Best Practices in Modeling Crack Propagation with the Extended Finite Element Method (XFEM) in Abaqus/Standard

Version 1.0 - 4/16/2018

Written by: Ying CHEN

Edited by: Arati Desai

SIMULIA Abaqus 2018

3DEXPERIENCE

Executive Summary

The Extended Finite Element Method (XFEM) in Abaqus/Standard can be used for modeling the fracture of bulk material by allowing crack initiation and propagation on the interior of finite elements. This document provides an overview of how to create an XFEM crack propagation model. The following four modeling steps are described in detail:

- *Definition of an enrichment region*
- *Definition of an initial crack*
- *Definition of a fracture criteria*
- *Definition of contact behavior for cracked element surfaces*

Modeling tips for each step and analysis settings to improve convergence are also discussed. Two example input files are available to illustrate the use of XFEM to predict crack initiation and propagation using two different fracture criteria: the cohesive segment method and the Linear Elastic Fracture Mechanics (LEFM) method.

The target audience for this Best Practice are Engineers and FEA specialists concerned with modeling fracture and failure.

3DS.COM © Dassault Systèmes

Contents

Best Practice

1. Introduction

Abaqus/Standard offers two different methods for performing fracture mechanics: the conventional finite element method and the Extended Finite Element Method (XFEM).

With the conventional finite element method, the onset of cracking can be studied by contour integral evaluation that computes the J-integral, the C_t-integral (for creep), the stress intensity factors, the crack propagation direction, and the T-stress. Focused meshes are generally required for these types of problems to obtain path independent contour integrals. Crack growth along predefined element boundaries can also be modeled in which the crack debonds along initially partially bonded surfaces.

With XFEM, crack initiation and propagation along an arbitrary, solution-dependent path can be simulated. Cracks nucleate and propagate in the element interior, allowing fracture of the bulk material to be modeled. Contour integral evaluation can also be performed without the need to refine the mesh around the crack tip. Modeling cracks using XFEM provides two main advantages: the crack location and geometry can be independent of the mesh; and the crack growth path need not be known in advance. XFEM can be also be used in conjunction with other crack propagation capabilities available with the conventional finite element method to model both bulk material failure and interfacial failure. Note that XFEM is intended for single (or non-interacting) cracks in the structure. Crack merging /branching cannot be modeled.

The main focus of this document is to provide best practices for modeling crack propagation with XFEM. For contour integral evaluation, using XFEM and fracture mechanics capabilities with the conventional finite element method, you may refer to the references given at the end of the document.

2. Creating an XFEM Crack Propagation Model

We will describe the development of an XFEM crack propagation model in four primary steps.

2.1. Define an Enrichment Region

XFEM models a crack in an element by enriching the degrees of freedom with special displacement functions. The enrichment regions in an XFEM analysis should consist of elements that are presently intersected by cracks and those that are likely to be intersected by cracks as they propagate.

When defining an enrichment region, you must specify the type of crack (stationary or propagating), the name for the enriched feature, and the element set of the enriched region. You may also need to assign a surface interaction property (Section 2.4).

Input File Usage:

```
*ENRICHMENT, TYPE=PROPAGATION CRACK, NAME=CRACK-1,
ELSET=SELECTED_ELEMENTS, INTERACTION=CONTACT-1
```

If several cracks are expected to sequentially nucleate at different locations during an analysis, multiple enrichment regions can be specified.

The enriched region must not include stress hot spots due to boundary conditions or other modeling features. Otherwise, unintended cracks may initiate at such locations.

2.1.1. Supported Element Types
Only stress/displacement or displacement/pore pressure solid continuum elements can be associated with an enriched feature. The displacement/pore pressure solid continuum elements are used for modeling hydraulically driven fracture which will not be covered in this document.

For propagating cracks, supported element types include first order 2D and 3D continuum elements (CPE4, CPS4, CAX4, C3D8, and C3D4), second order tetrahedral elements (C3D10), and first order displacement/pore pressure elements. For stationary cracks, supported element types include only first order 3D continuum elements (C3D8 and C3D4) and second order tetrahedral elements (C3D10). The reduced integration/incompatible mode/hybrid counterparts of these elements are also supported. Composite solid elements are not supported with an enriched feature. When modeling crack propagation in 3D, hexahedral elements typically give better results than tetrahedral elements.

