

Analysis of Fatigue Life Estimate for the M119 Cradle Assembly with a Gouge Cut Defect

C. M. Weaver, R. Terhune, B. Peterson

U.S. Army/ARDEC, Picatinny Arsenal, NJ 07806-5000

Abstract: The M119 howitzer is a lightweight towed 105mm field artillery weapon system. It is an indirect-fire (gunfire delivered to a target that is not in a direct line of sight) weapon that primarily fires projectiles along a curved trajectory. It is towed by the HMMWV (hum-vee), however, it can also be airlifted and dropped by parachute to its required location. The firing mechanism is mounted to a steel channel that is connected to the rails of the firing cradle by brackets. The firing mechanism plate is a support plate behind the attachment brackets. The plate was evaluated for critical flaw size and fatigue life. Critical flaw size was estimated using the Extended Finite Element Method (XFEM) capabilities in Abaqus. Fatigue analyses were completed using Linear Elastic Fracture Mechanics assumptions. Both efforts used strain gauge data collected during sampling firing test conducted by the U.S. Army. This paper will discuss the impact of a manufactured tooling groove defect in the firing plate on the life cycle of the cradle. Results predicted that the cradle with the gouge cut in the firing plate would have a design life greater than 1000 cycles. Results are consistent with the 200+ successful shots that have been completed on fielded M119 howitzers with this gouge.

Keywords: 95-15 steel, crack, fatigue, failure, fe-safe, implicit, NASGRO, XFEM

1. Introduction

The M119 howitzer is a lightweight towed 105mm field artillery weapon system. The M119A2 is the current manufactured version of this weapon system. During the manufacturing process in 2011, 46 M119A2 systems were manufactured with a tooling groove defect in the 125934242 Cradle. The worst case tooling groove was 0.071-in deep and 2.300-in long, spanning the full length of the channel. A cradle with this defect is shown in Figure 1.

These tooling grooves were located in the firing mechanism plate region of the cradle. The firing mechanism is mounted to a steel channel that is connected to the rails of the firing cradle by brackets. The firing mechanism plate is a support plate behind the attachment brackets.

The structure of this plate is critical to the survivability of this weapon system because it is impacted by the force of the gun loads during firing. As a result, any change to the structural integrity of this plate could impact the intended lifecycle for this weapon system (Terhune 2011).



Figure 1. Rear Cradle Structure with Groove Highlighted in Red.

This paper describes the analysis of the cradle of the M119 howitzer with a manufactured tooling defect. The purpose of this analysis was to determine if the cradle would survive the minimum acceptable firing requirement of 1100 rounds. Results were analyzed using the XFEM capabilities in Abaqus and fatigue capabilities in fe-safe and NASGRO, (Simulia 2004-2011, Safe Technology, NASA).

2. Experiment

2.1 Test Description

The strain data used in these simulations was obtained from three strain gauges used in 50 test shots that were performed and documented from May 23 to 27, 2011 at Yuma Proving Ground (YPG), Yuma, AZ. The test shots were performed using a M119A2 howitzer that had passed all manufacturing inspection criteria. The 50 shots were conducted at various quadrant elevations (QEs) and propellant charge zones that are seen in field to get the maximum or worst case strain data. Strain data was recorded for 1 second in 0.0005-s increments. The purpose of these tests were to obtain strain data to be used in these simulations. The strain data was recorded using the CEA-06-125UW-350 uniaxial strain gauge designed by the Micro-Measurements division of the Vishay Precision Group (Micro-Measurements, 2010).

The maximum strain data recorded was for the test shots fired at a QE of 300 mils and a charge zone of 8. The average of the strain data from the 10 test shots for this case was calculated and multiplied by a factor of safety of 1.25 for each of the three sensors. The stress was calculated using the equation

$$\sigma = \varepsilon \cdot E, (1)$$

where σ is stress, ε is strain and E is Young's modulus. This calculated pressure data was used in the load module of Abaqus.

3. Method

3.1 Modeling and Simulation, Overview

The tooling defect was modeled as a plate specimen with a gouge using the dimensions of the area of the channel affected by the tooling defect with the worst case size defect. This plate was modeled using Abaqus 6.11 Explicit and Implicit. Explicit modeling and simulation was used to determine plasticity, stress and displacement response. These simulations were run as flat files so they could be later used in fe-safe simulations. The implicit analysis was done using the XFEM capability in Abaqus.

The XFEM capability in Abaqus was used to model the structural effects of the recorded strain data converted into pressure data on a plate specimen with a gouge to represent the channel with the tooling defect. The data used was an average of the maximum recorded strain values from the test case at a QE of 300 mils and a charge zone of 8 (Powers, 2011).

