XFEM Modeling of Mixed-Mode Cracks in Thin Aluminum Panels

Lyonel Reinhardt, J.A. Cordes

U. S. Army

Abstract: The Extended Finite Element Method (XFEM) capabilities of Abaqus V6.9-EF1 could have a significant impact on finite element modeling of failure for the U. S. Army. The Army has many areas where fracture is important from failure of components, to penetration, to warhead development. To assess the value of XFEM under static loading, comparisons were made with experimental data of notched panels. The panels had different angles of notches. The crack growth direction and applied forces for crack growth were compared to experiments using ABAQUS. The applied force at failure was within 5% of experimental values. The predicted crack growth direction was accurate. Since much of the Army's work involves dynamic explicit analysis, the XFEM technique may only find limited use until it is extended into the ABAQUS Explicit program.

Keywords: fatigue, failure, fracture, XFEM, 6061-T6 aluminum, crack, notch, mixed-mode

INTRODUCTION

Several center notched panels were tested to determine the applied force at crack growth and the direction of crack growth. In earlier studies, cohesive zone models were used to predict the mixed mode fracture results [1]. The previous analysis used cohesive elements parallel to and perpendicular to crack growth. Cohesive zone properties were derived from experimental KR versus crack growth curves. Critical crack tip opening displacements were incorporated into the cohesive elements. The resulting applied stress predictions were within 8% of experimental results.

This paper describes analysis of the same panels using the XFEM capabilities in ABAQUS V6.9-EF1 [2]. The full Kr verses crack growth curves were not used. Fracture toughness was used to determine a critical energy criteria and a critical displacement criteria. Both results are presented. The current method is simpler to model since cohesive elements are not required and the direction of crack growth does not need to be specified. Experimental parameters are limited to fracture toughness, yield strength, Young's modulus, and Poisson's ratio. Some issues with numerical convergence had to be resolved. Predicted applied loads at fracture were within 5% of experimental results

EXPERIMENT

Test Description

Test were conducted on 0.504 mm thick, 6061-T6 aluminum panels Figure 1. The panel width (2w = 17.8 mm), three crack angles (Φ = 30, 60 and 90 deg) and a/w = 0.33 were tested [3]. Panels were 152-mm in length. The three center-notched panels were created with slots that had rounded ends. The slots were made by a cutting tool with a thickness of 0.178-mm wide. The radius of at the end of the slot was roughly half of the width of the cutting tool.

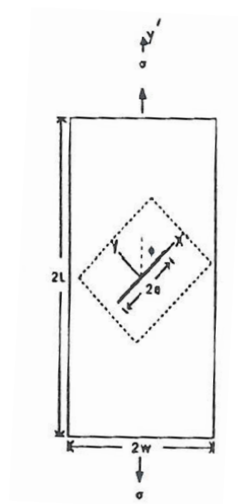


Figure 1.

Experimentally-determined material properties for the panels are given in Table 1. True stress versus true strain points from one tensile test are shown in Table 2, used for analysis. The fracture toughness was determined from three ASTM E-562-87 tests to be 40.1-MPa√m [3,4].

Table 1. Material Properties

Yield Strength, MPa	*269 ± 7*

Ultimate Tensile Strength, MPa	317 ± 7
Young's Modulus, GPa	66.3
Strain at Ultimate Failure, percent	13. ± 0.9
Poisson's ratio, literature	0.33

Table 2. True Plastic Stress versus True Strain

Stress	True Plastic
MPa	Strain
111.0	0.0000
112.6	0.0019
113.6	0.0038
116.1	0.0067
116.9	0.0086
118.7	0.0135
120.7	0.0183
122.5	0.0231
124.6	0.0279
127.5	0.0326
130.0	0.0421
133.3	0.0514
136.1	0.0607
139.3	0.0699
141.8	0.0790
144.4	0.0881
146.3	0.0971

Uni-axial tension stresses in the y' direction were applied to all panels. The experimental results for the three different center-notched angles are shown in Table 3. Photographs of the fractured parts are shown in Figure 2.

