DEVELOPMENT OF A FATIGUE CRACK GROWTH COUPON FOR HIGHLY PLASTIC STRESS CONDITIONS

Phillip A. Allen\*, Pravin K. Aggarwal\*, and Gregory R. Swanson\*
NASA Marshall Space Flight Center
Huntsville, AL
phillip.a.allen@nasa.gov

ABSTRACT

The analytical approach used to develop a novel fatigue crack growth coupon for highly plastic stress field condition is presented in this paper. The flight hardware investigated is a large separation bolt that has a deep notch, which produces a large plastic zone at the notch root when highly loaded. Four test specimen configurations are analyzed in an attempt to match the elastic-plastic stress field and crack constraint conditions present in the separation bolt. Elastic-plastic finite element analysis is used to compare the stress fields and critical fracture parameters. Of the four test specimens analyzed, the modified double-edge notch tension - 3 (MDENT-3) most closely approximates the stress field, J values, and crack constraint conditions found in the flight hardware. The MDENT-3 is also most insensitive to load misalignment and/or load redistribution during crack growth.

INTRODUCTION

This paper presents the analytical approach used to develop a novel fatigue crack growth coupon for a highly plastic three-dimensional (3-D) stress field condition. The flight hardware investigated in this paper is a large separation bolt that fractures using pyrotechnics at the appointed time during the flight sequence. The separation bolt has a deep notch that produces a severe stress concentration and a large plastic zone when highly loaded. For this geometry, linear-elastic fracture mechanics (LEFM) techniques are not valid due to the large nonlinear stress field at the notch root. Unfortunately, industry codes that are commonly available for fracture mechanics analysis and fatigue crack growth (e.g. NASGRO\*) are generally limited to LEFM and are available for only a limited number of geometries. The results of LEFM based codes are questionable when used on geometries with significant plasticity. Therefore elastic-plastic fracture mechanics (EPFM) techniques using the finite element method were used to analyze the separation bolt.

A testing program was developed to verify the EPFM separation bolt analysis results. Testing of full-scale flight hardware is very costly in terms of assets, laboratory resources, and schedule. Therefore to alleviate some of these problems, a series of novel test specimens were developed to simulate the elastic-plastic stress field present in the bolt. Finite element models were created to predict the stress fields in the test specimens and the flight hardware. In addition, finite element models of cracked test specimens and flight hardware were created to compare critical fracture parameters along the crack front. A summary of the test specimen geometry development is given below.

SPECIMEN GEOMETRY DEVELOPMENT

Notched Plate Specimen

The first test coupon developed was the notched plate (NP) specimen as shown in Figure 1-(a). The gage section of the NP matches the cross sectional dimensions of the notch region of the separation bolt. The specimen is pin loaded on a narrow land (0.20 in. wide) in the center of the pin holes to help ensure load alignment. Unfortunately this specimen design is very sensitive to load placement, and, therefore, a small change in load alignment results in widely varying stress fields in the notch region.

Modified Double-Edge Notch Tension Specimens

To reduce the alignment sensitivity of the specimen, a modified double-edge notch tension (MDENT) specimen was developed (Figure 1-(b)). This specimen has the advantage of being symmetric, which reduces bending sensitivity, while still preserving the free surface on the back face of the notch section. This specimen was an improvement over the NP specimen, but was still relatively sensitive to load alignment due to the stiffness of the transition section below the pin hole.

To reduce the alignment sensitivity of the specimen even more, the distance between the two notch sections was increased and fillets were cut into the transition section below the pin holes. Using two different transition fillet radii resulted in the MDENT-2 (Figure 1-(c)) and MDENT-3 (Figure 1-(d)) specimens. The moment of inertia, I, for the fillet section on MDENT-2 is approximately 53% that of the notched section, and I for the fillet section on MDENT-3 is approximately 8% of that of the notched section. For misaligned loads, the thin fillet section acts as a flexure and therefore helps preserve the desired stress state in the notch sections.

