

X-FEM for Abaqus (XFA) Toolkit for Automated Crack Onset and Growth Simulations

Jay Shi¹, Jim Lua², Haim Waisman³, Phillip Liu⁴,
Ted Belytschko⁵, and N. Sukumar⁶

Global Engineering and Materials, Inc.

9 Goldfinch Terrace

East Lyme, CT 06333

¹jshi@gem-consultant.com

²jlua@gem-consultant.com

³hwaisman@gem-consultant.com

⁴pliu@gem-consultant.com

⁵Northwestern University

tedbelytschko@northwestern.edu

⁶University of California at Davis

nsukumar@ucdavis.edu

Abstract: A software tool for automated crack onset and growth simulations based on the eXtended Finite Element Method (X-FEM) has been developed. For the first time, this tool is able to simulate arbitrary crack growth and composite delamination without remeshing. The automated tool is integrated with Abaqus/Standard and Abaqus/CAE via the customization interfaces. It seamlessly works with the Commercial, Off-The-Shelf (COTS) Abaqus suite. Its unique features include: 1) CAE-based insertion of 2D or 3D multiple cracks with arbitrary shape of crack front independent of an existing mesh; 2) simulation of crack growth inside or between solid elements, and potentially along a shell/solid interface or along a shell/shell interface; 3) simulation of non self-similar crack growth along an arbitrary path or a user-specified interface; 4) extraction of near tip strain energy release rates via the modified VCCT; and 5) CAE-based data processing and visualization. The levelset method coupled with X-FEM is used to enrich the displacement field with jump and asymptotic near-tip solutions and track the crack geometry as it grows. A penalty-based formulation is developed within the UEL framework to simulate crack closure and frictional contact. To account for energy dissipation associated with the frictional contact, a modified VCCT approach is employed for an arbitrary crack front element with a partial contact zone. A fracture front tracking and levelset update module is used for either a user-specified growth size or a Paris-type fatigue law. Both the validity and applicability of the toolkit have been demonstrated via numerical examples.

Keywords: X-FEM, VCCT, Fatigue, Fracture & Failure, Crack Growth, Delamination, Abaqus/Standard, UEL

1. Introduction

The fracture failure mode is particularly significant in the damage tolerance of advanced composites or metallic joints, since manufacturing flaws and in-service damage most often manifest themselves as initial cracks. To assess the criticality of an initial flaw and its impact on the residual strength and life, a finite element analysis is performed, typically for various crack configuration, location, shape, and size. Nevertheless, conventional approaches are severely limited as any change in the geometrical characteristics of cracks requires re-meshing the finite element domain. This is a severe restriction and burdensome both for 2D and 3D analysis, especially for crack growth problems with complex geometries. Moreover, the convergence rate of conventional FEM is dominated by the nature of the solution near the point of singularity, which requires appropriate crack-tip singular elements to accurately represent cracks in FEM approximations.

To alleviate the computational burden associated with insertion of arbitrary cracks into an FEM model without conforming to its mesh, the extended finite element method (X-FEM) has recently been developed by Ted Belytschko and his research group (Belytschko, 1999, Moës, 1999, and Sukumar 2000). The X-FEM enables the modeling of stationary and growing cracks in solids subjected to static, dynamic, and fatigue loading. In essence, the idea is to enrich the finite element solution space with a Heaviside step function for displacement discontinuity and asymptotic branch functions for near tip singular fields. This has been shown to greatly improve convergence rates and in particular separate the geometry of the cracks from its finite element mesh.

We employ the levelset method coupled with X-FEM to represent the jump and asymptotic displacement enrichments and track the crack geometry as it grows. For example, a 3D crack can be defined by two almost-orthogonal levelsets that are derived from the signed distance functions. One function describes the crack surface (plane) in a 3D space and the other is used to describe the crack front. With this implicit description of the crack front, arbitrary evolving crack can be modeled without explicit geometric representation of the crack. Once the crack starts to propagate, the levelset values are updated, by either solving the governing hyperbolic equations or re-initializing the levelsets by calculating the signed distance functions for the new crack location. Under the Air Force funded SBIR program for residual strength and delamination failure prediction of composite structure GEM has successfully developed an add-on X-FEM software toolkit that works seamlessly with the common commercial, off-the-shelf (COTS) version of Abaqus/Standard and Abaqus/CAE for an automated crack onset and growth. Our partner LM Aero is currently using a 2D version of the toolkit for damage tolerance analysis of composite joints. We will release the 3D version to public by summer of 2009. The main XFA tool features are: 1) arbitrary insertion of multiple initial cracks that are independent of the finite element mesh; 2) characterization of a growing crack without remeshing; 3) accurate extraction of Strain Energy Release Rate (SERR) using the enriched near-tip displacement field via the modified Virtual Crack Closure Technique (VCCT); and 4) characterization of crack closure via a penalty contact algorithm.

The results of this R&D will have significant benefits in military and commercial industries. Specific applications of the software include structural integrity and damage tolerance design, evaluation of structural integrity and damage tolerance requirements to reduce reliance on testing, optimization of design concepts and reduction of cost associated with multiple configuration component testing, exploration of innovative manufacturing and/or composite bonding technologies for composite or metallic joints, and specification of damage tolerance design guidelines, inspection and repair requirements.

