The Implementation of Regularized Extended Finite Element Method (Rx-FEM) in ABAQUS

Yu-Jui Liang,1 and Endel V. Iarve.2

The University of Texas at Arlington, Arlington, TX, 76019, USA

J. Scott McQuien3

University of Texas at Arlington Research Institute, Fort Worth, TX, 76118, USA

Rx-FEM is a discrete damage modeling (DDM) method, which represents an approach to progressive failure modeling in composites when multiple individual damage events such as matrix cracks and delamination are introduced into the model via displacement discontinuities. Rx-FEM is a variant of the eXtended Finite Element Method (x-FEM), where a continuous approximation is used in place of the Heaviside step function. To date, this methodology has been implemented in BSAM, an in-house program, and extensively applied to static and fatigue analysis of laminated composite structures. The regularization of the Heaviside step function offers unique possibility for implementation of Rx-FEM in commercial software by using superimposed native elements of the parent software. The proposed implementation capitalizes on utilization of original Gauss integration schema in Rx-FEM even after the enrichment is introduced to accommodate a mesh independent crack. Thus, the enrich displacement field is represented by superposition of native ABAQUS elements such as CPS4, which is used in the present work. Several examples of unnotched and open hole composite unidirectional coupons are considered, including the splitting phenomenon of an axially loaded open-hole coupon. The method correctly predicts separation and fiber direction stress concentration reduction as a result of the splitting.

I. Introduction

Discrete Damage Modeling (DDM) is based on direct simulation of displacement discontinuities associated with individual instances of matrix cracking occurring inside the composite plies, and delaminations at the interfaces between the plies. Some DDM methods employ techniques for mesh independent modeling of cracks based on variants of eXtended Finite Element Method (x-FEM) [1] and its regularized implementation (Rx-FEM) [2, 3] in particular. The regularized formulation deals with continuous enrichment functions, and replaces the Heaviside step function with a continuous function changing from 0 to 1 over a narrow volume of the so called gradient zone. The formalism tying the volume integrals in the gradient zone to surface integrals in the limit of mesh refinement was discussed in [2]. The simulation begins without any initial matrix cracks, which are then inserted based on a failure criterion during the simulation. The propagation of each MIC is performed by using the cohesive zone formulation. To date this methodology is implemented in BSAM, an in-house finite element code, and extensively applied to static and fatigue analysis of laminated composite structures [4, 5].

Commercial finite element software has the advantages including the established markets, reputations, and its many additional functions which are impractical to implement again in a research code. The implementation of DDM

1 Ph.D. student, Department of Mechanical and Aerospace Engineering, Non-Member.
2 Professor, Department of Mechanical and Aerospace Engineering, Member.
3 Research Engineer, UTARI/IPPM, Non-Member.
methodologies in commercial code is significantly more challenging than implementation of Continuum Damage modeling (CDM) methodologies, where the damage is represented as a change of element stiffness properties. To date several implementations of x-FEM methodology into commercial software and ABAQUS [6] in particular have been reported [7, 8, 9]. What is common in these implementations is that a user defined element (UEL) was created encapsulating the kinematics of the x-FEM element pair, which is produced after the degrees of freedom (d.o.f) are enriched. This type of implementation allows wide freedom in customizing the integration scheme of the cracked elements and introducing either phantom nodal d.o.f. [9] or d.o.f. representing displacements at the crack surface and element edge intersection [8]. The disadvantages of this approach are that this type of implementation allows to take advantage only of a very small subset of features of parent software package. Thus this approach allows to take advantage of the solver portion of the code, while even visualization represents challenges, not to speak of more essential feature such as contact, etc.

In this article a different implementation path is proposed for Rx-FEM where the regularization of the Heaviside step function offers unique possibility of implementation superimposing native parent software elements. The proposed implementation capitalizes on utilization of original Gauss integration schema in Rx-FEM even after the enrichment is introduced to accommodate a mesh independent crack. Thus, the enriched displacement field is represented by superimposed native ABAQUS elements such as CPS4 used in this article. The discussion will begin with a brief overview of the Rx-FEM theory and Cohesive Zone Model (CZM). The implementation method of Rx-FEM in ABAQUS will be discussed next. Examples of composite fracture simulation will conclude the paper.