---

\* Structural Analyst, Strength Analysis Group/ED22, NASA/MSFC
\* AIAA Member, Deputy Group Lead - Strength Analysis Group/ED22, NASA/MSFC
\* Structural Analyst, Strength Analysis Group/ED22, NASA/MSFC
MATERIAL PROPERTIES
The separation bolt housing is made of 4340 steel. Material properties for 4340 were obtained from uniaxial tensile tests performed at Marshall Space Flight Center (MSFC). A summary of the general material properties for 4340 steel is shown in Table 1. Elastic-plastic finite element analysis using the commercial finite element code ABAQUS requires a table of true stress versus plastic strain, $\sigma$ vs $\epsilon$. Therefore, $\sigma$ and $\epsilon$ values were calculated from the uniaxial tensile data and used as inputs for the von Mises plasticity model in ABAQUS. Data points from the actual test record were used for the initial portion of the table values. After the maximum load, the test record no longer represents a uniaxial stress-strain response due to necking of the test specimen. Therefore, an additional point at $\epsilon_f = 1.0$ and $\sigma = 210$ kip was added to the end of the data to complete the table and to ensure stability in the plasticity solution algorithm. A plot of the ABAQUS plasticity inputs compared to the uniaxial tensile tests performed at Marshall Space Material properties for 4340 were obtained from a table of true stress versus plastic strain, $\sigma$ vs $\epsilon$ values were calculated from the uniaxial tensile data and used as inputs for the von Mises plasticity model in ABAQUS. Data points from the actual test record were used for the initial portion of the table values. After the maximum load, the test record no longer represents a uniaxial stress-strain response due to necking of the test specimen. Therefore, an additional point at $\epsilon_f = 1.0$ and $\sigma = 210$ kip was added to the end of the data to complete the table and to ensure stability in the plasticity solution algorithm. A plot of the ABAQUS plasticity inputs compared to the uniaxial tensile tests performed at Marshall Space.

FINITE ELEMENT MODELING

Notched Plate Finite Element Model
A 3-D FEM of the notched plate specimen was created using 6,384 20-node reduced integration bricks and 29,783 nodes as shown in Figure 6. Two symmetry planes were utilized in creating the FEM by dividing the geometry along the longitudinal axis and along the center of the notch root. A concentrated force was applied to a node or nodes along the top of the “pin hole” to simulate centered or off-centered loading.

MDENT Finite Element Models
Three-dimensional FEM’s of the MDENT, MDENT-2, and MDENT-3 geometries were created using 20-node reduced integration bricks. Two symmetry planes were utilized in creating the FEM’s by dividing the geometry along the longitudinal axis and along the center of the notch root. A concentrated force was applied to a node or nodes along the top of the “pin hole” to simulate centered or off-centered loading. In addition, a 3-D FEM of the MDENT-3 with a crack located in one notch root was created using the multi-step processed described for the 3-D cracked separation bolt model. The 3-D cracked FEM consisted of 17,032 20-node reduced integration bricks and 79,129 nodes, and the FEM is illustrated in Figure 7.

FINITE ELEMENT RESULTS

Stress Field Comparisons
Axial stress across the notch section of the separation bolt at proof load is compared to axial stress across the notch section for the NP at an equivalent load in Figure 8. The equivalent load was based on a cross sectional area ratio, and the NP stress values are taken along the symmetry plane at the center of the NP specimen geometry. Notched plate stress values are plotted for the case of the load centered on the NP land and for two cases where the load is offset ±0.0521 in. on the land.
The offset distance of 0.0521 in. represents an offset of approximately one-quarter land width and was chosen as a matter of convenience due to FEM node spacing. The centered NP load case slightly overestimates the axial stress in the vicinity of the notch but predicts a significant compressive stress on the backface of the section. The case with the load offset toward the notch over-predicts the axial stress for approximately 70% of the notch section and then produces a very large compressive stress on the backface, indicating a significant bending load on the notch section. The case with the load offset toward the backface slightly underestimates the separation bolt stress profile, but does not predict a compressive stress on the backface.