2.1.2. Mesh Considerations
The enriched portion of the finite element mesh needs to be sufficiently refined, as would be the case in a conventional stress analysis; a mesh sensitivity study is recommended. If the mesh is too coarse, the crack propagation process may not be captured in detail. If the mesh is excessively refined, the crack propagation may be slow. A structured mesh predominantly consists of regular hexagonal elements in 3D and quadrilateral elements in 2D. Structured mesh is preferred over a free tetrahedral mesh because structured meshes typically give a more accurate prediction of stress and strain fields ahead of the crack tip. If the regions expected to crack are known in advance, partition the geometry so that the refined regions can be kept to a reasonable size. If needed, tie constraints can be introduced to join the finely meshed enriched regions to the rest of the model.

The XFEM method does not require the mesh to conform to the crack geometry however; the method does not permit a crack to lie along the element boundary. Therefore, in simple problems where the crack is expected to propagate along a straight line; the mesh should be designed such that the expected crack propagation path does not lie along the element boundary.

2.1.3. Material Options

Any of the mechanical constitutive models in Abaqus/Standard, including user-defined materials, can be used to model the mechanical behavior of enriched elements in a crack propagation analysis. The inelastic definition at a material point must be used in conjunction with the linear elastic or hypoelastic material model. When evaluating the contour integral for a stationary crack, only isotropic elastic materials are supported.

When a user-defined material (UMAT or UHYPER) is included in an XFEM analysis, the penalty stiffness of the cracked element contact definition must be manually provided.

Input File Usage:

```
*SURFACE BEHAVIOR, PRESSURE-OVERCLOSURE=LINEAR
*SURFACE BEHAVIOR, PENALTY
*SURFACE BEHAVIOR, AUGMENTED LAGRANGE
```

2.2.   Define an Initial Crack

Unlike the crack propagation capability within the conventional FEM framework, an initial crack is not required in an XFEM analysis. If there is no initial crack, a crack will nucleate when the crack initiation criterion (Section 2.3) is satisfied. If one or multiple initial cracks do exist, the location of the initial cracks must be specified. The initial crack is defined by using the *INITIAL CONDITION, TYPE=ENRICHMENT keyword option as illustrated in Figure 1.
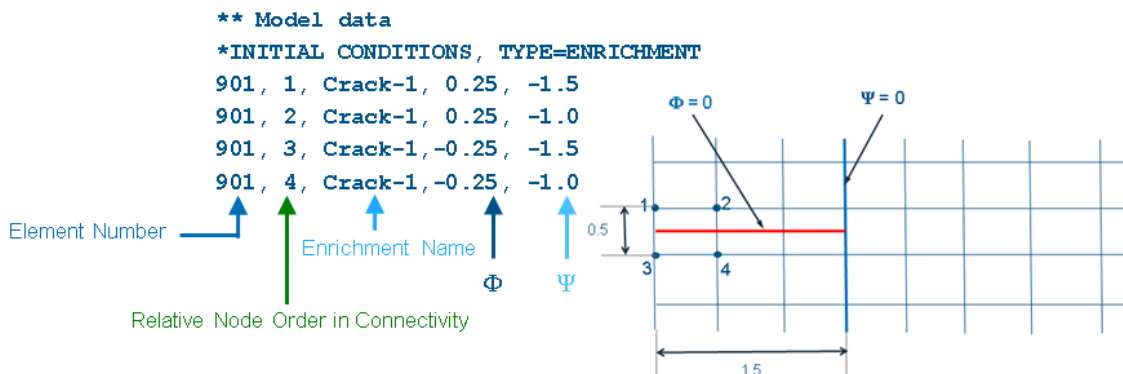

Figure 1. Keyword interface for defining an initial crack

Abaqus uses the level set method to locate the crack surface and the crack front. With the level set method, two signed distance functions per node, φ and ψ, are used to completely describe the crack. The nodal value of the first signed distance function, φ, is the *signed* distance of the node from the crack surface (positive value on one side of the crack face, negative on the other). The nodal value of the second distance function, ψ, is the *signed* distance of the node from an almost-orthogonal surface passing through the crack front (zero value on this surface and negative on the side towards the crack). The values of the signed distance functions need to be specified only at nodes belonging to elements cut by the initial crack.