The explicit analysis and the initial implicit analysis contained 90,534 nodes and 78,702 elements. The explicit analysis was compiled on a UNIX cluster using 48 cpus and took approximately 72 hours to complete. The initial implicit analysis was compiled on a UNIX cluster using 12 cpus and took approximately 16 hours to complete. An additional implicit analysis was done containing 186,345 nodes and 166,530 elements – also using 12 cpus – and took approximately 37 hours to complete. Analyses were non-linear.

The initial analysis was done using a plate specimen containing 90,534 nodes and 78,702 elements. Based on the crack propagation observed in this analysis, a second analysis was done using a plate specimen containing 186,345 nodes and 166,530 elements. Both analyses showed similar crack propagation.

Separate precracked simulations were done using a crack that was 0.015-in in length horizontally and vertically along the gouge cut. This dimension was based off a florescent penetrant test that was conducted and documented on June 11, 2011 at YPG. In this test, 212 rounds were fired using an M119A2 with the tooling defect. After testing the channel was inspected for cracks. Florescent penetrant testing showed the presence of a 0.015-in crack in the tooling defect area (Le, 2011).

A fatigue life cycle analysis was performed using fe-safe v6.2. A dynamic/explicit flat file from the Abaqus analysis was imported into fe-safe. The peak applied pressure located at time step 0.2s was used for the applied load. This load case was repeated 5 times to simulate reverberation from gun launch.

A crack propagation and fracture mechanics analysis was also done using NASGRO v5.0. A surface crack plate specimen was chosen from the NASGRO geometry library. The same part geometry from the plate and pre-cracked simulation cases used in Abaqus were used in NASGRO.

3.2 Model Geometry

The geometry of the plate specimen was modeled in Abaqus/CAE. All models used in the analyses were meshed with 8-node hexahedral elements (C3D8R). Based off of the STATUSXFEM results seen in the first analysis, the analysis was rerun using more than double the amount of elements for better accuracy. Precracked simulations were also done based off the results of a fluorescent penetrant test.

A dynamic, explicit simulation was performed to determine plasticity, stress and displacement response. Results were used in fe-safe to determine the fatigue lifecycle of the specimen.

For the NASGRO analysis a surface crack specimen (SC17) was used from the NASGRO geometry library. Figure 2 shows this geometry case. The geometry parameters for this case is shown in Table 1.

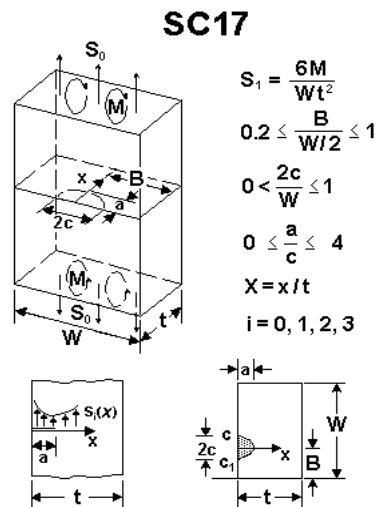


Figure 2. Surface Crack Specimen SC17 (NASGRO).

Table 1. NASGRO Analysis Parameters.

Parameter	Value
Thickness (t), in	0.08
Width (W), in	2.35
Crack center offset (B), in	1.175
Initial flaw size	1238.1
Fracture Toughness ksi-in ^{1/2}	202
Viscosity coefficient, psi-s	1.45E-10

3.3 Material Model

All parts were modeled with elastic/plastic material models. The material data for these simulations were obtained from elastic-plastic fracture toughness and tensile testing performed by Michael Hespos, an engineer specializing in materials testing at Picatinny Arsenal, NJ. The yield stress and plastic strain values were taken from the tensile test performed on specimen #4. Twenty points between yield stress and failure were used for the plasticity data in the material properties (Hespos 2011). The material properties used for the Abaqus and NASGRO analyses are shown in Table 2.

Table 2. Material Properties – 95-15 Stainless Steel.

Material Property	Value
Density, lbf-s ² /in ⁴	7.3298E-04
Young's Modulus, psi	3.031E07
Poisson's Ratio	0.270
Viscosity coefficient, psi-s	1.45E-10
Maximum Principal Stress, psi	1.565E5

The XFEM analysis requires additional material information in the enriched area of the crack. The maximum principal stress criteria ('Maxps Damage' in Abaqus) was selected for the damage initiation criteria. The measured peak stress was used as the limiting maximum principal stress. Fracture energy was used for the damage evolution criteria. The value for fracture energy was taken from specimen #2 in the elastic-plastic fracture toughness analysis (Hespos 2012).