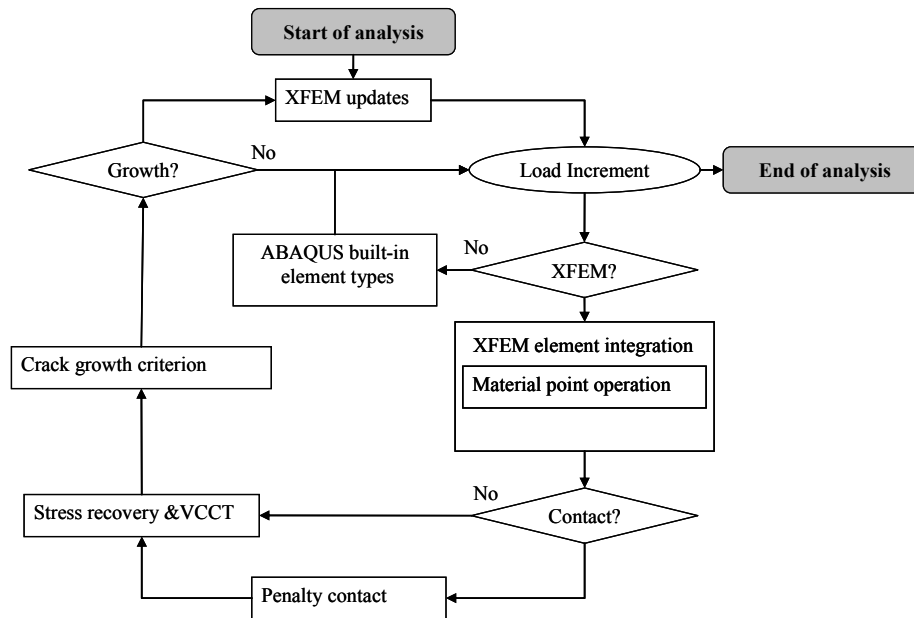


Figure 2. XFEM Flowchart for ABAQUS/Standard

For the 2D case the linear plane stress/strain quadrilaterals (CPS4, CPE4) are implemented and for the 3D case linear hex and tetra solids (C3D8, C3D4) will be implemented. For shell-shell or shell-solid interfaces an enriched interface element will also be developed to allow arbitrary shapes of crack front. Currently we support isotropic and orthotropic elastic materials and elasto-plastic materials with isotropic hardening and eventually most materials for solid elements will be supported. For the sub-domain outside of the X-FEM region, any default element types and materials can be used.

3. Penalty Contact in X-FEM Elements

A penalty-based formulation is developed inside the element operator to simulate crack closure and frictional contact. Considering these effects is critical in load cases involving shear modes.

One approach to simulate crack closure behavior is to apply the so-called global-local iterative algorithm proposed by (Dolbow, 2001). The method may exhibit slow convergence rate when the number of contact pairs become large. This issue is mainly due to the displacement jump and contact force that are solved in a local system and the unknowns do not appear in the global system of equations, ie, the contact constraints are decoupled. To enhance the solution accuracy and numerical efficiency, we have developed the direct approach where displacement jumps of contact surfaces are naturally interpolated by the extended displacement representation via X-FEM. Furthermore, a penalty-based formulation is developed so there is no Lagrange multiplier to take extra degrees of freedom and enter the system of equations via user elements. Following this approach, contact discontinuity can be resolved during the equilibrium iterations.

The contact surface is defined based on nodal levelsets. The discretization of the contact surface is achieved by assigning the accumulation points along the surface segments. For each crack

segment the contact condition with a frictional law is examined, which gives the additional stiffness terms and RHS. The element stiffness matrix and the residual force vector are updated for the next iteration. It should be noted that the condition of zero relative motion is approximated by a stiff elastic model. The model stiffness for the frictional condition is chosen such that the relative motion from the position of zero shear stress is bounded by a value, or, the allowable maximum elastic slip.

4. Modified VCCT

The modified VCCT is implemented to estimate the energy release rates. For the case that the crack fronts (tips) coincide with the element nodes, the algorithm can be summarized as the following:

1. Extend jump enrichments to the node in front of the tip. The elements connecting the tip node and the newly-enriched node become cut-through type. The tip node now contains displacement jump. The artificial “crack surface” between the tip node and the newly-enriched node can be treated as “crack processing zone” or “cohesive zone”.
2. Apply bonding constraint to the cohesive zone such that the displacement jump at the tip is minimized. This constraint is enforced using a penalty method. When the displacement jump is ignorably small, the penalty force can be regarded as the traction force in front of tip. The resultant force at the tip node is the equivalent to the reaction force in the conventional VCCT’s spring double-node model.
3. Multiply the nodal force with the crack opening displacement behind the crack and at the distance equivalent to the size of cohesive zone to obtain the total energy released.

A variant of this method can also be applied to solve internal crack problems. The actual implementation of the VCCT is analogous to the contact mechanics algorithm mentioned previously.

5. XFA Tool Validation

5.1 Validation 1: Edge-cracked plate under surface traction (Liu, 2004) to verify the accuracy of stress intensity factors estimation using the modified VCCT in XFA toolkit for an internal crack

An edge-cracked plate, as shown schematically in Figure 3, is fixed at the bottom and subjected to a uniform shear stress of magnitude 1.0 unit along the top edge. The plate has a length (L) of 16 units, width (W) of 7 units, and a crack length (a) of 3.5 units. A plane strain condition is assumed with a Poisson’s ratio of 0.3 and elastic modulus of 100 units. A uniform mesh consisting of 23×47 quad elements is created. The resulting respective diagonal length of an element is 0.457 units.

