ABAQUS Implementation to Analyze the Stress Intensity Factor for Laminated Composite Plate

Achchhe Lal ¹ and M. B. Vaghela ²*

¹Assistant Professor, Department of Mechanical Engineering, S. V. National Institute of Technology, Surat-395007, Gujarat, India.
²Ph. D. Research Scholar, Department of Mechanical Engineering, S. V. National Institute of Technology, Surat-395007, Gujarat, India.
*Email: manoj0085@gmail.com

Abstract: The objective of this work is Abaqus implementation to analyze the effect of fibre orientation angle on stress intensity factor for Single Edge Notched (SEN) 3D laminate composite plate under tensile loading by various cases. The Abaqus software is used to analyze the various problems through the extended finite element method (XFEM). The effects of various fibre orientation angles on Stress Intensity Factor (SIF), Von Misses stress and displacement are also investigated. The presented work would be useful for enhancing the use of Abaqus software to analyze the composite structure with crack and other discontinuities under various conditions.

1. Introduction

The simulate the crack domain into material using conventional finite element methods is quite challenging due to meshing limitations. The extended finite element method (XFEM) is quite useful to solve various discontinuities problems in the material, due to its independent remesh facilities for the crack domain. The crack tip enrichment function is also used in the XFEM to help the better numerical result. The various literatures have been reported on the theories of XFEM which have thrown light on the basic ideas of XFEM.

The material discontinuities like holes, flaw, crack and inclusion are affecting the fracture behaviour of the material. The study about stress, displacement and stress intensity factor will play the major role in the direction of its fracture failure study. It is important in designing the any structure. The investigated crack in a laminated composite plate under tensile loading by implementing ABAQUS by XFEM. In this direction, Sih [1] presented the stress intensity factor evaluation by the energy field theory to measure energy density factor ‘S’ in the fracture mechanics. The critical value of S gives information on crack initiation and fracture toughness of materials. The strain energy density theory is also capable to handle a variety of mixed-mode crack problems. Ke and Liu [2] presented strain at the crack tip as fracture criteria for ductile material and is compared to J-integral and crack surface opening displacement (CSOD) and it is confirmed that CSOD can be used as one of the fracture criteria but the correlation of these criteria with stress intensity factor has not been established. Chien et al. [3] used the FEM with superposition on the singularity at crack-tip. In this study stress intensity factor at the crack tip of a single edge, the crack plate is calculated and found better accuracy in the results.

In the last decades, researchers have proposed many numerical methods to understand the fracture behaviour of the isotropic plate under different loading conditions. Finite element method (FEM) is broadly used numerical method for many engineering applications, but this method is not so efficient
to do desire meshing specially at the crack tip and interior boundary of discontinuities as inclusion, void etc. In FEM, it's necessary to update meshing during the progress of crack and it is time-consuming and costly. In this regard extended finite element method (XFEM) provides extra enrichment functions for the crack face and tip where remeshing is not required during crack growth. The FEM is having meshing limitation at complex shape, so many researcher had emphasize on evaluate fracture mechanics problem using the XFEM.

In this direction, Belytschko et al. [4] developed a better methodology to handle discontinuity by XFEM where discontinuity is incorporated without remeshing and this method is successfully implemented to analyze crack growth in the material under loading. The result obtained from this new methodology is very near to the experimental results. Sukumar and Prevost [5] implemented computer program Dynaflow based on finite element method, where the 2-D crack in isotropic and bimaterial plate is analyzed by implementing XFEM in finite element code. This method provided a simple platform where discontinuous fields by the partition of unity framework are incorporated in a standard finite element package.