Fe-safe did not contain a property model for 95-15 stainless steel in its material database. As a result, 4140 steel (SAE 4140) was chosen for this analysis because the ultimate tensile strength (156,060 psi) was very similar to the ultimate tensile strength for 95-15 stainless steel.

For the NASGRO analysis a user material file was created using the values for stress, fracture energy, and the Paris coefficient crack parameters (C and n) from the test data provided by M. Hespos. Additional crack growth parameters were taken from the material file for 15-5PH H1025 stainless steel because it has similar elastic-plastic mechanical properties as 95-15.

3.4 Analysis Method

All XFEM simulations were performed using dynamic, implicit analysis. The Abaqus advice under 'XFEM' was very helpful for convergence issues (Abaqus answers 4396, 4397). Based on previous work and the Abaqus answers, the minimum time step was set to 1.E-10 s and the number of increments was increased to 100,000 (Reinhardt, L., 2010). The initial time step was set to 1s, which was the time increment of the recorded data from the strain gauge testing.

The following changes were also made to in the Step Module (Reinhardt, L., 2010):

- Step Module → Other → General Solutions Control → Manager → Step-1 → Edit → Time Increment (tab on top) → Specify: → Discontinuous analysis (check box) → More (second tab down on the left) → IA = 20 → Okay
- Step Module → Step → Manager → Step-1 → Edit → Basic (tab on top) → Application: → Moderate dissipation (pull down tab) → Okay

The fe-safe simulation was performed by importing a dynamic, explicit analysis that had been run as a flat file. The material model SAE 4140 was specified in the material section. For the load settings, the peak applied pressure located at time step 0.2s was used for the applied load. This load case was repeated 5 times to simulate reverberation from gun launch.

The NASGRO simulation was performed using a surface crack plate specimen (SC17). The normalized stress applied in the analysis was calculated using 11 node points located at the thinnest part of the plate specimen from the time step with the maximum von Mises stress of 154,808 psi from the dynamic, explicit analysis. The stress scale factor, S0 was set to this maximum von Mises stress. The analysis was scheduled to run for 1000 flights.

3.5 Specifying XFEM in Abaqus

XFEM is specified in the Interaction Module in Abaqus for a model that does not contain a precrack as follows (Reinhardt, L., 2010):

- Interaction Module → Special → Crack → Manager → Create → XFEM → Continue → Select sections or part for crack location → Allow crack growth (check box) → Specify contact property (check box) → Select contact property (pull down tab) → Okay

For a model containing a precrack the following steps needed to be done:

- Part Module → Create a Planar Shell of the correct crack length and twice the depth. Placement was easier. Figure 4 shows one of several cracks used.
- Assembly Module → Translate the crack instance so that the mid surface node of the crack corresponds to an accessible node in the part. Then move the crack-instance slightly so that crack doesn't correspond to an element edge.
- Interaction Module → Special → Crack → Manager → Create → XFEM → Continue → Select sections or part for crack location → Allow crack growth (check box) → Crack location (check box) → Select Planar Shell part that represents the crack → Specify contact property (check box) → Select contact property (pull down tab) → Okay
- Mesh Module – several elements along the crack are desirable. An aspect ratio near 1 for the element dimensions in the enriched area is preferred.

In order to visualize the crack in the output database the following changes need to be made to the Step Module:

- Step Module → Output → Field Output Requests → Manager → F-Output-1 → Edit → Fracture/Failure → PHILSM → State/Field/User/Time → STATUSXFEM → Okay

3.6 Boundary Conditions, Loads, and Constraints

The model for this analysis was constructed as a plate specimen with a gouge using the dimensions of the area of the channel affected by the tooling defect. To simulate the fixed position of the channel on the cradle, the bottom face of the plate in the x-z plane was constrained in all directions and rotations using an Encastre boundary condition and the top face in the x-z plane was constrained directionally and rotationally in the z-direction.

To apply the load, the top face in the x-z plane was partitioned evenly into three equal parts. The pressure data calculated from the strain gauge testing was applied to the corresponding left, middle and right part of the top face of the x-z plane. This load setup is shown in Figure 2.