Table 3. Applied force at Failure

Crack Angle, Degree	Applied Force, N
90	1695
60	1784
30	2162

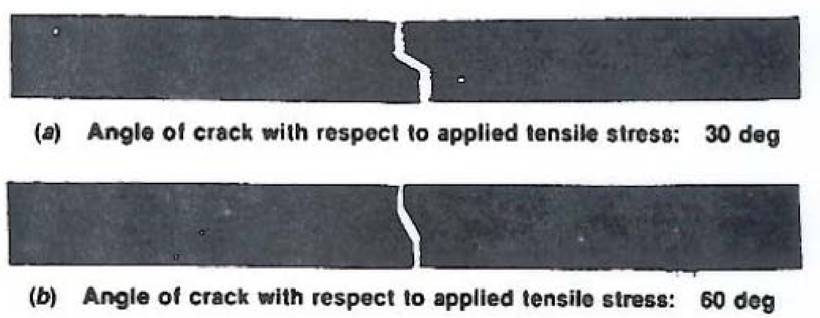


Fig. 2 Photographs of Fractured Panels

METHOD

Modeling and Simulation, Overview

The XFEM capability in Abaqus was used to model tension on the center-notched panels. It was not necessary to pre-model a crack tip singularity or its direction of crack growth. The minimum required is to define a part or area of a part as an enriched zone. This will be the zone where a crack can initiate and grow. The method can determine where a crack will initiate and where it will propagate without any extra modeling effort. An initial crack a geometric line for 2d or a face in 3d can be inserted into the mesh. This can be selected as an initial crack.

Model Geometry

The three cracked panels were modeled by inserting a geometric line making an infinitesimally thin sharp crack. The cracks and center-notches were inserted at angles of 30, 60 and 90 degrees with respect to the applied tensile load. Figure 3 shows the series of panel with angled cracks. The series was repeated for sharp crack line and for the notched tooling. The method required a separate part for the crack or notch. For the crack, a deformable, 2-dimensional wire part was formed and placed at the center of the panel.

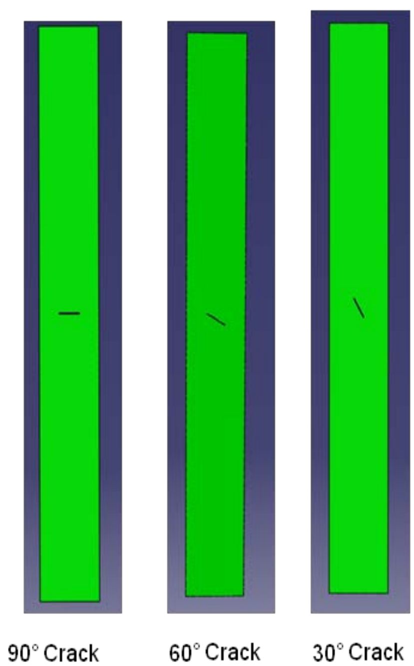


Fig. 3. Crack Angles

Material Model

The panels were modeled with an elastic/plastic material model using the data in Tables 1 and 2. Similar to crack panels with cohesive elements, a damage criteria and damage evolution criteria were specified on the material card in Abaqus. For this study, the maximum principal stress criteria was chosen for the damage criteria ('Maxps Damage' in Abaqus). The bottom stress in Table 2 was used for the maximum principal stress. Damage evolution was based on an energy criteria equivalent to the strain energy release rate, G_I. The damage energy was calculated assuming $G_1 = K_{IC}^2/E$, or 24.2 kN/m. The sub-option for viscosity coefficient was used with a value of 1.E-6.

Analysis Method

Currently, the XFEM capabilities are limited to static analysis. Because of convergence issues [5], a number of changes to the usual step sequence were required.[6] The minimum time step was set to 1.E-10-second and the number of increments increased. The initial time step was set to 0.02-second. In addition, the following steps were required in the Abaqus Step Module:

1. Step Module => Other => General Solution Controls => Manager => Edit Step => Time Increment => Discontinuous Analysis => more => IA =20
2. Step Module => Step => Step Manager => Edit => Automatic Stabilization => Specify dissipated energy fraction => accept default
3. Step Module => Other => General Solution Controls => Manager => Edit Step => => Time Increment => select Discontinuous Analysis

Specify XFEM in ABAQUS

XFEM is specified in the ABAQUS Interaction module:

1. Interaction Module => Special => Manager
2. Create => XFEM => Select the panel for the Crack Domain => Select the wire-crack line for the crack

Boundary Conditions, Loads, and Constraints

As shown in Figure 4 the base was held fixed and displacement control was used to displace the top of the panel, putting the panel in uni-axial tension. Displacement was applied in increments.