The XFEM crack is not allowed to lie along the element boundary. Therefore, the first signed distance function must be either greater than or less than zero and cannot be equal to zero. If an initial crack has to be defined at the boundaries of an element, a very small positive or negative value for the first signed distance function must be specified. This slightly offsets the crack from the element boundary. However, an initial crack very close and parallel to the element boundary should be avoided because it may cause convergence difficulties.

2.3. Define the Fracture Criteria

Abaqus offers two approaches to define the fracture criterion in XFEM: the cohesive segments method and the Linear Elastic Fracture Mechanics (LEFM) method.

The cohesive segments method can be used for modeling brittle or ductile fracture. It is defined by a linear elastic traction-separation model, a damage initiation criterion, and a damage evolution law, all specified in the bulk material definition. The linear elastic traction-separation model relates traction stresses to crack surface separations, and is defined by the elastic properties of the material in the enriched elements. The damage initiation represents the beginning of degradation of the cohesive stiffness in an enriched element. A crack nucleates in an enriched element when the stresses or strains satisfy the damage initiation criterion. The damage evolution law describes the rate at which the cohesive stiffness is degraded until the complete failure of the enriched element.

The LEFM method is more appropriate for modeling brittle fracture. It uses the modified Virtual Crack Closure Technique (VCCT) to calculate the strain energy release rate at the crack tip. The crack extends when the strain energy release rate at the crack tip reaches the critical value. The critical energy release rates are specified in the fracture criterion defined in the surface property definition. The surface property needs to be referenced when defining the enrichment region. If there is no pre-existing crack, the crack initiation criterion needs to be specified in the material definition, and it is only used for the crack nucleation of the first element. The LEFM method can also be used to simulate growth of

a discrete crack subjected to sub-critical cyclic loading in a low-cycle fatigue analysis. The topic of low-cycle fatigue analysis will not be covered in this document.

By default, in the cohesive segments method, when a maximum principal stress/strain damage initiation criterion is used; the newly introduced crack will be orthogonal to the maximum principal stress/strain direction. If a different initiation criterion is used, the crack propagation direction must be specified. In the LEFM method, the crack propagation direction must be specified.

Viscous regularization can be included to help overcome convergence difficulties that may arise with the associated softening behavior and stiffness degradation in a crack propagation analysis.

Input File Usage, cohesive segments method:

```
*MATERIAL
*DAMAGE INITIATION, CRITERION=[MAXPS | MAXPE | MAXS | MASE |
QUADS | QUADE], NORMAL DIRECTION=[1 | 2], TOLERANCE=0.05
 22E6
*DAMAGE EVOLUTION, TYPE=ENERGY, MIXED MODE BEHAVIOR=[BK | POWER
LAW], POWER=1.0
 2870.0,2870.0,2870.0
*DAMAGE STABILIZATION
 1.0E-5
```

Input File Usage, LEFM method:

```
*MATERIAL
*DAMAGE INITIATION, CRITERION=[MAXPS | MAXPE | MAXS | MASE |
QUADS | QUADE], NORMAL DIRECTION=[1 |2], TOLERANCE=0.05
 22E6
*DAMAGE EVOLUTION, TYPE=ENERGY, MIXED MODE BEHAVIOR=[BK | POWER
LAW], POWER=1.0
 2870.0,2870.0,2870.0
*DAMAGE STABILIZATION
 1.0E-5
*SURFACE INTERACTION, NAME=CONTACT
*SURFACE BEHAVIOR
*FRACTURE CRITERION, TYPE=[VCCT | ENHANCED VCCT], MIXED MODE
BEHAVIOR=[BK | POWER | REEDER], NORMAL DIRECTION=[MTS | 1 | 2],
VISCOSITY=1.0E-5, TOLERANCE=0.2
 2870.0, 2870.0, 2870.0, 1.0, 1.0, 1.0
```

2.4.   Damage Initiation

There are three stress-based and three strain-based built-in crack initiation criteria. The stress-based crack initiation criteria are typically suitable for brittle fracture, and the strain-based crack initiation criteria are typically suitable for ductile fracture. A user-defined crack initiation criterion can also be specified in user subroutine UDMGINI.