II. Rx-FEM Theory

The DDM methodology is based on Mesh Independent Crack (MIC) technique termed as Rx-FEM. The simulation begins without any initial matrix cracks, which then are inserted based on a failure criterion during the simulation. A level set function of the signed distance function of the crack surface in introduced and a Heaviside functions associated with this level set represents the displacement discontinuity due to the crack. In Rx-FEM, this step function is replaced by a continuous function changing from 0 to 1 over a narrow volume called gradient zone. The Rx-FEM formulation [2, 3] uses displacement approximation shape functions to approximate the step function. An advantage of Rx-FEM is in maintaining a fixed Gauss integration schema throughout the analysis without regard to location and direction of the crack created during the analysis. The elements, as opposed to x-FEM are not partitioned since the crack surface in not inside an element, but rather spread over several neighbor elements. The approximation of the Heaviside step function is:

\[
\tilde{H}(x) = \sum_{a=1}^{N_{\text{nodes}}} N_a(x) \tilde{H}_a
\]

(1)

where \(N_{\text{nodes}}\) is the total number of nodes, \(x\) is the spatial coordinate, \(N_a\) is the shape function associated with node \(a\), summed over all elements sharing he node, and the coefficients \(\tilde{H}_a\) are calculated as follows:

\[
\tilde{H}_a = \frac{1}{2} \left[ 1 + \frac{\int_v N_a(x)f(x)dv}{\int_v N_a(x)|f(x)|dv} \right]
\]

(2)

where \(x\) is the global coordinate, \(f\) is the signed distance function of the crack surface. The principle of minimum potential energy for the enriched displacement field can be written as

\[
\delta(\tilde{H}(x)W(\nu^{(1)}) + (1 - \tilde{H}(x))W(\nu^{(2)}) + M) = 0.
\]

(3)

where \(W(\nu^{(1)})\) and \(W(\nu^{(2)})\) are strain energy on the two sides of the crack surface and \(M\) is the cohesive energy of the Mesh Independent Crack (MIC). In this work, a bilinear cohesive zone model (CZM) [10] in terms of opening
tractions and opening displacements will be used. Initially, the surface traction is related to the interfacial displacement jump by a high penalty stiffness. If the displacement jump exceeds a critical traction, then the cohesive softening will initiate. Complete separation is achieved when the displacement jump exceeds the final displacement. The area under the traction-displacement curve is equal to the fracture toughness. In the application of CZM to Rx-FEM, the displacement jump is evaluated in the volume of the gradient zone, where $$|\nabla H| > 0$$, as opposed to the traditional crack surface based evaluation. Next an equivalent crack surface energy is calculated by using the following volume integral

$$M = \sum_{i=1}^{N_{el}} \int_{V_i} g \cdot |\nabla H| \cdot dV$$  \hspace{1cm} (4)

where the M is cohesive energy associated with crack, $$v_i$$ is the element domain, g is pointwise cohesive energy corresponding to the displacement jump in the gradient zone, and $$N_{el}$$ is the number of elements.

III. Rx-FEM implementation in ABAQUS

The implementation in commercial software ABAQUS is based on representing the displacement field corresponding to the first two terms of Equation (3) by using native displacement based elements of the software of choice. Fig. 1 shows two CPS4 elements containing nodes 1-4 and 5-8 respectively, which correspond to the first and second terms on Equation (3). Node 1-4 and 5-8 are pairwise coincident. User material capability UMAT is employed to multiplying the material elasticity matric by $$\mathbf{R}(x)$$ and $$\left(1 - \mathbf{R}(x) \right)$$ respectively. We shall call these elements as original and duplicate and note that these elements are defined in the input deck and active through the analysis. It introduces an overhead, which can be avoided in standalone implementation but not commercial software implementation. Note that this overhead is present in any x-FEM type implementation where the respective UEL contains additional (phantom or floating) d.o.f. Fig. 2 shows the schematics of regularized Heaviside step function $$\mathbf{R}(x)$$ for the original and duplicated elements at hand, and the gradient of regularized Heaviside step function $$\nabla \mathbf{R}$$ within a single central crack region. In additional, a user element UEL is created to accommodate the cohesive tractions M which is the third term of Equation (3). Note that this UEL does not require visualization in order to post-process the results since all the kinematics is accommodated by ABAQUS native elements and is analogous to interface UEL often created by users for implementation of specific cohesive laws. The UEL contains nodes 1-8 and calculates the cohesive traction as a result of separation, which can be obtained as

$$\Delta u(x, y) = (N)\{u_1 - u_5 \ u_2 - u_6 \ u_4 - u_8 \ u_3 - u_7\}^T \quad \text{and} \quad \Delta v(x, y) = (N)\{v_1 - v_5 \ v_2 - v_6 \ v_3 - v_7 \ v_4 - v_8\}^T$$

where the shape function vector $$\{N\}$$ is that of the original or duplicate element, which are identical. In the present work the cracks are inserted parallel to the fiber direction and therefore the normal and shear components of the displacement separation vector can be written as

$$\begin{align*}
\Delta u_n &= \sin \theta \Delta u - \cos \theta \Delta v \\
\Delta u_t &= \cos \theta \Delta u + \sin \theta \Delta v
\end{align*}$$  \hspace{1cm} (5)

where $$\theta$$ is the fiber orientation with respect to the x axis of the global coordinate system. A specific form of the traction separation law of the implemented CZM is then used to calculate a pointwise traction vector $$\{t\}$$, which prior to damage initiation is defined by high penalty coefficients $$K_1$$ and $$K_2$$ as

$$\{t_n\} = \begin{Bmatrix} K_1 \Delta u_n \\ K_2 \Delta u_t \end{Bmatrix}$$  \hspace{1cm} (6)