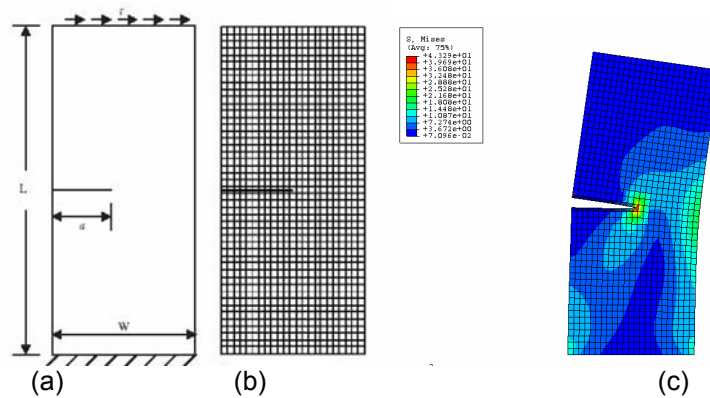


Figure 3. Edge-cracked plate under shear: (a) geometry and loads, (b) FEM model with the 23×47 mesh, and (c) XFA simulation result

The reference solution provided in the paper is $K_1=34.0$ and $K_2 = 4.55$. Our estimation using VCCT gives K_1 and K_2 of 34.3 and 4.51, respectively. The relative error as compared with the reference solution is about 0.88%.

5.2 Validation 2: Bi-material interface central crack in a plate under tensile load (Nagashima, 2003) to verify the VCCT procedure for bi-material interface

In this test we consider the following model with bi-orthotropic interface crack: a rectangular plate with a bi-material center crack subjected to a surface traction, as shown in Figure 4. The width $2W$ and length $2H$ of the plate is given by 100 and 200 mm, respectively.

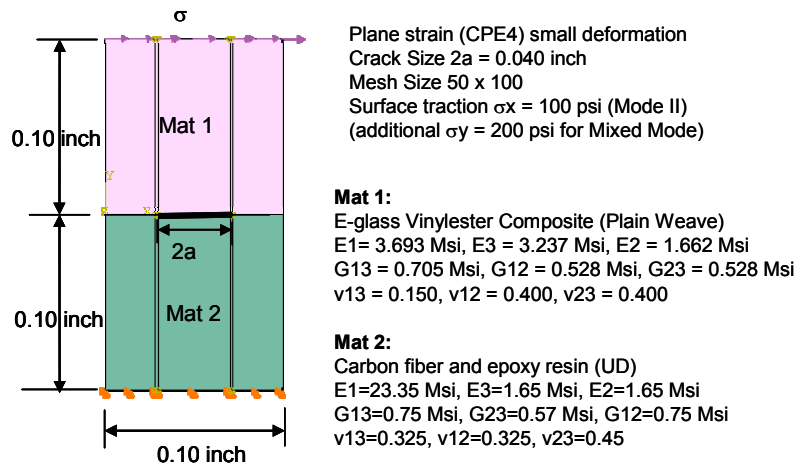


Figure 4. Rectangular plate with a bi-material center crack

The XFA result is compared CAI's formulation (provided by LM Aero) and excellent agreement is found as in Table 1.

	XFA's VCCT	CAI's VCCT
Left tip:		
Displacement n	2x2.36208e-006	
Displacement s	2x5.47384e-007	
Traction n	2.49299e+000	2.480
Traction s	1.29666e+000	1.283
GI	2.94432e-003	0.002934
GII	3.54884e-004	0.000353
Right tip:		
Displacement n	2x8.81282e-007	
Displacement s	2x3.94411e-007	
Traction n	7.47925e-001	0.7436
Traction s	9.04461e-001	0.8945
GI	3.29566e-004	0.000328
GII	1.78365e-004	0.000178

Table 1. Results using XFA's VCCT and CAI's VCCT.

5.3 Validation 3: DCB/ENF tests to verify VCCT for composite delamination

Both DCB/ENF analyses and tests have been performed extensively by research communities and government labs. Both the material system and component geometry of DCB and ENF specimen were defined by the Navy Lab in Carderock (NSWCCD). The composite material system is a plain weave laminated composite with E-glass fiber and vinyl ester resin. Both the VCCT in Abaqus and the cohesive model for LS-DYNA have been used by NSWCCD and GEM, respectively, for response and delamination onset prediction.

The 3D orthotropic material properties and model parameters are defined in Figure 5 below. Using the same material and model parameters, the X-FEM based delamination tool is used to compute the SERRs at the onset of delamination growth. The X-FEM results are compared with the VCCT and cohesive model predictions.

Orthotropic Material Model

E1=3.693e+06 psi; E2=3.237e+06 psi; E3=1.662e+06 psi; G12=0.705e+06 psi
G13=0.528e+06 psi; G23=0.528e+06 psi, $\nu_{12}=0.15$; $\nu_{13}=\nu_{23}=0.4$

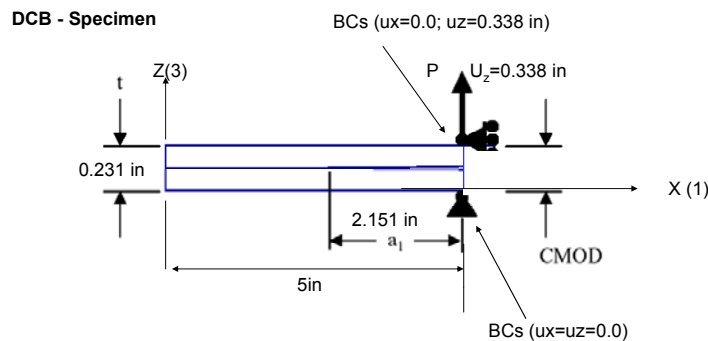


Figure 5. Problem statement of DCB specimen

The mesh size used in the XFEM model is the same as used by NSWCCD for their Abaqus + VCCT model with the mesh size of 0.012 inch. The GI prediction from Abaqus' VCCT and the LS-DYNA's cohesive model is 4.77 in lb/in² and 4.785 in lb/in², respectively. The applied load measured from tests is $P = 20$ lb at the onset of delamination growth.