Bellec and Dolbow [6] presented the modified enrichment in the XFEM to evaluate the SIF at crack tip using ramp function. This study has been carried out for the special case where crack approaches the local nodal spacing. Belytscho et al. [7] presented the importance of XFEM with level set methods and its applications, especially for complicated geometry, crack growth and moving interface. XFEM has increased the capability of the finite element method to solve the fracture mechanics problems which are very difficult for the conventional finite element method. Asadpour et al. [8] used XFEM for modelling and analysing of 2D orthotropic media where crack–tip displacement fields are added to enrich the finite element with the partition of unity. Here, crack growth is implemented without remeshing and MMSIF is calculated by interaction integral (M-integral). LI and Chen [9] developed formulations for the prediction of the variations of stress intensity factors, using the transformation toughening theory. Mousavi and Sukumar [10] proposed the Gaussian integration method to investigate the integrals weak form by the XFEM. Here, a point elimination algorithm is adopted, where quadrature has a minimal number of Gauss points. Kumar et al. [11] investigated the dynamic response and dynamic SIF at crack-tip in the XFEM. And ramp function is used in the Heaviside function to evaluate the complex problem. Belytschko and Gracie [12] studied a method to calculate Peach-Koehler force which is based on XFEM where discontinuities have been modeled by Volterra dislocation model for a more complex problem. Sukumar et al. [13] presented work on modelling holes and inclusions by level set in XFEM, where they described the modelling the internal boundaries where local enrichment functions in the generalized finite element method for modelling corners in two dimensions are used. Wiroj et al. [14] presented an adaptive finite element method to determine stress intensity factors $K_1$ and $K_2$ of the polycarbonate edge crack plate with with considering void and inclusion using FEM and found better results as compared to photoelastic method. Jiang et al. [15] investigated the crack growth with the interaction of various discontinuities like crack, void, hard and soft inclusion by XFEM. Natrajan et al [16] presented a numerical analysis of inclusion crack interaction by XFEM where both inclusion and crack are modelled within the XFEM framework and found better accuracy and flexibility in the results. Sharma [17] simulated the material discontinuities interaction effect on fracture behaviour by XFEM. Stress intensity factor (SIF) is calculated by the interaction integral method and effect of crack orientation under mechanical loading considering hard inclusion.

Ismael Rivero Arévalo et al [18] is evaluated the Abaqus XFEM capabilities to analyze the crack growth of aeronautical structure and it could be a beneficiary in industrial application. M. Al-Khalil et al [19] used the Abaqus for investigating the damage reduction in composite structure during bird impact. Kaman [20] investigated the effect of fibre orientation angle on the laminated composite plate through the experimental study. He has also used Ansys for numerical study. Lal et al [21] presented the study of stochastic analysis of the SIF for SEN laminated composite plate using Matlab. Xu et al [22] implemented the Abaqus software for crack problem through the extended finite element method.

From the above literature study, it is observed that researchers are focused on Abaqus implementation of composite structure, fracture analysis through the XFEM with the interaction of crack under different loading condition.
In this present work main focus is Abaqus implementation to analyze the stress intensity factor, von misses stress and displacement for SEN laminated composite plate under tensile loading. It is investigated the SIF $K_I$, von misses stress and displacement for various case studies, like various fibre orientation angle under tensile loading.

2. Mathematical Formulation

2.1. Calculation of critical stress intensity factor based on the experimental results

The stress intensity factor for the Mode I can be written as,

$$ (K_I) = \sigma f\left(\frac{b}{w}\right)\sqrt{\pi b} $$

(1)

where, $\sigma$ is the applied normal stress and $f\left(\frac{b}{w}\right)$ is a geometrical factor. $K_I$ provides the severity of the crack tip environment and it is logical to characterize resistance to fracture by a critical value that is $K_{IC}$. When the applied normal stress reaches the failure stress, the stress intensity factor $K_I$ becomes ($K_{IC}$)EXP, the critical stress intensity factor, which is taken as the fracture toughness of the composite material. The value of critical stress intensity factor ($K_{IC}$) is given by,

$$ (K_{IC}) = \sigma f\left(\frac{b}{w}\right)\sqrt{\pi b} $$

(2)

where, $\sigma_f$ is the applied maximum normal stress or failure stress. The geometrical factor for single edge notched specimens [20] is,

$$ f\left(\frac{b}{w}\right) = 1.12 - 0.231\left(\frac{b}{w}\right) + 10.55\left(\frac{b}{w}\right)^2 $$

$$ -21.72\left(\frac{b}{w}\right)^3 + 30.39\left(\frac{b}{w}\right)^4 $$

(3)

2.2. Numerical study of Displacement Correlation Method (DCM) [20]

The DCM is used to evaluate the fracture toughness from the failure load. For singular finite element model at the crack tip Figure 1 the crack opening displacement (COD) and the crack sliding displacement (CSD) are given by,