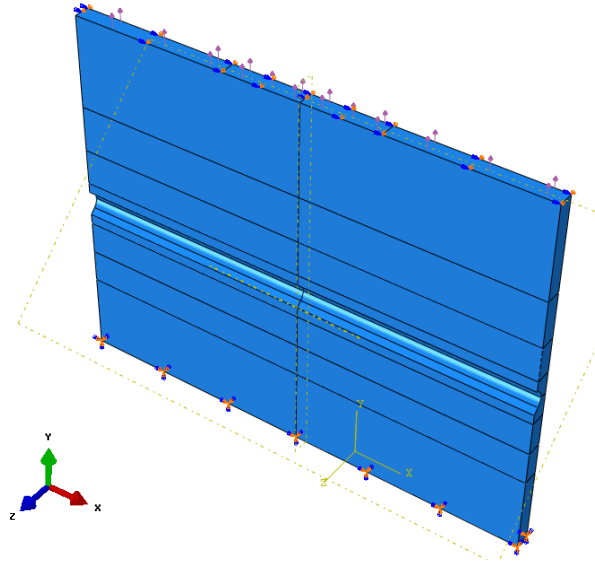


Figure 2. Boundary Conditions and Loads.

For the fe-safe analysis, the peak applied pressure located at time step 0.2s was used for the applied load. This load case was repeated 5 times to simulate reverberation from gun launch.

For the NASGRO analysis, the normalized stress applied was calculated using the maximum Von Mises stress output across 11 nodes in the thinnest section of the plate specimen. The normalized stress applied in this analysis is listed in Table 4.

Table 4. Normalized Data (NASGRO).

Normalized X	Normalized S0	Stress from Abaqus ODB (psi)
0	1	154808
0.1	0.989083252	153118
0.2	0.967068885	149710
0.3	0.823122836	127426
0.4	0.577328045	89375
0.5	0.380914423	58968.6
0.6	0.221089349	34226.4
0.7	0.105762622	16372.9
0.8	0.142635394	22081.1
0.9	0.322923234	49991.1
1	0.427379722	66161.8

3.7 Mesh Details

All elements for parts in this analysis were modeled using C3D8R elements. In order to improve the aspect ratio of the elements, a finer mesh was used for the second analysis of the plate specimen. A finer mesh was used in the gouge area since it was the area of interest for a crack initiation site. Figures 3 and 4 show the mesh details for the models analyzed.

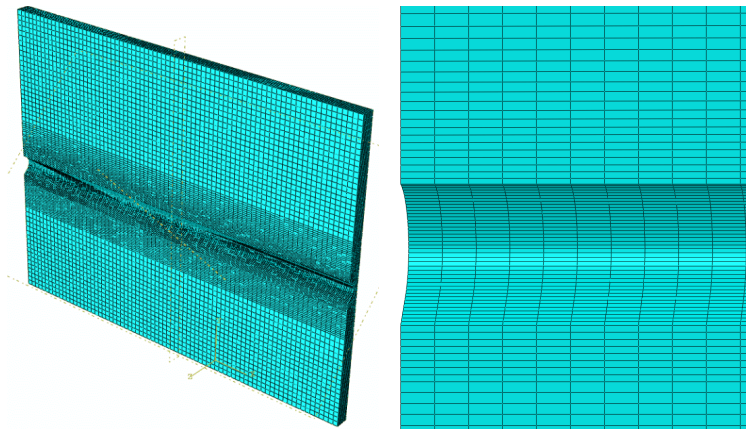


Figure 3. Coarse Mesh.

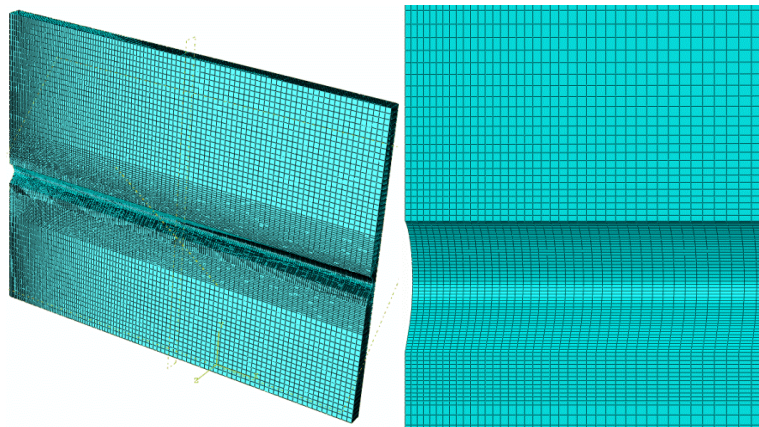


Figure 4. Fine Mesh.

4. Results

4.1 Results Course Mesh, Crack Initiation and Growth (XFEM)

Using the pressure data calculated from the strain gauge testing, simulated partial crack growth was observed. The crack initiation site was on the left side of the plate which is the area that experienced the highest strain values during the strain gauge testing. The crack grows along the x-

direction and varies between one and two elements through the thickness of the y-z plane. This crack growth is show in Figures 5 and 6.