The response prediction from the XFA tool is given below. The following Figure 6 show the Mises stresses of the DCB.

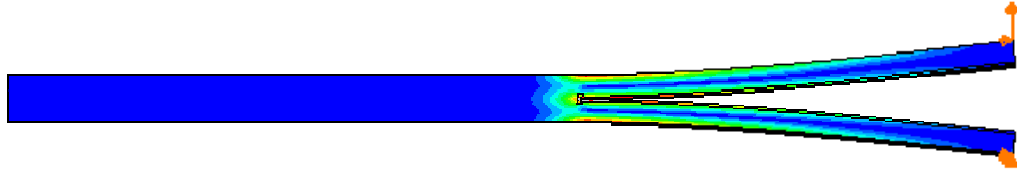


Figure 6. Mises stress of the DCB specimen using the XFA tool

The X-FEM predicts $GI = 4.765$ in lb/in² and $GII = 2.33e-04$ in lb/in² with the reaction force of 21.67 lb. An excellent prediction in GI has been achieved from the XFEM tool simulation of the DCB specimen. Given the dominant Mode I characteristic in the DCB specimen, the predicted GII is extremely small.

The setup and modeling parameters of the ENF test coupon is the shown in Figure 7. The measured center deflection at the onset of Mode II delamination growth is 0.168 in. The corresponding load at the delamination growth is 205 lb.

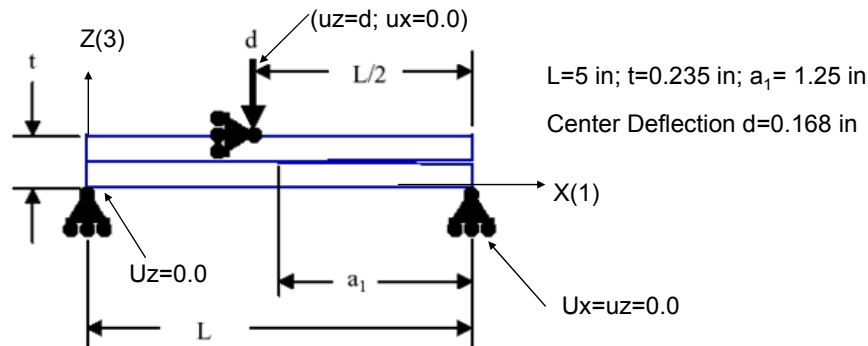


Figure 7. Problem Statement of ENF Specimen

The GII predicted from NSWCCD's VCCT and GEM's cohesive model is 6.80 in lb/in², and 6.90 in lb/in², respectively. The GII prediction from the XFA tool is 6.61 in lb/in², which is slightly lower than the VCCT prediction (3%). In addition, the failure load predicted from the XFA tool is 211.5 lb, which slightly high than the measured failure load of 205 lb. Again the excellent agreement in load and SEER prediction has demonstrated the validity and capability of the XFA tool in characterizing Mode I and II. The Mises stress contour of the ENF specimen under zoom-out and zoom-in configurations is given in Figure 8 below.

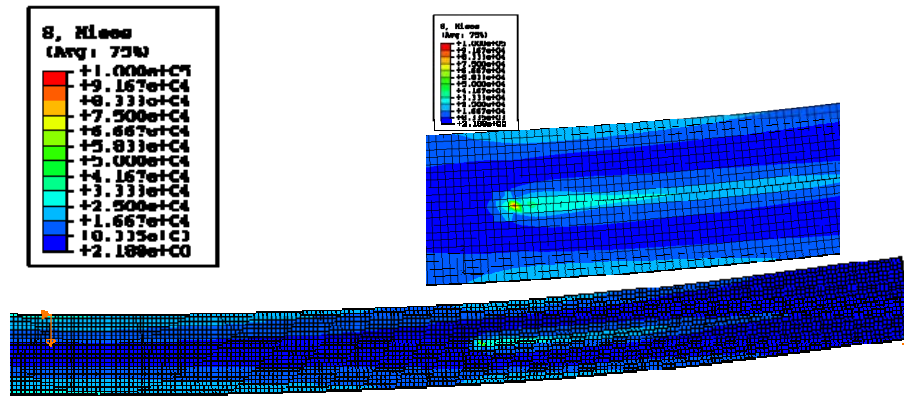


Figure 8. Bending stress distribution of the ENF specimen

5.4 Validation 4: 3D cube under mixed mode loading to verify VCCT compared to the conventional spring double-node model in Abaqus

A simple model that contains a 1x1x1 unit cube and 800 brick elements with elastic material is considered, as illustrated in Figure 9. The cube is loaded with surface traction in all directions over the top and bottom surfaces. The side surface in front of the crack front is completely fixed.