The formulation of the UEL requires a residual load vector often denoted as $$\{\text{RHS}\}$$. In the present case it will contain 16 components corresponding to 8 nodes with x and y d.o.f. each. Adopting the following sequence of the d.o.f.

$$\{u_1 \ u_2 \ u_3 \ u_4 \ u_5 \ u_6 \ u_7 \ u_8 \ v_1 \ v_2 \ v_3 \ v_4 \ v_5 \ v_6 \ v_7 \ v_8\}$$  \hspace{1cm} (7)
One obtains the following expression for \( \{ \text{RHS} \} \)

\[
\{ \text{RHS} \} = \int_{V_{\text{elem}}} \nabla H \left\{ \begin{array}{l}
(\sin \theta_t n + \cos \theta_t t) [N] \\
(-\cos \theta_t n + \sin \theta_t t) [N] \\
(-\sin \theta_t n - \cos \theta_t t) [N] \\
(\cos \theta_t n - \sin \theta_t t) [N]
\end{array} \right\} dV
\]

The expression for the tangent stiffness matrix can be obtained by standard methods. When a tensile load is applied in the axial direction, the cohesive elements with non-zero \( \nabla H \) value will start to generate a displacement jump. When the cohesive zone begins softening, the original and duplicate elements will move against each other and create separation. The energetics of such separation has been shown previously to closely approximate the fracture mechanics formulation [2, 3].

Fig. 3 shows the Rx-FEM implementation in ABAQUS framework. The mesh-independent crack (MIC) related information is stored in Rx-FEM data structure, which is initiated at the first call of UEXTERNALDB before the first increment. In the beginning of the calculation \( \bar{H}(x) = 1 \) for original elements, and the elastic stiffness of the duplicated element \( \left(1 - \bar{H}(x)\right) \) is respectively 0. The cohesive UEL is initially set to provide a tie between the original and duplicated elements and making them move move together. Since the stiffness of the latter is 0 there is no resistance and/or separation.

On all subsequent calls of UEXTERNALDB at the end of each increment, the stress tensor calculated by UMAT is evaluated by failure criterion. If there is any element reaching failure, one or more cracks will be inserted and the MIC related information will be updated. It includes calculation of the step function \( \bar{H}(x) \) and its gradient \( \nabla \bar{H} \) in the affected elements. When \( |\nabla \bar{H}| > 0 \) the UEL formulation [10] is invoked and the stiffness of the original and duplicate elements is deferent and not 0. In this case they start moving independently and when the applied load is sufficient to overcome cohesive forces the separation occurs. The examples following in the next section illustrate the above implementation.
Fig. 2 The schematics of Regularized Heaviside step function and its gradient within the crack region

Fig. 3 The framework of Rx-FEM implementation in ABAQUS
IV. Results and Discussion

A. Isotropic Nine Elements Test

First a row of 9 elements with a vertical central crack was simulated as shown in Fig. 4(a). The reason why the final configuration contains 10 elements is that the visualization is performed within ABAQUS by using state variable parameters (SDVs), which easily allows to show or hide the element which satisfies a certain criterion defined by user in ABAQUS/CAE. Fig. 4(b) shows the only element which have \( \tilde{H}(x) > 0.5 \) at any integration point in the element, where \( \tilde{H}(x) \) is the regularized Heaviside step function. In this case, low initiation strength and high fracture toughness, CZM properties were considered for easy convergence. The stiffness properties were isotropic and plane stress conditions imposed via CPS4 elements.

![Fig. 4 The displacement field of 9 elements with vertical central crack (a) pristine, (b) failure](image)

B. Composite Off-Axis Tension/Compression Test

The geometry of unidirectional ply in this test is shown in Fig. 5(a). The ratio of ply length to width (L/W) is 5, and the structured mesh size is 1mm. The ply level properties used for the analysis are shown in Table 1 including the stiffness, strength properties, as well as the input parameters for the cohesive law. The ply-level strength properties are used only for MIC initiation. Crack initiation is determined by checking the ply-level failure criterion, LaRC04 [11], at each integration point at the end of each increment. If the crack initiation criterion is met or exceeded, one or more MIC are inserted at that location. The crack is assumed to be planar, parallel to the fiber orientation, and to extend to the edges of the model; however, it is introduced in the fully closed state. Fig. 5(b) to 5(g) show the crack insertion of six different lamina fiber angles (15°, 30°, 45°, 60°, 75°, 90°) when reaching failure criterion of crack initiation, and show the visualization of \( \tilde{H}(x) \) distribution through the entire model which means the crack region. Fig. 6 shows the comparison of the predicted failure strength and several classic failure criteria envelopes. The predicted values are in good agreement with Sun criteria and follow the trends characteristic to CFRP.