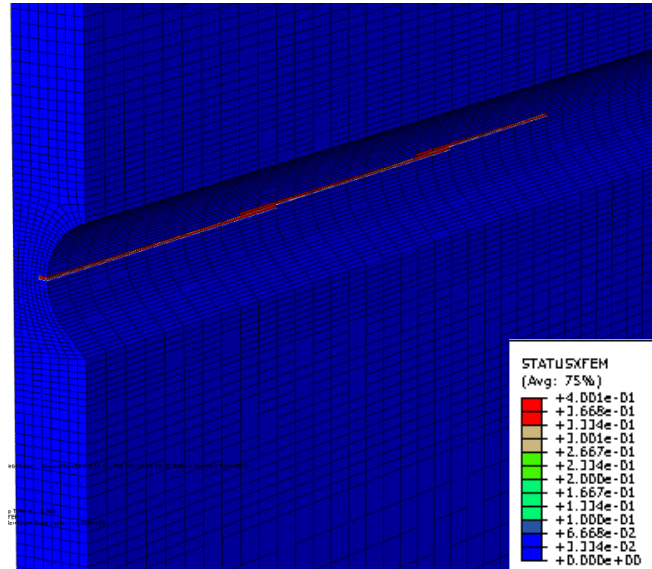


Figure 5. XFEM Crack – Course Mesh.

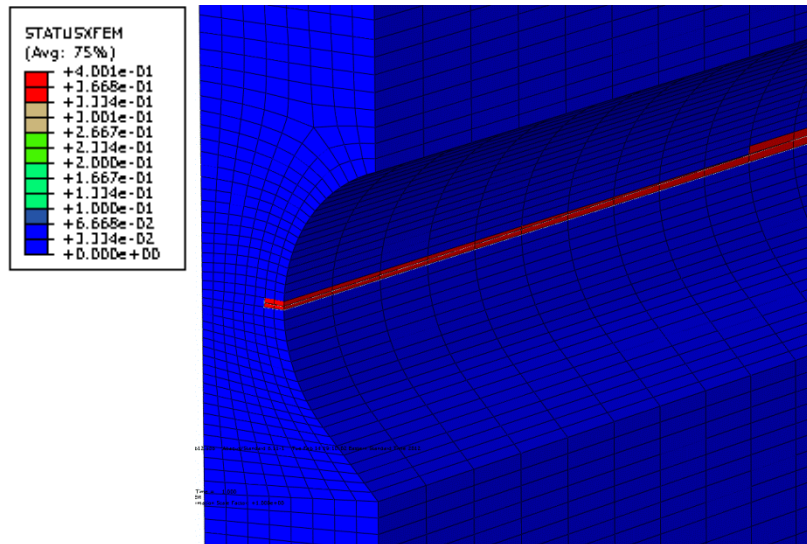


Figure 6. XFEM Crack – Course Mesh (Zoomed In View).

4.2 Results Fine Mesh, Crack Initiation and Growth (XFEM)

A finer mesh was used in order to improve the aspect ratio of the elements. The same crack initiation and propagation from the course mesh plate was observed with the finer meshed plate. This crack growth is show in Figures 7 and 8.

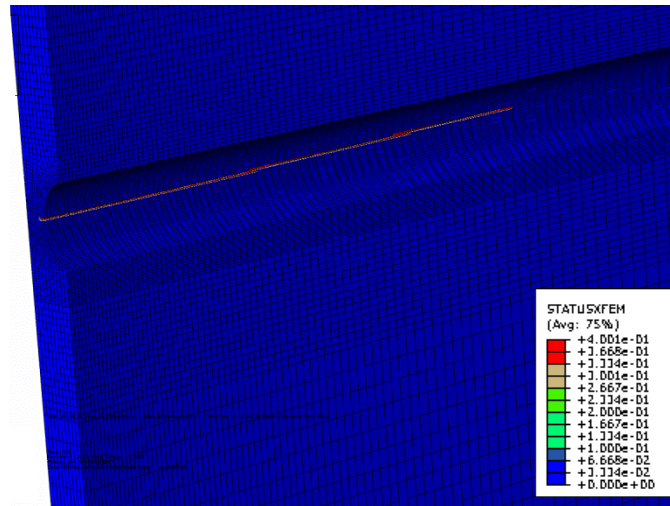


Figure 7. XFEM Crack – Course Mesh.