Figure 1: Finite element model at the crack tip
where, \( b \) is the crack length and \( \Delta b \) is the characteristic length of the quarter point crack tip elements, \( \Delta u_{34}, \Delta u_{56} \) and \( \Delta v_{34}, \Delta v_{56} \) are the relative displacements with respect to crack-tip in the x, y-directions, respectively. Displacements at the crack tip can be written as,

\[
\begin{align*}
    u &= K_i \frac{2\pi}{\mu_1} \operatorname{Re} \left[ i \frac{\mu_1 p_2 - \mu_2 p_1}{\mu_1 - \mu_2} \right] + K_{ii} \frac{2\pi}{\mu_1} \operatorname{Re} \left[ i \frac{\mu_1 q_3 - q_2}{\mu_1 - \mu_2} \right], \\
    v &= K_i \frac{2\pi}{\mu_2} \operatorname{Re} \left[ i \frac{\mu_2 p_2 - \mu_1 p_1}{\mu_2 - \mu_1} \right] + K_{ii} \frac{2\pi}{\mu_2} \operatorname{Re} \left[ i \frac{\mu_2 q_3 - q_2}{\mu_2 - \mu_1} \right],
\end{align*}
\]

where, \( r \) is the distance from the crack tip along the x axis, \( \mu_1 \) and \( \mu_2 \) are the eigenvalue of the field equation with positive imaginary part of the orthotropic body calculated from,

\[
S_{11}\mu^4 + (2S_{12} + S_{66})\mu^2 + S_{22} = 0
\]

where, \( S_{ij} \) (\( i, j = 1, 2, 6 \)) are the compliance coefficients of the composite material

\[
\varepsilon_{ij} = S_{ij}\sigma_j (i, j = 1, 2, 6)
\]

Here \( \varepsilon_{ij} \) (\( i, j = 1, 2, 6 \)) is the strain and \( \sigma_j \) is the stress vector \( p_k \) and \( q_k \) (\( k = 1, 2 \)) are given by

\[
\begin{align*}
    p_k &= S_{11}\mu_k^2 - S_{16}\mu + S_{12}, \\
    q_k &= S_{12}\mu_k^2 + S_{22} - S_{36}.
\end{align*}
\]

By using Eqs. (5-8), formulas for pure Mode I stress intensity factor (\( K_i \)) and Mode II stress intensity factor (\( K_{ii} \)) can be obtained:

\[
\begin{align*}
    K_i &= \frac{1}{4} \frac{2\pi}{\Delta b} \left( \frac{D(4\Delta u_{34} - \Delta u_{56}) - B(4\Delta v_{34} - \Delta v_{56})}{AD - BC} \right), \\
    K_{ii} &= \frac{1}{4} \frac{2\pi}{\Delta b} \left( \frac{A(4\Delta v_{34} - \Delta v_{56}) - C(4\Delta u_{34} - \Delta u_{56})}{AD - BC} \right).
\end{align*}
\]

where \( A, B, C \) & \( D \) are given by,

\[
\begin{align*}
    A &= \operatorname{Re} \left[ i \frac{p_2 - p_1}{\mu_1 - \mu_2} \right], \\
    B &= \operatorname{Re} \left[ i \frac{p_2 - p_1}{\mu_1 - \mu_2} \right], \\
    C &= \operatorname{Re} \left[ i \frac{q_3 - q_2}{\mu_1 - \mu_2} \right], \\
    D &= \operatorname{Re} \left[ i \frac{q_3 - q_2}{\mu_1 - \mu_2} \right].
\end{align*}
\]

2.3. Analysis of Stress Intensity Factor Laminate Composite Plate by Abaqus through the XFEM
To perform the analysis of stress intensity factor, the extended finite element analysis is used. The Abaqus software is used to perform the extended finite element analysis (XFEA).
Input options in the Abaqus for material properties are $E$ (Young’s modulus) and $v$ (Poisson’s ratio) for homogeneous linear isotropic material, $E_x$ and $E_y$ (Young’s moduli in the $x$- and $y$-directions, respectively), $\nu_{xy}$ (Poisson’s ratio) and $G_{xy}$ (shear modulus) for homogeneous linear orthotropic material on the plane. This element has higher order interpolation functions which valid to more accurate results. Material properties of laminates are calculated by using the classical laminated plate theory and are supplied to the program [20]. In a symmetric laminated composite plate stress–strain relation is:

$$
\begin{bmatrix}
    N_X \\
    N_Y \\
    N_{XY}
\end{bmatrix} =
\begin{bmatrix}
    A_{11} & A_{12} & A_{16} \\
    A_{21} & A_{22} & A_{26} \\
    A_{16} & A_{26} & A_{66}
\end{bmatrix}
\begin{bmatrix}
    \varepsilon_x \\
    \varepsilon_y \\
    \varepsilon_{xy}
\end{bmatrix}
$$

where,

$$N_X = \sigma_x t, \quad N_Y = \sigma_y, \quad N_Z = \sigma_y \quad \text{and} \quad A_j = \sum_{k=1}^{n_k} Q_{ki} h_k$$