By default, the crack initiation criterion and the crack propagation direction are based on values of the stress/strain at the element centroid ahead of the crack tip. However, if the finite element mesh in the vicinity of the crack tip is coarse relative to the gradients in the stress/strain fields, the default centroidal approximation may not be sufficient. In such cases you can use the stress/strain extrapolated to the crack tip for determination of both the crack initiation criterion and crack propagation direction. In the case of coarse and/or unstructured meshes you may use the stress/strain averaged over a group of elements around the crack tip in the enriched region to improve the accuracy of the computed crack propagation direction.

Input File Usage:

```
*DAMAGE INITIATION, POSITION=[CENTROID | COMBINED | CRACKTIP |
NONLOCAL]
```

2.5.   Define Contact Interaction of Cracked Element Surfaces

Contact between opposing surfaces of a crack or between crack surfaces and other types of surfaces can be modeled in XFEM. Two contact formulations are available: a small sliding formulation and a finite sliding formulation with general contact.

2.5.1. Small-sliding Formulation

The small-sliding formulation can only be used to model contact between the crack surfaces, and cannot be used to model contact between the crack surfaces and other surfaces.

Input File Usage:

```
*ENRICHMENT, INTERACTION=CONTACT
*SURFACE INTERACTION, NAME=CONTACT
*SURFACE BEHAVIOR
```

2.5.2. Finite-sliding Formulation with General Contact

The general contact capability in XFEM can not only be used to model contact between crack surfaces, but can also be used to model contact between crack surfaces and other

types of surfaces. General contact always takes precedence over the small-sliding contact formulation when both are included in a model.

The XFEM-based crack surfaces must be defined if they are to be included in the general contact domain.

Input File Usage:

```
*ENRICHMENT, NAME=region-1, TYPE=PROPAGATION CRACK, ELSET=SET-1
*SURFACE, TYPE=XFEM, NAME=CRACK_SURF
  region-1
```

If the XFEM-based crack surfaces are defined explicitly as described above, they will be included in the general contact domain automatically when the default all-inclusive exterior surface defines the contact domain. Alternatively, you can specify pairwise inclusions and exclusions by referencing the XFEM-based crack surfaces in the data lines of *CONTACT INCLUSION and *CONTACT EXCLUSION.

Input File Usage, all-inclusive surface example:

```
*SURFACE INTERACTION, NAME=INTERACTION_PROPERTY_NAME
*CONTACT
*CONTACT INCLUSIONS, ALL EXTERIOR
*CONTACT PROPERTY ASSIGNMENT
  , , INTERACTION_PROPERTY_NAME
```

Input File Usage, specific surface pairings example:

```
*SURFACE INTERACTION, NAME=INTERACTION_PROPERTY_NAME
*CONTACT
*CONTACT INCLUSIONS
  ,CRACK_SURF


*CONTACT PROPERTY ASSIGNMENT
  , , INTERACTION_PROPERTY_NAME
```

In this specification, the crack surface (named **CRACK_SURF**) is defined to interact with the all-inclusive exterior surface on the data line of *CONTACT INCLUSIONS.

2.6. Procedures

The following analysis procedures are supported for modeling crack propagation with XFEM:

- Static analysis
- Implicit dynamic analysis
- Low-cycle fatigue analysis using the direct cyclic approach
- Geostatic stress field analysis
- Coupled pore fluid diffusion/stress analysis

2.7. Prescribed Conditions

2.7.1. Initial Conditions

A pre-existing crack is specified by using *INITIAL CONDITIONS, TYPE=ENRICHMENT (Section 2.2).

2.7.2. Boundary Conditions

Boundary conditions can be applied to any of the displacement or pore pressure degrees of freedom. Boundary conditions at phantom nodes can be prescribed for enriched elements.

