plate, and the difference between the numerical values obtained by XFEM and those obtained by experiment is small, which proves that XFEM can be used to verify the relationship between deformation and loading of single-layer composite plate.

Fracture Growth Simulation of Two Layer Composite Laminates Under Pressure

Since the simulation in last section is only for single-layer plate, it cannot completely prove the universality of the extended finite element method described in this paper. In that case, another experiment will be adopted to test the stress fracturing of double-layer composite laminates to prove the availability of the theoretical method.

The idea of the experiment is exactly the same as the experiment in last section. Only the composite plate in the experiment is replaced with a double-layer one, and the following model is established in ANSYS for simulation:

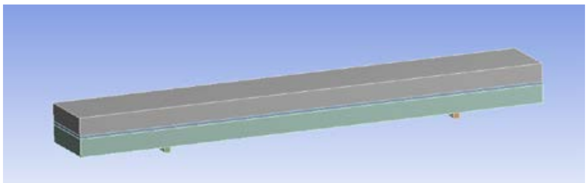

Fig. 8. ANSYS model for two-layer composite material laminates fracture growth simulation.

Tiebreak contact is set in the middle layer of laminates, and uniform mesh and progressive mesh are respectively used for modeling. The following figure is the comparation of experimental data get by Weiming Zhuang's team.(2019) and the simulation result of ANSYS. Set a series of dimensionless time steps, the first time step as 0, the last time step as 0, means the composite plate is completely broken. The loading and displacement curve comparison between simulation and experiment data is shown below:

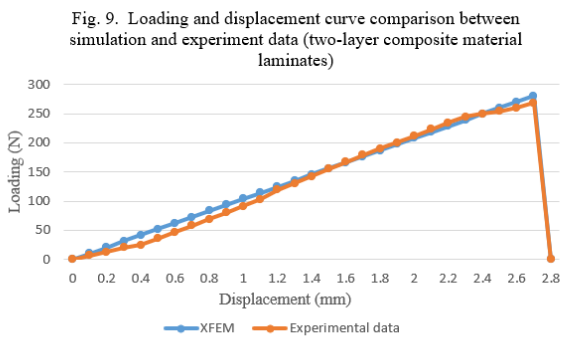

Fig. 9. Loading and displacement curve comparison between simulation and experiment data (two-layer composite material laminates).

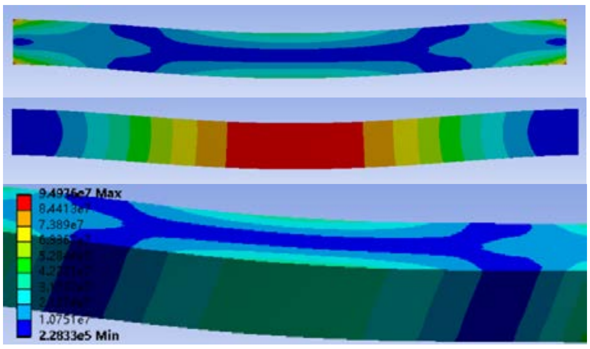

Fig. 10. The stress cloud, the deformation state cloud, and the part where the fracture will start of the first status. (From the top downward). Time step 0.93.

As figure 10, this is the status at the beginning of the simulation. Same as the result in section *Damage Process Analysis*, the middle part of the composite plate is deformed due to pressure, and there is no fracture. The figures are the stress cloud, the deformation state cloud and the details of the crack. Same with the single layer experiment, in this status the fracture process is not start yet, but the deformation of the structure is reaching its limit.

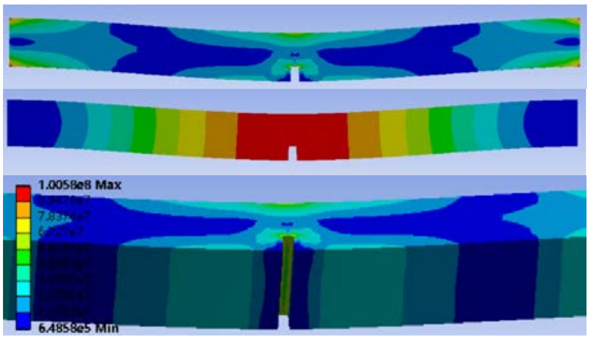

Fig. 11. The stress cloud, the deformation state cloud, and the part of fracture of the second status (From the top downward). Time step 0.96.

As figure 11, this time period represents the fracturing of the lower composite plate due to the increase of load, and the fracture is towards the contact layer between the two layers. It can be predicted that in the next time period, the contact layer will fracture due to the increase of stress, and eventually lead to the complete fracture of the whole structure. The figures are the stress cloud, the deformation state cloud and the details of the crack. In that status, the fracture process started, and because of this is a two layer composite plate with a contact layer between them, the stress value in this experiment is smaller than that in the single layer experiment.