In which $Q_{ki}$ are reduced stiffnesses and $h_k$ the thickness of the $k$th layer. The longitudinal ($E_x$), transverse ($E_y$), shear ($G_{xy}$) moduli, and Poisson’s ratio ($\nu_{xy}$) of laminated composite plates are respectively equal to

$$
E_x = \frac{N_x / t}{\varepsilon_x}, \quad E_y = \frac{N_y / t}{\varepsilon_y}, \quad G_{xy} = \frac{T_{xy} / t}{\gamma_{xy}}, \quad \nu_{xy} = \frac{-\varepsilon_y}{E_x}
$$

And it can be derived by inverting the $3 \times 3$ stiffness matrix $[A]$.

$$
[A]^{-1} =
\begin{bmatrix}
    A_{11}' & A_{12}' & A_{16}' \\
    A_{21}' & A_{22}' & A_{26}' \\
    A_{16}' & A_{26}' & A_{66}'
\end{bmatrix}
$$

Longitudinal modulus ($E_x$) for the laminated composite plates as,

$$
E_x = \frac{N_x / t}{\varepsilon_x} = \frac{1}{A_{11}'} \left( A_{11}' A_{12}' - 2A_{12}' A_{16}' + A_{16}' A_{66}' - A_{16}' A_{66}' - A_{16}' A_{66}' \right) / t
$$

$E_y$ and $G_{xy}$ expressed as,

$$
E_y = \frac{N_y / t}{\varepsilon_y} = \frac{1}{A_{22}'} \left( A_{22}' A_{12}' - 2A_{12}' A_{16}' + A_{16}' A_{66}' - A_{16}' A_{66}' - A_{16}' A_{66}' \right) / t
$$

$$
G_{xy} = \frac{T_{xy} / t}{\gamma_{xy}} = \frac{1}{A_{66}'} \left( A_{66}' A_{12}' - 2A_{12}' A_{16}' + A_{16}' A_{66}' + A_{16}' A_{66}' \right) / t
$$

$E_{xy}$ can be written as,

$$
E_{xy} = \sqrt{E_x \cdot E_y}
$$

By using Eq. (19-21) respectively, the Poisson’s ratio ($\nu_{xy}$) can be calculated from $-\frac{A_{12}'}{A_{11}'}$. Finally, the compliance coefficients of composite material $S_{ij}$ ($i, j = 1, 2, 6$) for plane element can be written as
Eq. (6) can be written by using calculated material properties ($E_x$, $E_y$, $G_{xy}$ and $m_{xy}$) of laminated plates,

$$S_y = \begin{bmatrix} S_{11} & S_{12} & S_{16} \\ S_{12} & S_{22} & S_{26} \\ S_{16} & S_{26} & S_{66} \end{bmatrix} \equiv S_y = \begin{bmatrix} \frac{1}{E_x} & 0 & -\nu_{xy} \\ 0 & \frac{1}{E_y} & 0 \\ -\nu_{xy} & 0 & \frac{1}{G_{xy}} \end{bmatrix}$$ (23)

Material properties required as an input are taken from following Table 1. For every lamina configuration the properties are different in X and Y axis. The properties required for orthotropic material are $E_x$, $E_y$, $E_z$= Young’s modulus of elasticity in x, y, z direction respectively (MPa), $G_{xy}$, $G_{yz}$, $G_{xz}$= Shear modulus in xy, yz, xz plane respectively (MPa) and Poisson’s ratios in xy, yz, xz plane.

Table 1: Laminate Material Properties

| Fibre Orientation $[0^\circ/\theta^\circ]$ | $E_x$ (Mpa) | $E_y$ (Mpa) | $G_{xy}$ (Mpa) | $\nu_{xy}$(-) | $E_{xy}$ (Mpa) |
|----------------------------------------|-------------|-------------|----------------|---------------|----------------|
| $[0^\circ/15^\circ/15^\circ/0^\circ]$ | 1.191 x 10^5 | 1.076 x 10^4 | 6.514 x 10^3 | 0.511 | 3.580 x 10^4 |
| $[0^\circ/30^\circ/30^\circ/0^\circ]$ | 9.771 x 10^4 | 1.143 x 10^4 | 1.184 x 10^4 | 0.546 | 3.342 x 10^4 |
| $[0^\circ/45^\circ/45^\circ/0^\circ]$ | 9.238 x 10^4 | 1.349 x 10^4 | 1.050 x 10^4 | 0.474 | 3.530 x 10^4 |
| $[0^\circ/60^\circ/60^\circ/0^\circ]$ | 9.129 x 10^4 | 1.922 x 10^4 | 6.572 x 10^3 | 0.329 | 4.189 x 10^4 |
| $[0^\circ/75^\circ/75^\circ/0^\circ]$ | 9.180 x 10^4 | 4.007 x 10^4 | 4.311 x 10^3 | 0.139 | 6.065 x 10^4 |
| $[0^\circ/90^\circ/90^\circ/0^\circ]$ | 9.230 x 10^4 | 9.230 x 10^4 | 3.638 x 10^3 | 0.038 | 9.230 x 10^4 |