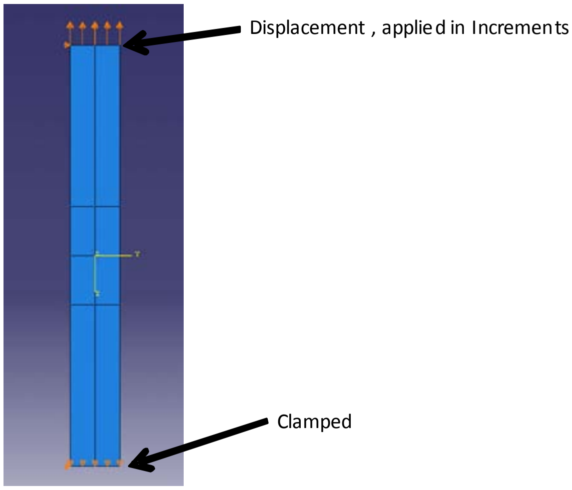


Fig. 4 Boundary Conditions and Load

Mesh Details

Figure 5 shows the mesh around at the center-notch where the mesh is refined around the ends. Figure 6 shows that the mesh does not conform to crack created by the inserted geometry (line for 2-d and face for 3d). Two dimensional plane stress elements were specified (CPS4R in Abaqus). The two ends of the crack abutted the middle of a finite element edge; the crack tips were not coincident with a finite element node.

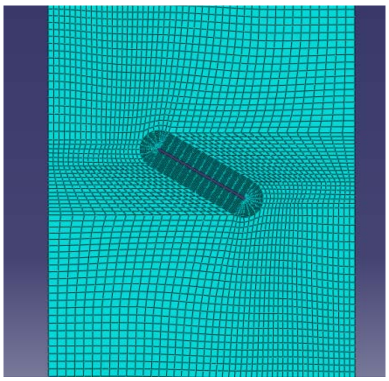


Fig. 5 60 Degree Center-notched

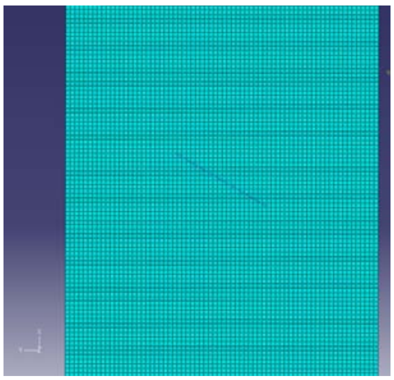


Fig. 6 60 Degree Crack

SIMULATION RESULTS

Crack Growth Abaqus XFEM

The simulated crack growth for the pre-cracked panels is shown in Figures 7 to 9. The simulated growth for the center-notched models are shown in Figure 10 to 12. In all cases crack initiation, growth and direction was reasonable. There were some problems with the crack separation in the simulation. In some areas the crack did not separate completely due to the face that there is still some cohesive strength between the cracked element.

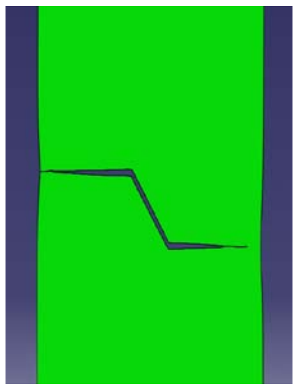


Fig.7 30° Crack Growth

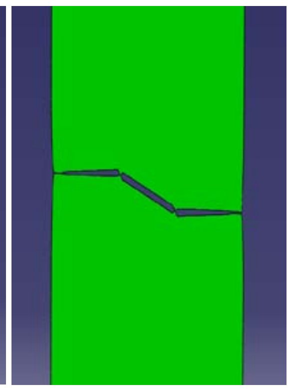


Fig.8 60° Crack Growth

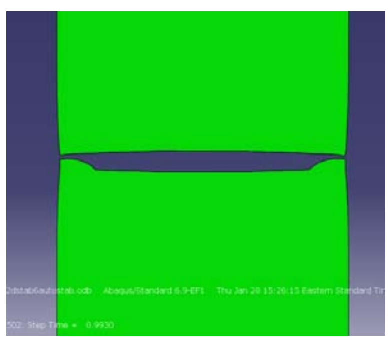
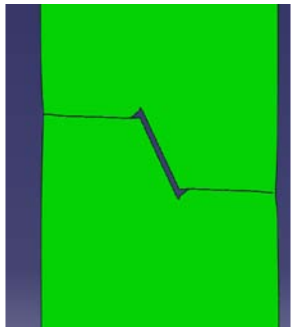