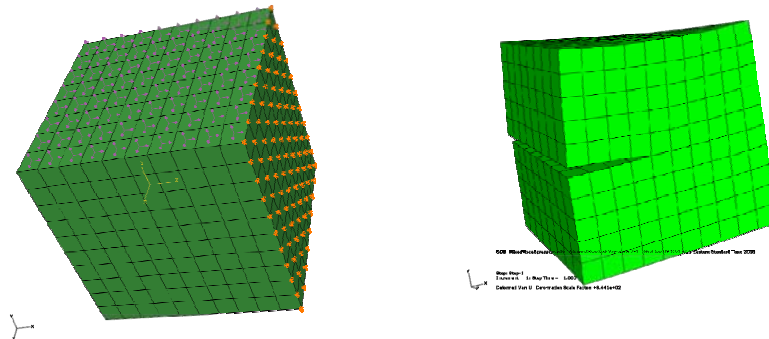


Figure 9. Illustration of mixed mode loading and its corresponding displacement

In Table 2 the displacement jumps and the concentrated forces at all 9 tip nodes are compared with the conventional, spring double-node model using Abaqus. Again the excellent agreement between the results is found.

Node	Fx (Abaqus)	Fx (XFA)	Fy (Abaqus)	Fy (XFA)	Fz (Abaqus)	Fz (XFA)
1	-3.431	3.4309e+000	2.544	2.5437e+000	0.7420	7.4199e-001
2	-5.316	5.3161e+000	5.456	5.4564e+000	1.179	1.1785e+000
3	-3.944	3.9441e+000	5.781	5.7808e+000	1.312	1.3119e+000
4	-2.850	2.8501e+000	5.853	5.8534e+000	1.427	1.4274e+000
5	-1.884	1.8843e+000	5.875	5.8754e+000	1.474	1.4743e+000

6	-0.9440	9.4396e-001	5.853	5.8534e+000	1.370	1.3701e+000
7	5.664E-02	5.6647e-002	5.781	5.7808e+000	1.144	1.1439e+000
8	1.212	1.2117e+000	5.456	5.4564e+000	0.8245	8.2452e-001
9	1.145	1.1454e+000	2.544	2.5437e+000	0.4071	4.0714e-001

Table 2. Forces extracted from XFA and the Abaqus double-node model

5.5 Validation 5: Non-planar growth of an inclined penny-shape crack to verify the capability of nonplanar crack growth

Finally, to illustrate our 3D crack growth capability, a penny shaped crack in a 3D elastic body is analyzed as shown in Figure 10. We consider an inclined penny crack ($\theta = \pm 45$ degrees) that is under uniaxial tension. Since the loading at the crack front is of mixed-mode, the crack kinks. In this model a 17-thousand-node mesh is used. The snapshots of crack propagation at $t=0.00, 0.33, 0.67$, and 1.00 are plotted, respectively, in Figure 10. The simulations reveal the non-planar character of the growth and the results illustrate that our crack update algorithm is able to capture the representation of the crack at each step.

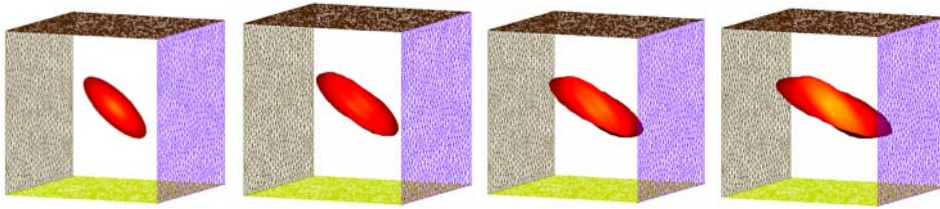


Figure 10. Nonplanar growth of the penny-shape crack under simple tension

6. XFA tool application: Fatigue crack growth of aluminum T-joint and fatigue life prediction

T-weldments are widely used components in various engineering structures. Its general structure contains three regions: base, web and welding. Typically the base and web are connected by the welding while each component is made of different materials. The dimensions of our T-weldment model are based on (Latorre, 2002), which is a T-stiffener used in hull panels of ship structures.

Material heterogeneity is an important concern in the T-weldment fatigue analysis. The welding process causes hot spots at the weldment toe which sometimes reaches a temperature of over 800 K. Often the temperature ramps down to the room temperature on the other side of the panel due to high heat conductivity. Phase transformations in the weldment and the Heat Affected Zone (HAZ), and the selection of different base, web, and weldment materials are eventually the factors leading to a significant amount of residual stresses near the welding zone. The size of the HAZ can only be determined empirically. The final geometry of the simplified T-joint model and the mesh is shown in Figure 11. The component is simply supported with the concentrated load, P equal to 55N, which is applied on a reference point. This reference point is then rigidly coupled to the top-end surface of the web.