3. Result and Discussion

3.1. Validation study for Single Edge Notched (SEN) laminated composite plate using Abaqus

The SEN 3D laminated composite plate under tensile loading is considered for a validation study. To analyze the SIF KI for SEN composite plate, define the problem model as shown in Figure 2.

Figure 2: The SEN laminated composite plate under tensile loading

To analyze the SIF $K_I$ for 3D laminated composite plate used the material properties as per Table 1 and for geometry dimension and loading as per Table 2. Average failure load is applied on the top surface of plate and applied boundary condition at the bottom surface as fixed as shown Figure 2. Abaqus 6.13 is used to analyze the SIF $K_I$, Von misses stress and displacement for SEN laminated
composite plate. For meshing chosen Hex element shape, sweep technique and medial axis algorithm. Total numbers of nodes are 5238 and total number of elements is 1540.

The results are presented in three main sections: (a) experimental results \( (K_{IC})_{\text{EXP}} \) with using the empirical relation for SEN specimens, (b) numerical simulation results \( (K_{IC})_{\text{PLANE}} \) and \( (K_{IC})_{\text{SHELL}} \) of two applied elements Plane82 and Shell99 from experimental failure load (c) analysis of \( (K_{IC})_{\text{ABAQUS}} \) from experimental failure load, respectively.

Table 2: Experimental failure loads for different fiber orientation angle

| Fibre Orientation Angle \( \theta \) (°) | Crack length \( b \) (mm) | Width of plate \( w \) (mm) | Total laminate thickness \( t \) (mm) | Average failure load (N) | Failure stress (MPa) |
|---------------------------------------|--------------------------|---------------------------|-----------------------------|-------------------------|------------------|
| 15                                    | 9                        | 18                        | 1.235                       | 6516.112                | 293.077          |
| 30                                    | 9                        | 18                        | 1.235                       | 5145.807                | 231.480          |
| 45                                    | 9                        | 18                        | 1.235                       | 4722.581                | 212.442          |
| 60                                    | 9                        | 18                        | 1.235                       | 5229.024                | 235.224          |
| 75                                    | 9                        | 18                        | 1.235                       | 4193.829                | 188.656          |
| 90                                    | 9                        | 18                        | 1.235                       | 3680.767                | 165.577          |

As per experiment result the maximum average failure load is 6516.112 N for \([0°/15°]_s\) lamina configuration. Generally, failure load tends to decrease with the increasing fiber orientation angle \( \theta \). When \( \theta \) reaches 90°, the minimum failure load is obtained (3680.767 N). The SIF is getting the maximum for \([0°/15°]_s\) lamina configuration in experimental, FEA through Ansys and by Abaqus and minimum for \([0°/90°]_s\) lamina configuration as shown in Figure 3. As shown in Figure 3, Abaqus results are getting quite nearer to the experimental result.

Figure 3: Validation of SIF \( K_I \) of SEN laminated composite plate for various fibre orientation angle (\( \theta \))

Figure 4 and Figure 5 shows the Von Mises stress and displacement of the SEN laminated composite plate respectively from Abaqus 6.13. The maximum value of Von Mises stress and displacement are getting at fibre orientation angle 15° and decrease as per for fibre orientation angle 15°, 30°, 45°, 60°, 75°, 90° increase respectively.
3.2. Analyze the Single Edge Notched Composite Plate for variable fibre orientation angle using Abaqus

To analyze the SIF $K_I$ for SEN 3D laminated composite plate defined the problem model as shown in Figure 2. The material properties are considered as per Table 1 and for the geometry dimension and loading as per Table 2. Average failure load is applied on the top surface of plate and applied boundary condition at the bottom surface is fixed as shown Figure 2. Abaqus 6.13 is used to analyze the SIF $K_I$ for SEN 3D laminated composite plate, in meshing chosen Hex element shape, sweep technique and medial axis algorithm. Total numbers of nodes are 2944 and total number of elements is 630.