Fig. 9 90° Crack Growth



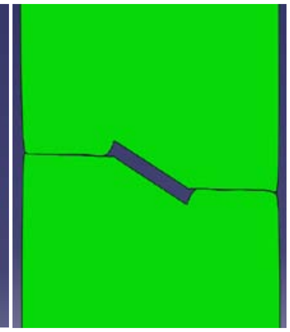


Fig.10 30° Center-notched   Fig.11 60° Center-notched

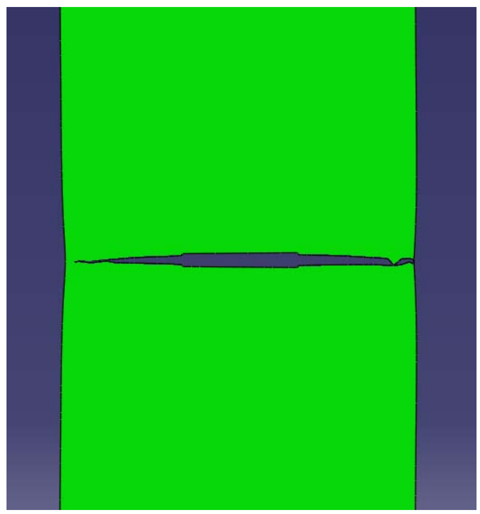


Fig.12 90° Center-notched Crack Growth

Plastic Zone of Cracked and Notched Bars at Fracture Initiation

Figures 13-14 show the plastic zone for the 60° crack and notch prior to crack intitiation. The von Mises criteria was used for the Abaqus equivalent plastic strain 'peeq'. Edge effects were evident in the angled pre-cracked panels.

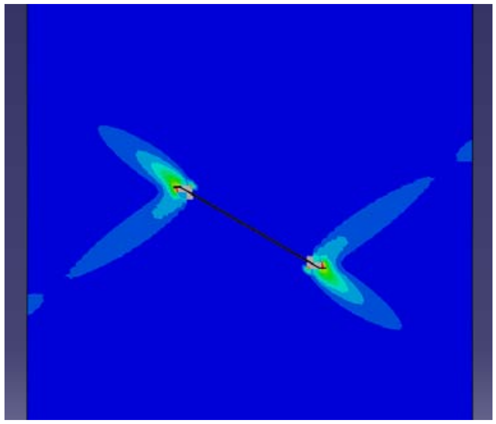


Fig 13 60° Crack

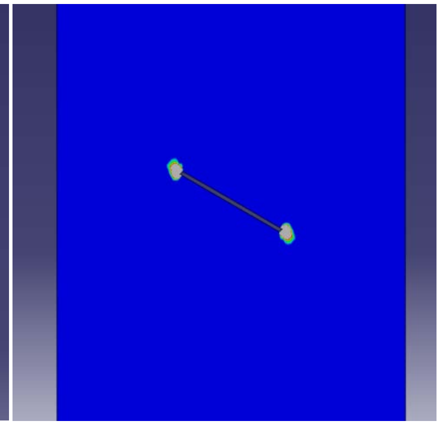


Fig 14 60° Crack

Comparison Between Center-notched and Crack Plates Abaqus XFEM

Figure 15 compares applied force versus applied displacement for the six simulations. For the 90-degree crack, a slightly higher force was required for failure of the center-notched panel compared to the crack panel. This is expected since the notched panels do not have the crack tip singularity. In this study, the force required for failure on the 60 and 30 degree notched panels was lower than for the cracked panels.