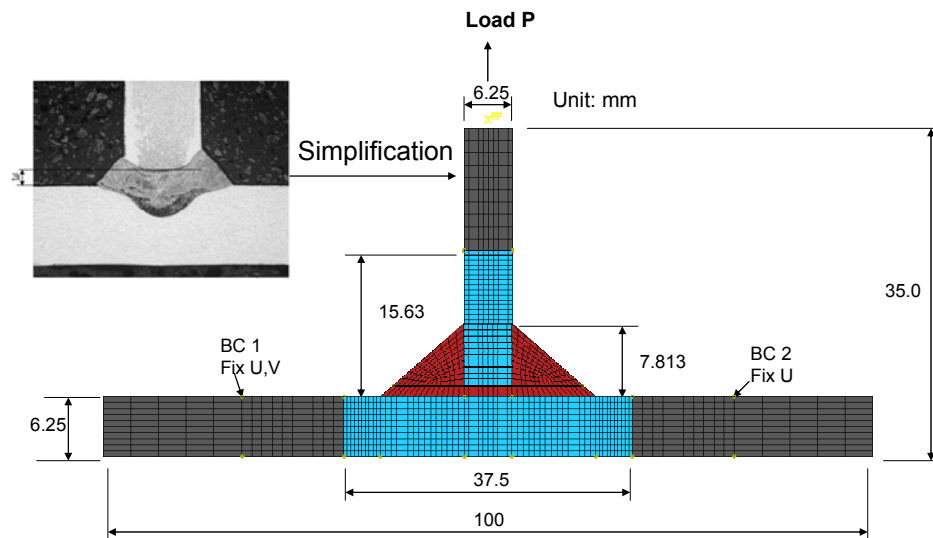


Figure 11. The XFA model of aluminum T-joint specimen

In this example, the base and the web are aluminum alloy 5083 and the weldment is aluminum alloy 5183. The properties of both alloys as well as the HAZ are adopted from (Galatolo, 1997). A Ramberg-Osgood model, typically used for aluminum, is used for each material component with material parameters obtained from experimental data. The material parameters for each region are listed in the Table 3.

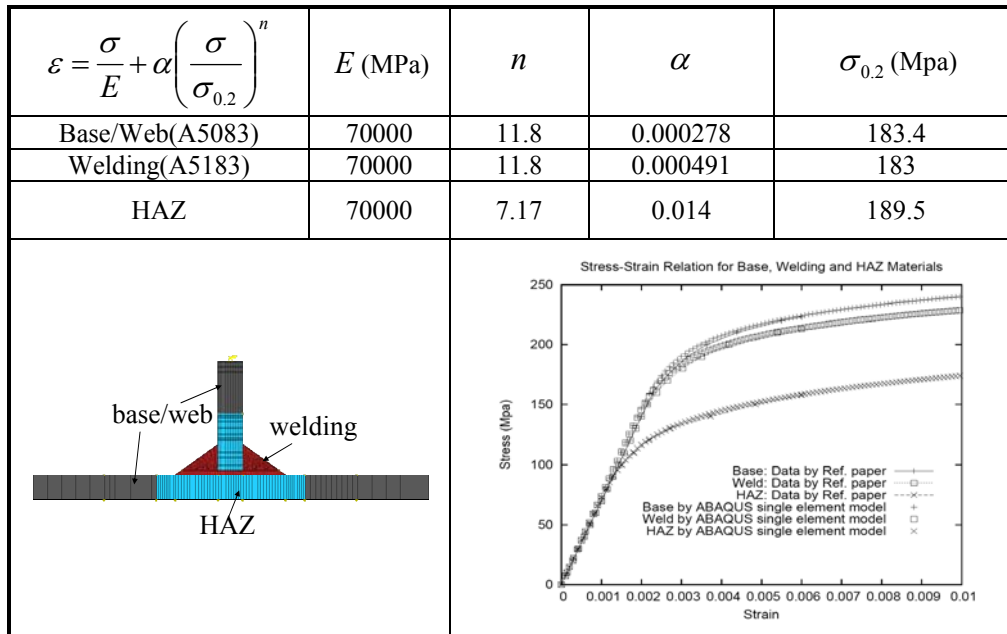


Table 3. Material parameters for T-joint base, welding, and HAZ regions

The static analysis of the uncracked model suggests that it is a high-cycle fatigue problem with a maximum equivalent plastic strain (PEEQ) less than 0.2%; therefore, a simple, elasto-plastic with isotropic hardening UMAT subroutine is developed for the user element domain and the equivalent material model (*PLASTIC) is specified for the rest of the model using the built-in element type: CPE4. It should be noted that for X-FEM elements, both the location and number of Gauss points between load increments may be changed due to the propagating crack and hence material state variables must be updated consistently. For all analyses small deformation theory is assumed.

The residual stress distribution is affected by the material heterogeneity, the component geometry, and the temperature field. Generally it is very difficult to obtain accurate distribution by simulation that considers all influential factors. Nevertheless, it is common to use measured distribution from experiments. For instance, in (Mann, 2006) the residual stress components of a similar aluminum structure are measured. The tensile stresses are found near the weldment toe region, which then turn to compressive stresses away from the toe, and rapidly attenuate to zero further away. To simulate a similar residual stress distribution, the user subroutine SIGINI is developed to calculate the initial stress distribution at Gauss points. Since such a superimposed stress distribution is unbalanced, a static step is first performed to obtain self-balanced distribution, which is then followed by the actual, fatigue analysis. The self-balanced residual stress distribution is shown in Figure 12. In the same figure, the simulated stress profile starting from the weldment toe and along the base panel is plotted. The general profile matches well with the experimental result in (Mann, 2006).