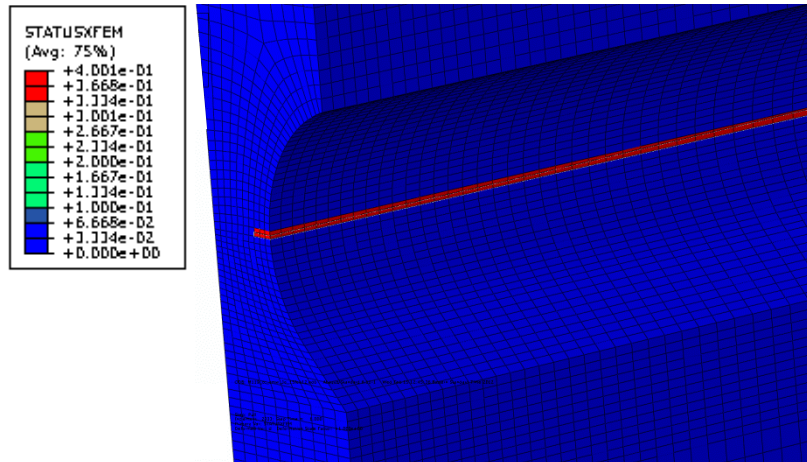


Figure 8. XFEM Crack – Course Mesh (Zoomed In View).

4.3 Precracked Simulations

All precracks have a length of 0.015-in based off of the observed crack seen in the fluorescent pentrant test. Results for the 0.015 x 0.011-in crack in the x-axis showed partial crack growth in

both the x- and z-direction. The crack propagation for this precracked case was very similar to the crack initiation cases shown in section 4.2. This crack growth is shown in Figure 9.

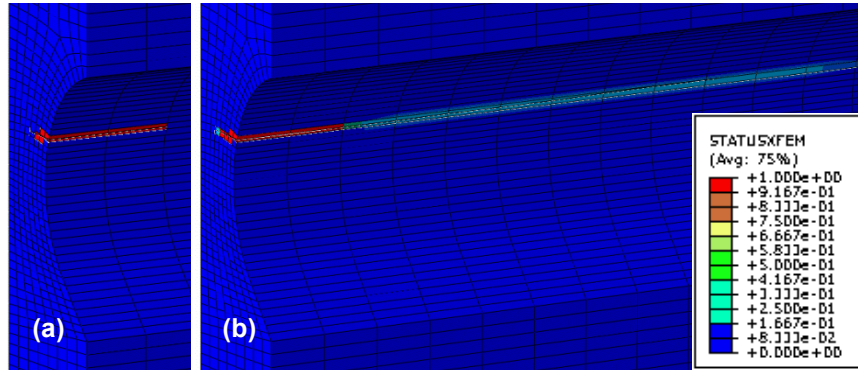


Figure 9. XFEM Crack 0.015 x 0.011-in (X-axis); (a) at t=0s, (b) at t=1s.

Results for the 0.015 x 0.011-in crack in the y-direction showed no crack growth in the y-direction and partial crack growth in the x- and z-direction. The crack propagation for this precracked case was not similar to the crack initiation cases shown in section 4.2. This crack growth is shown in Figure 10.

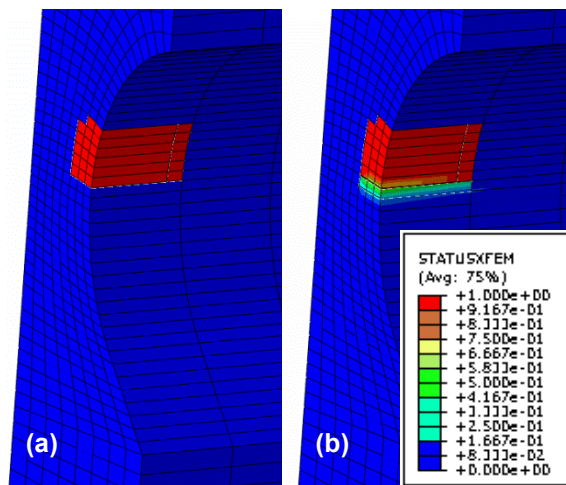


Figure 10. XFEM Crack 0.015 x 0.011-in (Y-axis); (a) at t=0s, (b) at t=1s.

4.4 Maximum Stress and Plastic Strain

Results for stress using the Von Mises criterion and plastic equivalent strain (PEEQ) are shown in Figures 11 and 12. Crack initiation occurs at a Von Mises stress of 144,562 psi, which is slightly lower than the yield stress of the material. As the crack continues to propagate yield stress is reached. Plastic strain was not exceeded.