The axial stress across the separation bolt notch section at proof load is compared to the axial stress across the notch section in the three MDENT specimen geometries at an equivalent load in Figure 9. The equivalent load was based on a cross sectional area ratio, and the MDENT stress values are taken along the symmetry plane at the center of each MDENT specimen geometry. For each MDENT specimen the load application point is offset 0.0512 in. towards the "right" (as labeled in Figure 7) to simulate approximately one-quarter land width off-center loading. Stresses are then plotted for both the "left" and "right" notch sections. For all of the MDENT offset cases, the right notch stresses overestimate the bolt axial stresses and the left notch stresses underestimate the bolt stresses. The difference between the left and right notch stress predictions decreases significantly for the MDENT-3 specimen, verifying the effectiveness of the flexure section in reducing misalignment induced bending effects. The MDENT-3 specimen with centered load predicts stress distributions that essentially match the separation bolt stresses in the vicinity of the notch and that reasonably match the majority of the bolt stress distribution. All of the MDENT geometries predict varying amounts of compressive stress on the backface.

The effective (von Mises), axial, radial, and hoop stress profiles across the separation bolt notch section at proof load are compared to the corresponding stress values across the notch section of the MDENT-3 specimen at an equivalent load in Figure 10. For this case, the equivalent load for the MDENT-3 was scaled to approximately match the plastic zone size in the notch region. Therefore, the effective stress predictions and corresponding plastic zone sizes are very similar for the separation bolt and the MDENT-3. The MDENT-3 underpredicts the peak axial stress in the bolt by approximately 3% and predicts a compressive axial stress on the backface. The radial stress predictions for the bolt and MDENT-3 are almost identical. The MDENT-3 significantly underestimates the hoop stress in the bolt due to the lack of hoop constraint inherent in the MDENT-3 geometry.

**CONCLUSIONS**

In summary, four test specimen configurations were analyzed in an attempt to match the elastic-plastic stress field and crack constraint conditions present in the separation bolt. Of the four test specimens analyzed, the MDENT-3 most closely approximates the stress field, J values, and crack constraint conditions found in the flight hardware. The MDENT-3 is also most insensitive to load misalignment and/or load redistribution during crack growth.

**Table 1. General Material Properties for 4340 Steel**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elastic Modulus, E (psi)</td>
<td>$30 \times 10^6$</td>
</tr>
<tr>
<td>Poisson’s Ratio, $\nu$</td>
<td>0.29</td>
</tr>
<tr>
<td>Tensile Yield Strength, $F_y$ (psi)</td>
<td>$180 \times 10^3$</td>
</tr>
<tr>
<td>Tensile Ultimate Strength, $F_u$ (psi)</td>
<td>$198 \times 10^3$</td>
</tr>
</tbody>
</table>
Figure 1. Illustration of: (a) Notched Plate, (b) MDENT, (c) MDENT-2, (d) MDENT-3

Figure 2. ABAQUS FEM Plasticity Inputs Compared to Uniaxial Tensile Test Data

Figure 3. Axisymmetric FEM of Separation Bolt
Figure 4. 3-D FEM of Separation Bolt with Crack in Notch Root

Figure 5. 20-Node Brick Element with a Collapsed Face to Create Singular Fields

Figure 6. 3-D FEM of Notched Plate Specimen

Figure 7. 3-D FEM of MDENT-3 with Crack in Notch Root
Figure 8. Axial Stress Across the Notch Section of the Separation Bolt and the NP Geometry

Figure 9. Axial Stress Across the Notch Section of the Separation Bolt and Three MDENT Geometries

Figure 10. Comparison of Stress Components Across the Notch Section for the Separation Bolt and the MDENT-3 Specimen

Figure 11. Comparison of $J$ Along the Crack Front for the Separation Bolt and the MDENT-3 Specimen
Figure 12. Comparison of $\beta$ Along the Crack Front for the Separation Bolt and the MDENT-3 Specimen

REFERENCES