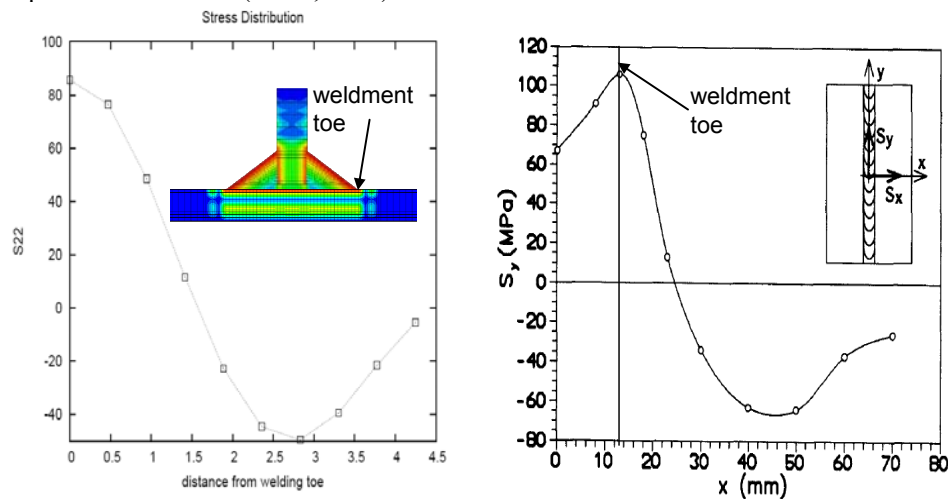


Figure 12. Residual stress profile introduced at the beginning of fatigue analysis

It is usually assumed that the fatigue crack is initiated where the maximum Mises stress, or the “hot spot” stress, is located. This assumption is supported by experimental observation such as in (Latorre, 2002). More interestingly, as shown in Figure 13, the fatigue crack starts at the weldment tip not only because the stress is magnified but also because the weld filler tends not to properly fuse with the base metal and introduces the cold lap. The direction of the initial crack is assumed

to normal to the maximum principal stress at this location. An initial crack size of 0.5mm is assumed.

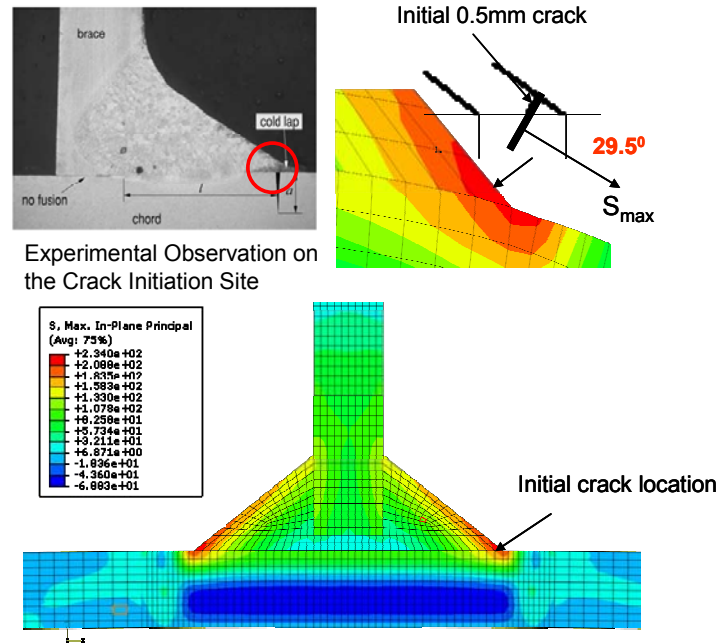


Figure 13. The fatigue crack starts at the weldment tip not only because the stress is magnified but also because the weld filler tends not to properly fuse with the base metal and introduces the cold lap

To consider the effect of residual stresses, a variant of the Paris fatigue law, is used. The total time-to-failure in the fatigue analysis step is treated as the number of cycles, dN . At the end of each increment the range of stress intensity factor (SIF), ΔK is computed at the crack tip. The stress ratio, R , is then calculated as the ratio of the residual stress component at the crack growth normal direction to the stress component along the same direction and under the full load condition. Once the modified SIF range and the stress ratio are obtained, the crack growth size, da , can be estimated using the Walker's model: $da / dN = C(\Delta K_{Walker})^m$, where

$$\Delta K_{Walker} = \frac{\Delta K}{(1-R)^{1-\gamma}}$$

and the fatigue strength parameters: $C = 1.24E-14$, $m = 4.01$, and

$\gamma = 0.6$, are adopted from (Brant, 1998). Similar to the initial crack orientation, the crack growth direction is assumed normal to the maximum principal stress and is estimated at the end of each increment. The component is considered failure once the total crack size has reached the critical size of 2.0 mm. The fatigue life as the number of cycles is equivalent to the step time when the critical size is reached. It is noted that the X-FEM elements are used only in the region around the initial crack. In Figure 14 a few snapshots of the Mises stress in the X-FEM region and corresponding nodal enrichment type are shown.