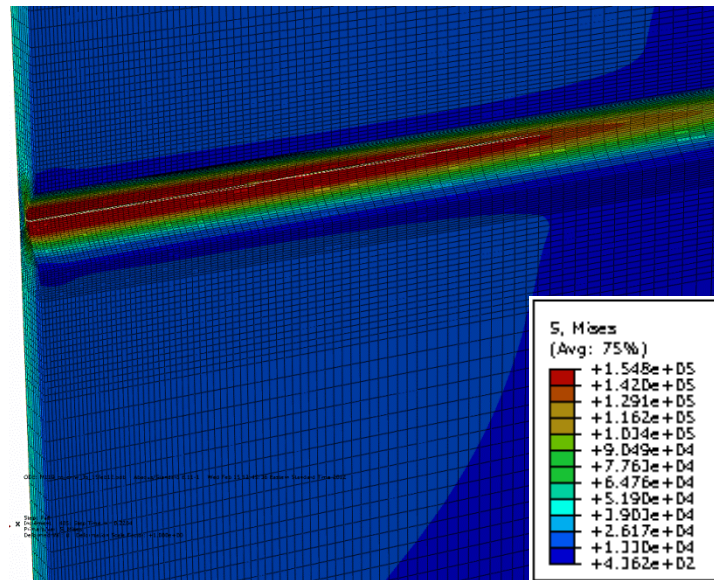


Figure 11. Von Mises Stress.

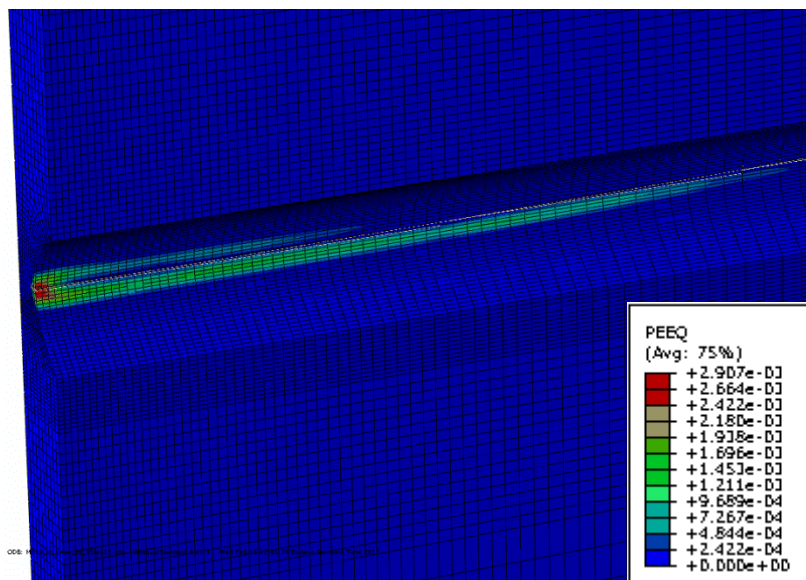


Figure 12. Plastic Strain.

4.5 Results of fe-safe Analysis

The results of fe-safe analysis showed a life cycle of 1071 cycles with the worst case location in the gouge cut portion of the plate where the highest stress was applied. Results are shown in Figure 13.

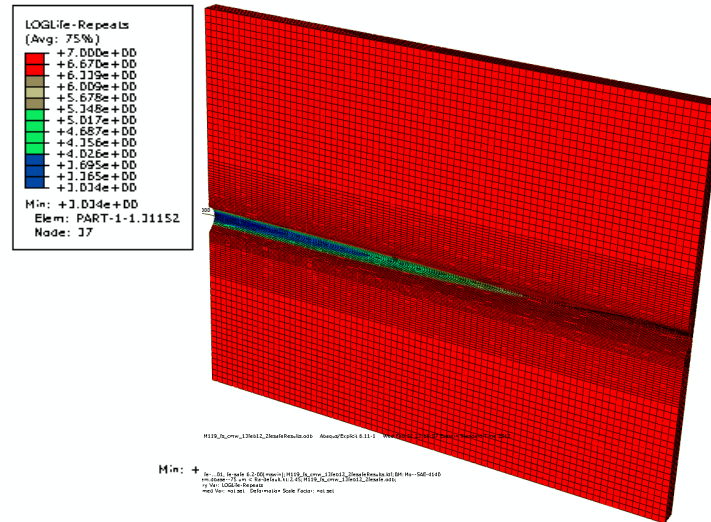


Figure 13. Log-life cycles (fe-safe).