![Fig. 5 Off-Axis Tension/Compression Test. (a)geometry, (b)15°, (c)30°, (d)45°, (e)60°, (f)75°, (g)90°](image)
Table 1. Unidirectional stiffness and strength properties for IM7/8552

<table>
<thead>
<tr>
<th>Property</th>
<th>IM7/8552</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_{11}$ (MPa)</td>
<td>157600</td>
</tr>
<tr>
<td>$E_{22}$ (MPa)</td>
<td>8977</td>
</tr>
<tr>
<td>$G_{12}$ (MPa)</td>
<td>5000</td>
</tr>
<tr>
<td>$v_{12}$</td>
<td>0.32</td>
</tr>
<tr>
<td>$Y_t$ (MPa)</td>
<td>92</td>
</tr>
<tr>
<td>$S$ (MPa)</td>
<td>89</td>
</tr>
<tr>
<td>$G_{IC}$ (KJ/m²)</td>
<td>0.24</td>
</tr>
<tr>
<td>$G_{IIC}$ (KJ/m²)</td>
<td>0.739</td>
</tr>
</tbody>
</table>

C. Composite Open Hole Tension (OHT) Coupons

A rectangular plate with a central hole is considered. The diameter of the hole with 6.375mm (1/4 inch) and the ratio of width to hole diameter (W/D) and length to diameter (L/D) are 6 and 18, respectively. Three examples with 90°, 45° and 0° fiber orientations were examined. The structured mesh size is 1mm for 90° and 45° plate and 0.5mm for 0° plate. The stresses and displacements in the pristine and failed specimens are examined. In the case of the 90° and 45° the normal transverse component $\sigma_{22}$ and the $u_x$ displacements are shown, whereas for the 0° the fiber direction $\sigma_{11}$ stress and $u_x$ are shown. Fig. 7 and 8 display the results for the 90° and 45° specimens respectively. As one can see the crack insertion location coincides with the transverse stress concentration location. In both cases two cracks, one on each side of the hole get inserted and after load increase the specimen separates immediately after insertion. After separation the stress in the specimen vanishes and is not shown. The displacement distribution after separation indicates a complete two piece failure as shown. Note that the ability to readily display the displacement, stress and...
separation of the specimens by using standard ABAQUS visualization options can only be achieved by using native ABAQUS elements for formulation.

Next a 0° coupon was considered. In this case 4 longitudinal cracks were inserted at locations corresponding to LaRC04 failure criteria at approximately ±78° angle with respect to the loading direction from at the edge of the hole. The axial displacement field before the crack insertion and after separation are shown in Fig. 9. As one can see the specimen separates into 4 part. The outer ligaments and two middle ligaments. The latter ligaments are pulled out, whereas the outer ligament are displaying linear displacement patterns. The stress field corresponding to such kinematics is represented in Fig. 10. Before cracking (splitting) one observes a high fiber stress concentration at the ±90° locations at the hole edge. After separation, as seen on Fig. 10(b), the top and bottom ligaments are uniformly loaded with no stress concentration, whereas the middle ligaments are completely unloaded. The splitting phenomena is important for understanding of various failure patterns and strength effects in composite laminates and needs to be accurately modeled on ply level.

![Fig. 7 OHT specimen with two 90° cracks, (a) pristine stress and (b) displacement, (c) failure displacement](image1)

![Fig. 8 OHT specimen with two 45° cracks, (a) pristine stress and (b) displacement, (c) failure displacement](image2)
V. Conclusion

A successful Rx-FEM implementation in ABAQUS commercial finite element software was demonstrated. The implementation is based on superposition of native ABAQUS elements which are connected by cohesive UEL. The UMAT capability is utilized to facilitate the step function based energy conditions required for crack propagation in the superimposed elements. The analysis begins without any cracks present. At the end of each loading increment LaRC04 failure criterion is examined in each integration point and one or more cracks are inserted in the model at the locations where the criteria is met. The propagation of each crack is then governed by cohesive zone model implemented in a UEL.

Several examples of unnotched and open hole composite unidirectional coupons were considered including the splitting phenomenon of an axially loaded open hole coupon. The method correctly predicts separation and fiber direction stress concentration reduction as a result of splitting.
Acknowledgments

This work was funded under NASA contract number NNL16AA02C with the University of Texas at Arlington, Arlington, TX.

References