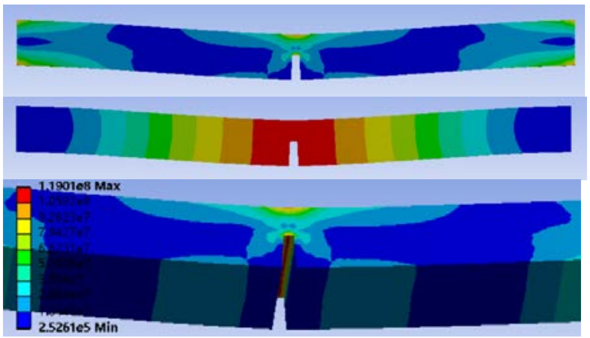

Fig. 12. The stress cloud, the deformation state cloud, and the part of fracture of the third status (From the top downward). Time step 0.98.

As figure 12, at this time period, as a result of the load increase, the fracture of the middle contact part of laminated composite plates happens, within a very short time, in the future, the fracture will be extended to the upper structure, leading to the rupture of the whole structure, same with the phenomena in section *Damage Process Analysis*, at this time, besides the fracture position, other parts of the deformation had mainly disappeared. Because the most vulnerable part of the structure is above the fracture, the stress in other parts is not enough to deform the main structure. The figures are the stress cloud, the deformation state cloud and the details of the crack. The difference between this status and any other ones is that the maximum stress occurs at the crack growth point, because the fracture process is complete in the middle layer, while the other layer of composite material does not begin to fracture.

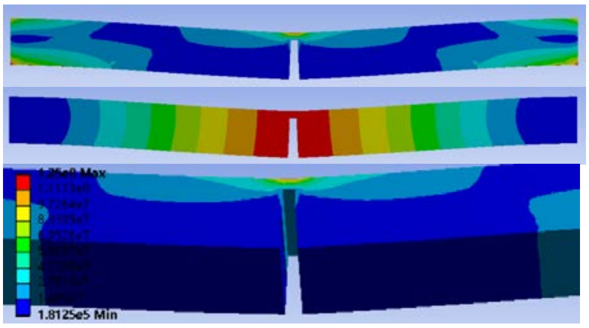

Fig. 13. The stress cloud, the deformation state cloud, and the part of fracture of the third status (From the top downward). Time step 0.99.

In a very short time after the previous status, the composite plate of the upper layer also fractures. For the same reason as section *Damage Process Analysis*, this status is the last status of this experiment simulation, the final layer of mesh is about to be deleted and the structure is about to completely fracture. The figures are the stress cloud, the deformation state cloud and the details of the crack.

Similar to the situation of single-layer composite plate, the data obtained by XFEM in this experiment is also very correspond with the experimental data, indicating that XFEM has a high accuracy in the loading and deformation prediction of multi-layer composite plate.

In the process of bending damage of composite plate, the propagation rate of interlaminar fractures are different from that of intramolecular fractures. Interlaminar fractures begin to appear before the material fracture produces intramolecular fractures, but the propagation rate of fractures is slow. After that, the material's fracture successively from bottom to top, and fractures within the layers develop and expand. Meanwhile, the growth rate of fractures between the layers increases rapidly until the materials are brittle fracture completely.

It should be noted that in the second experiment, there are two fracturing forms of fractures, one is interlayer fracture and the other is intra-layer fracture. Interlayer fracture means that the corresponding layer fractures completely and the fracture expands to the next layer. The intra-layer fracture means that a fracture has started in the corresponding layer but has not yet spread to other layers.

Set the value L_c as the length of crack at the current time step, and L_{max} as the maximum crack length. Set the parameter $\beta = L_c / L_{max}$, the β-time step curve shows the variation of the crack length between laminates and the crack length within laminates with time, representing two failure modes of laminates: delamination failure and fiber fracture. The result is shown as figure 14.

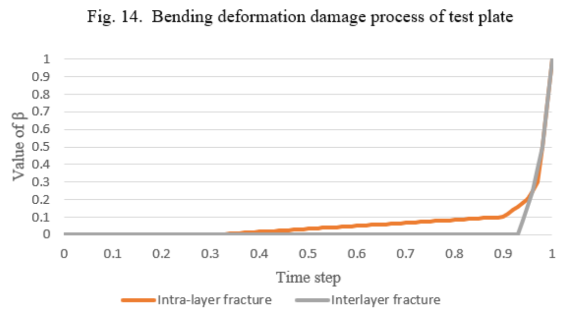

Fig. 14. Bending deformation damage process of test plate.

From the result above, before the time step reaches 0.3, the interface stress of the laminate increases with the continuous loading until the interface limit stress value is reached, but the interface separation distance has not reached the limit value, the contact has not failed, and the interlayer crack has not occurred, as shown in Fig. 6: when the time step reaches the contact failure condition at 0.3, it begins to fail and fractures occur. When the time step is between 0.3 and 0.9, multiple groups of interlayer contact points fail continuously and the crack expands slowly. After the time step of 0.9, the crack growth rate increases sharply until the delamination failure occurs completely.