3.2.1. The SIF $K_I$ result analysis for variable fibre orientation angle.

The results are presented in four main sections: (a) experimental results ($K_{IC}^{exp}$) with using the empirical relation for SEN specimens, (b) numerical simulation results ($K_{IC}^{PLANE}$ and $K_{IC}^{SHELL}$ of two applied elements Plane82 and Shell99 from experimental failure load (c) analysis of ($K_{IC}^{ABAQUS}$ from experimental failure load and (d) analysis of ($K_{IC}^{MATLAB}$ from experimental failure load, respectively.

As per author considered result of the maximum average failure load is 6516.112 N for $[0°/15°]_S$ lamina configuration. Generally, failure load tends to decrease with the increasing fiber orientation angle $\theta$. When $\theta$ reaches 90°, minimum failure load is obtained (3680.767 N). The SIF $K_I$ is getting maximum for $[0°/15°]_S$ lamina configuration in experimental, FEA through Ansys, MATLAB and Abaqus by XFEM and minimum for $[0°/90°]_S$ lamina configuration as shown in Figure 6. The Abaqus results are getting very nearer to Matlab result and more improved result as compare to Ansys result.
3.2.2. The von misses stress and displacement result analysis for variable fibre orientation angle.

The result of Von misses stress and displacement for the SEN laminated composite plate are getting as shown Figure 7 and Figure 8 respectively from Abaqus 6.13. The maximum value of Von misses stress and displacement are getting at fibre orientation angle 15° and decrease as per for fibre orientation angle 15°, 30°, 45°, 60°, 75°, 90° increase respectively.

4. Conclusions
The present work is analyzed, the Abaqus implementation to analyze the effect of fibre orientation angle on stress intensity factor for the SEN 3D laminate composite plate under tensile loading. The following evaluations are found during this study;

- The SIF $K_I$ results through the Abaqus software are getting nearer to experimental results [20] for taken case.
- The SIF $K_I$ results through the Abaqus software are getting very nearer to the Matlab results and improved as compared to Ansys software result for taken case.
- The SIF $K_I$ is getting maximum for $[0^\circ/15^\circ]$ lamina configuration and minimum for $[0^\circ/90^\circ]$. And found decreasing nature by increase the fibre orientation angle increased.
- The maximum value of Von misses stress and displacement are getting at fibre orientation angle $15^\circ$ and decrease as per for fibre orientation angle $15^\circ$, $30^\circ$, $45^\circ$, $60^\circ$, $75^\circ$, $90^\circ$ increase respectively.
- Through this study, it is found that the Abaqus implementation through the XFEM results are improved and would be more applicable for industrial application as well in research.

References

[1] Sih G C 1973 Eng. Fract. Mech. 5, 1037-1040.
[2] Ke J S and Liu H W 1973 Eng. Fract. Mech. 5, 187-202.
[3] Chien W Z, Xie Z C, Gu Q L, Yang Z F and Zhou C T 1982 Eng. Fract. Mech. 16, 95-103.
[4] Belytschko T, Chen H, J Xu, and Zi G 2003 Int. J. Numer. Meth. Engng 58, 1873-1905.
[5] Sukumar N and Prevost J H 2003 Int J Solids Struct. 40, 7513-753.
[6] Belytschko T and Gracie R 2007 International Journal of Plasticity 23, 1721-1738.
[7] Sukumar N, Chop D L, Moes N and Belytscko T 2001 Appl. Mech. 190, 6183-6200.
[8] Limtrakarn W and Dechaumphai P 2011 T Can Soc. Mech. Eng. 35, 355-368.
[9] Jiang S, Du C and Gu C 2014 Struct. Eng. Mech. 49-5, 597-618.
[10] Sharma K 2014 J. Solid Mech. 6-4, 410-421.
[11] Arevalo I R and Martin J G E 2015 SIMULIA Community Conference Proceedings 23-36.
[12] Khalil M A, Rigby R and Kirtil E 2015 SIMULIA Community Conference Proceedings 48-62.
[13] Kaman M O 2011 Eng. Fract. Mech. 78, 2531-2534.
[14] Lal A and Kapania R K 2013 54th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference 4, 2653-2675.
[15] Xu Y and Yuan H 2009 Eng. Fract. Mech. 76, 165–181.