4.6 Results of NASGRO Analysis

The results of the NASGRO analysis suggest that the crack becomes unstable after 106 cycles. The crack grows to 0.0721207 inches before failure, which is almost the thickness of the part. Results are shown in Figure 13.

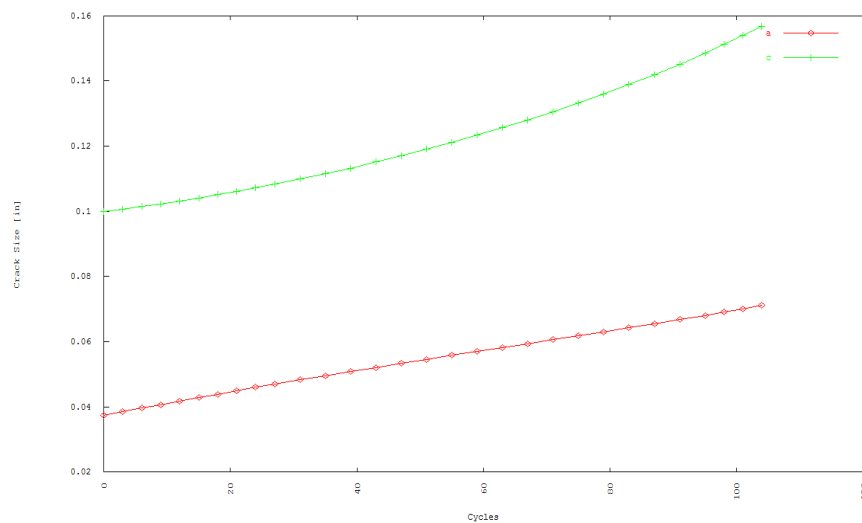


Figure 13. Crack size growth vs. Cycles (NASGRO).

5. Discussion

The Abaqus XFEM simulation method has been studied and shown to be an accurate predictor of a through-crack in a thin aluminum panel (Reinhardt, L., 2010). In this study, the method was implemented for the cradle of the M119 howitzer with a manufactured tooling defect. The method employed was able to show that partial crack growth is seen in the plate specimen used to model the portion of the cradle that has the gouge. All three methods used in this analysis show that the plate specimen fails the reliability requirement of 1100 mean rounds. As a result, the plate specimen was not able to prove that the channels with the tooling defects would survive the required amount of firings/cycles.

6. Conclusions

The Abaqus XFEM and NASGRO simulations for critical crack analysis and the fe-safe simulations for fatigue provide valuable insight into the pressure loads required in order to produce a crack and fatigue failure in the plate specimen. This method was done as a conservative case study to determine if the plate specimen could meet the reliability requirement of the cradle. As a result, further modeling of the channel is needed to determine the fatigue life cycle of the cradle with this tooling cut defect.

7. References

1. “Abaqus Users Manual V6.11”, Dassault Systems 2004 – 2011.

2. "Fe-safe User Manual Version 5.4," Safe Technology Limited, 2009.
3. "General Purpose Strain Gages – Linear Pattern," Micro-Measurements, Vishay Precision Group, 28 Jan 2010.
4. "Nasgro Fracture Mechanics and Fatigue Crack Growth Analysis Software Reference Manual Version 5.0," NASA Johnson Space Center and Southwest Research Institute, July 2006.
5. Powers, Gordon, "Test Plan Request for the 105-mm M119A2E1 Howitzer, Digital Fire Control System," Armaments Research, Development and Engineering Center, Yuma Proving Ground, Yuma, AZ, 9 May 2011.
6. Le, David, "Metrology and Simulation Division Visual and Fluorescent Penetrant Inspection," Ground Combat Systems Directorate, Yuma Test Center, Yuma, AZ, 11 June 2011.
7. Hespos, Michael, "M119 Cradle 95-15" Armaments Research, Development and Engineering Center, Picatinny Arsenal, NJ, 26 Oct 2011.
8. Hespos, Michael, "Elastic-Plastic Fracture Toughness Analysis," Armaments Research, Development and Engineering Center, 23 Jan 2012.
9. Abaqus Customer help answers 4396, 4397.
10. Reinhardt, L., Cordes, J.A., "XFEM Modeling of Mixed-Mode Cracks in Thin Aluminum Panels." Proceedings of the 2010 Simulia Customer Conference, Providence, RI, 24-27 May 2010.
11. R. Terhune, S. McDonald, M. Kotliar, "M119 HOWTIZER SADDLE GUN FIRE FINITE ELEMENT ANALYSIS." Technical Report ARMET-TR-11031. Armaments Research, Development and Engineering Center, Picatinny Arsenal, NJ, Dec 2011.