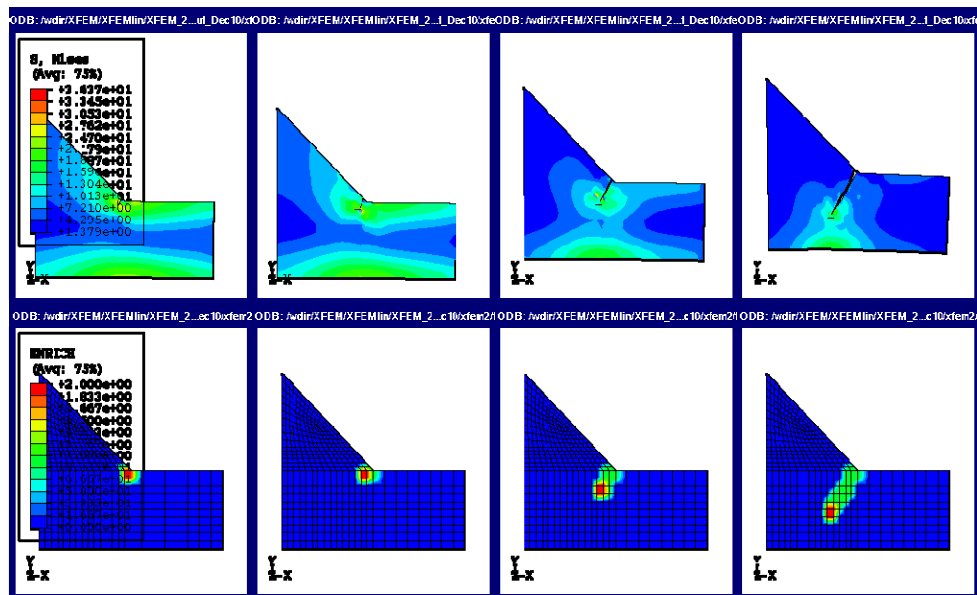


Figure 14. Snapshots of the Mises stress in the X-FEM region and corresponding nodal enrichment types

It is obvious that by using X-FEM, the crack growth path is completely independent on the mesh. The tip can move arbitrarily in the region and no remeshing is needed. For this particular model the component fails with final crack size equal to 2.0194 mm and the stress ratio equal to 0.4808. The number of cycles during fatigue crack growth is 4.58E+06.

To visualize the crack opening in its deformed shape, a post-processing module is developed as part of the XFA toolkit to slice the crack-intersected elements based on the levelset information and the enriched degrees of freedom. This kind of element slicing is for visualization purpose only. The process has nothing to do with local remeshing because the resulting subelements do not exist during the analysis phase. The final snapshot of Mises stress and the crack opening are shown in figure 15.

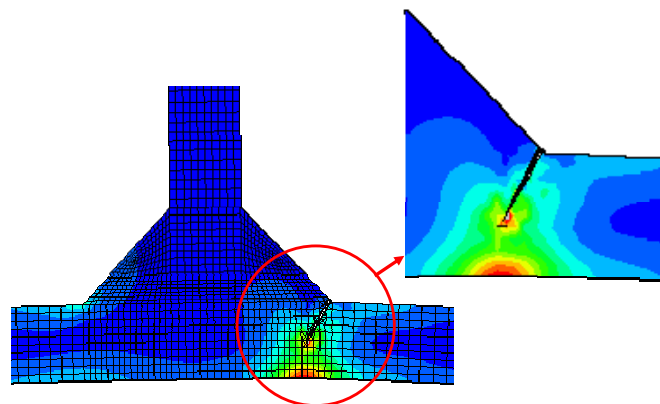


Figure 15. Mises stress and crack opening at failure

7. Conclusion

Global Engineering & Materials, Inc has developed an add-on X-FEM toolkit for Abaqus (XFA) that works seamlessly with the common commercial, off-the-shelf (COTS) version of Abaqus software suite for an automated crack onset and growth prediction analysis. During the development process, rigorous validation has been made to ensure the tool is robust, accurate and generates high fidelity predictions in the fracture and fatigue analysis. The tool has great potential to be used in various designs and analysis practices such as the one illustrated in the last example. We have scheduled to release this tool by summer 2009. Support from the Air Force SBIR program and our industry partners, including SIMULA of Dassault Systems, LM Aero, and Bell Helicopter are highly acknowledged.

8. References

1. Belytschko T, Black T, “Elastic crack growth in finite elements with minimal remeshing”, International Journal for Numerical Methods in Engineering, 45, 601-620, 1999.
2. Brandt, Lawrence, and Sonsino, “The fatigue behavior of 5 and 25 mm 5083 Aluminum-Alloy Weldments”, Research Report, IIW – Document XIII – 1718 – 98.
3. Dolbow, J., N. Moës, and T. Belytschko, “An Extended Finite Element Method for Modeling Crack Growth with Frictional Contact”, Comput. Methods Appl. Mech. Engrg. 190, 6825-6846, 2001.
4. Galatolo R. “Fatigue crack propagation in residual stress fields of welded plates”, Int. J. Fatigue, 19, 43-49, 1997.
5. Latorre R. G., “Stress analysis of a transversely loaded aluminium weldment”, Marine Structures, 15, 175–191, 2002.
6. Liu, et al, “XFEM for direct evaluation of mixed mode SIFs in homogenous and bi-materials”, International Journal for Numerical Methods in Engineering, 59:1103, 2004.
7. Mann, Tveiten, and Harkegard, “Fatigue crack growth analysis of welded aluminum RHS T-joints with manipulated residual stress level”, Fatigue Fract Engng Mater Struct, 29, 113–122, 2006.
8. Moës, N, Dolbow, J, Belytschko, T., “A finite element method for crack growth without remeshing”, International Journal for Numerical Methods in Engineering, 46, 131-150, 1999.
9. Nagashima, et al, “Stress intensity factor analysis of interface cracks using X-FEM”, International Journal for Numerical Methods in Engineering, 56:1153, 2003.
10. Sukumar, N, Moës, N, Moran, B, Belytschko T., “Extended finite element method for three-dimensional crack modeling”, International Journal for Numerical Methods in Engineering, 48, 1549-1570, 2000.