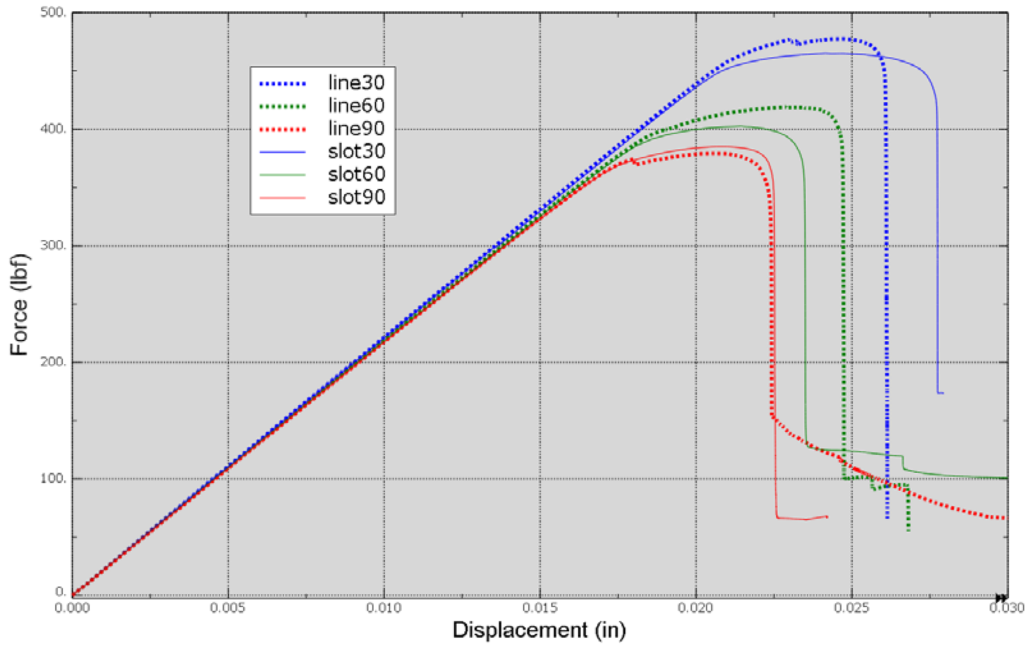


Fig.15 Force Displacement Plot at 30°, 60° and 90° Crack and Notch Angles

Comparison between experiment and analysis

The applied loads at crack growth are summarized in Table 4. The predictions for the notched panels were within 4.5% of experimental values.

Table 4. Results Summary

Crack Angle	Experimental Force, N	Simulated Force with a Crack, N	% Error	Simulated Force with a Center-notched, N	% Error
30°	2162	2122	-1.9	2068	-4.3
60°	1784	1864	4.5	1788	0.3
90°	1695	1686	-0.5	1708	0.8

DISCUSSION

The XFEM method has shown itself to be an accurate method to predict crack initiation and growth. In all the cases the maximum force required to generate the cracks was close to the experimental values. The locations and direction of the cracks were also good matches to experiment. That said there are still weaknesses in the method.

It was difficult to get converged answers. The Abaqus reference was very helpful [5]. The steps described were required to get the analysis to converge. There are also issues with the location of the tips of the line crack with the mesh. Since the XFEM method can only fail a whole element at a time though Abaqus somewhat accounts for the partially cracked case through the non-uniform cohesive strength on the cracked element, the tip of the line crack must be on the edge of an element. But the cracks do not run well along the edge of an element so it's best if the mesh is designed so the crack will run through the middle of element or you can perform an analysis without an initial crack. The newly generated crack during the analysis will not likely be coincident with the element edges. There is still a lot of work to do to determine where XFEM has the most utility.

There are numerous parametric studies that could be conducted:

1. Compare yield strength to the ultimate tensile strength for the damage initiation criteria. The size and shape of the damage zone could be compared to theoretical values.
2. Compare use of energy criteria to displacement criteria for damage evolution.
3. Several parameters are available for mixed mode behavior. The values in reference [1] might be inserted into the XFEM model. The use of the BK criteria could be investigated.
4. Results with 2D and 3D elements could be compared.
5. Sensitivity of crack tip locations with respect to the mesh could be investigated.

CONCLUSIONS

The XFEM capability in ABAQUS V6.9-EF1 provided accurate estimates of the failure condition for several center-notched and cracked panels. The next step will to be model 3-dimensional fracture. Additional studies, perhaps with composite materials, are also planned. The U.S. Army looks forward to a version of XFEM in ABAQUS Explicit.

REFERENCES

[1] J. Cordes and R. Yazici, *Mixed Mode Fracture in Plastically Deforming Materials*, Journal of Pressure Vessel Technology, ASME, V 115, N 4, pp. 348-352, 1993.

[2] "ABAQUS User Manual V6.9-EF1", Dassault Systems 2004-2010.

[3] Cordes, J., and Yazici, R., "A Fracture-Critical Failure Criteria for Composite Materials." Phase II Final Report, Contract N00164-87-C-0162, Castle Point Research Technologies, Hoboken, NJ.

[4]" ASTM E-562-86, Standard Practice for R-Curve Determination," Annual Book of ASTM Standards, 1986.

[5] "Improving XFEM Convergence", Dassault Systems, 2004-2010.

[6] "Defining an XFEM Analysis in Abaqus/CAE, 2004-2010.