At the time when the delamination damage occurred, the laminated plate intraformational fiber unit equivalent stress also increased due to the increasing of load, at time step of 0.93, the lower layer's element is deleted because it reaches maximum equivalent stress, the interlayer fracture starts, then, the elements begin deleted from the bottom to the top, and the crack expands in a rapid rate. As shown in process figure 10 to 13 at time step= 0.96, 0.98 and 0.99, the element deletion of layer 1, layer 2 and layer 3 (from bottom to top) occurred at the pressure loading place respectively. When time step= 1.00, the fiber element is completely deleted and the test plate is brittle fracture, and the whole bending damage process is complete.

According to the experiments in section *Damage Process Analysis* and this section, it is not difficult to find that XFEM method can indeed well simulate the compression fracture process of composite structures, and its coincidence with the real experiment is relatively high, no matter the simulation of compression fracture of monolayer composite material or multi-layer composite material.

CONCLUSIONS

The extended finite element method can accurately simulate the three-point bending and tension process of carbon fiber composite laminates, and the load-displacement curve of simulation analysis meets the requirement of accuracy compared with the experimental value.

In the process of bending damage of carbon fiber composite laminates, the growth rates of interlaminar cracks and interlaminar cracks are different, and the intra-layer cracks begin to appear before the fiber fracture produces interlayer cracks, but the crack growth is slow. After that, the fibers fracture one by one from bottom to top, and cracks within the layers develop and expand. Meanwhile, the growth rate of cracks between the layers increases rapidly until the fibers are brittle fracture completely.

Another characteristic found in the experiment is, the extended finite element model does not converge when the laminated plate is completely fractured, but it can reflect the fracturing process of the laminated plate well before fracture.

ACKNOWLEDGMENT

Financial support for this work was provided by the Professor Javid Bayandor and the Department of Mechanical and Aerospace Engineering in University at Buffalo, State University of New York.

REFERENCES

Libing Zhao., Yu Gong., and Jianyu Zhang., "Research progress in delamination behavior of fiber reinforced composite laminates," *Journal of aviation.*, Vol. 40(01), pp. 171-199 (2019).

Dolbow J., and Belytschko T., "Numerical integration of the Galerkin weak form in meshfree methods," *Computational Mechanics.*, Vol. 23(3), pp. 219-230 (1999).

Lilun Guo., Fuzhong Chen., and Jingrun Luo., "A review of extended finite element methods and their applications," *Mechanics quarterly.*, Vol. 32(04), pp. 612-625 (2011).

Zhuo Zhuang., and Binbin Cheng., "An extended finite element method based on CB shell element is developed to simulate 3d arbitrary propagation fracture," *Engineering mechanics.*, Vol. 29(06), pp. 12-21.(2012).

J. M, Melenk., and, et al., "The partition of unity finite element method: Basic theory and applications," *Computer Methods in Applied Mechanics and Engineering.*, Vol. 139(1-4), pp. 289-314 (1996).

Duarte C A., and Oden J T., "An h-p adaptive method using clouds," *Computer Methods in Applied Mechanics and Engineering.*, Vol. 139(1-4), pp. 237-262 (1996).

Weimin Zhuang., Nan Wang., and Di Wu., "Simulation of three-point bending damage of carbon fiber composite laminates," *Journal of Mechanical Engineering*, Vol. 55(10), pp. 109-114 (2019).

NOMENCLATURE

Ω solving region of unit decomposition method

Ω_l overlapping subfields of solving region Ω

$\varphi_I(x)$ a non-zero function only in Ω_l

$q_i(x)$ strengthening basis function

$u(x)$ displacement function

u_{cont} the continuous part in Ω

u_{disc} the discontinuous part in Ω

N^s all node sets in the discrete domain

N_i node shape function

U_i node displacement

N^{cut} the node set which is completely cut by fracture in the support domain of shape function

N^{tip} the node set with fracture tip in the support domain of shape function

b_i^j strengthening variables of corresponding nodes

$H(x)$ generalized step function

$\Phi_j(x)$ a displacement field function

r one of a polar coordinates of the crack tip

θ another polar coordinates of the crack tip

ρ the density of finite elements model

E_{11} one of the transverse Young's modulus of finite elements model

E_{22} another transverse Young's modulus of finite elements model

E_{33} the longitudinal Young's modulus of finite elements model

G_{12} the shear modulus of the compression plane of material

G_{13} one of the shear modulus of the side plane of material

G_{23} another shear modulus of the side plane of material

ν_{12} Poisson's ratio of material

G_{1C} front critical energy release rate of material

G_{2C} side critical energy release rate of material

複合材料結構斷裂仿真與預測的擴展有限元方法研究

張家淦　　賈維德·巴彥多爾
紐約州立大學布法羅分校機械與航空航天工程學系

摘 要

在結構強度的模擬中，有限元法一直是一種重要的手段。本文提出了一種適用於碳纖維複合材料板斷裂研究的擴展有限元法(XFEM)對碳纖維複合材料層合板的三點彎曲過程進行了數值模擬。將模擬得到的荷載-位移曲線與實驗結果進行對比，發現本文提出的擴展有限元法能夠準確模擬碳纖維複合材料層合板三點彎曲過程中的斷裂擴展過程以及其受力情況。本文的研究成果可以替代一部分的破壞性實驗，從而節約相關成本。