2.7.3. Loads

The following types of loading can be prescribed in a model with an enriched feature:

- Concentrated nodal forces can be applied to the displacement degrees of freedom (1–3) or the pore pressure degree of freedom (8);
- Distributed pressure forces (applied to the cracked element surface) or body forces can be applied.

Crack propagation may be unstable under load control analysis. In such cases, using displacement control loading may provide a solution.

2.7.4. Predefined Fields

The following predefined fields can be specified in a model with an enriched feature:

- Nodal temperatures (of field variable type; temperature is not a degree of freedom in stress/displacement elements). The specified temperature affects temperature-dependent critical stress and strain failure criteria.
- The value of user-defined field variables. The specified value affects field-variable-dependent material properties.

2.8.   Output Requests

The nodal output variable PHILSM (signed distance function to describe the crack surface) must be requested in order to visualize cracks in Abaqus/Viewer.

The following XFEM related output variables are useful to determine the damage status of an enriched element.

- STATUSXFEM

This is a whole element output variable that indicates the status of the enriched elements. The value of STATUSXFEM is 0.0 if the enriched element contains no crack and 1.0 if the enriched element is completely cracked with no traction across the crack faces. The STATUSXFEM value is 0.4 if the crack nucleates in the enriched element. After the crack nucleates, STATUSXFEM has a value between 0.4 and 1.0 until the enriched element is completely cracked.

- CRKDISP, CSDMG, and CRKSTRESS

These are surface output variables for the cracked surface, and are available only for propagating cracks modeled with first-order solid continuum elements. The output variable CRKSTRESS is the remaining residual stress on cracked surfaces. An enriched element is considered completely cracked only when no residual stresses remain on the cracked surfaces in the enriched element (CRKSTRESS=0). The output variable CSDMG gives the damage status on cracked surfaces in enriched elements. When an element is cracked, the remaining residual stresses on the cracked surface are ramped down. The value of CSDMG is 0.0 when the element has no crack and 1.0 when it is completely cracked. The output variable CRKDISP gives crack opening and relative tangential motions on the cracked surfaces. If the value of CRKSTRESS drops to 0, the value of STATUSXFEM and CSDMG will reach 1.0.

To request CRKSTRESS, CSDMG, and CRKDISP on the cracked surfaces, define an XFEM surface associated with an enrichment region and request the output on the *CONTACT OUTPUT keyword option.

Input File Usage:

```
*ENRICHMENT, NAME=XFEM_SET
*SURFACE, TYPE=XFEM, NAME=XFEM_SURFACE
 XFEM_SET
 …
*OUTPUT, FIELD
*CONTACT OUTPUT, SURFACE=XFEM_SURFACE
 CRKSTRESS, CSDMG, CRKDISP
```

Note: The energy release rate, ENRRTXFEM, is available as element output when the LEFM-based method is used.

2.9. Analysis Settings to Aid Convergence

An XFEM crack propagation analysis can sometimes fail to converge, in spite of reasonable damage properties and a suitably refined mesh in the enriched region. Several analysis settings can be modified to facilitate convergence.

2.9.1. Time Incrementation and Solution Controls

The following time incrementation and solution controls can be modified to improve the convergence behavior.

- Set reasonable minimum and maximum time increment sizes for the step.
- Increase the number of increments for the step from the default value of 100.
- Increase the number of equilibrium iterations (without severe discontinuities) after which the check is made whether the residuals are increasing in two consecutive iterations (default is 4).
- Increase the number of consecutive equilibrium iterations (without severe discontinuities) at which logarithmic rate of convergence check begins (default is 8).
- Increase value of maximum number of attempts (cutbacks) allowed for an increment (default is 5).

Input File Usage:

```
*STEP, NLGEOM=YES
*STATIC, INC=10000
 0.01, 1.0, 1.0E-9, 0.01
*CONTROLS, ANALYSIS=DISCONTINUOUS
*CONTROLS, PARAMETERS=TIME INCREMENTATION
 ,,,,,,,20
```

In this specification, the *CONTROLS, ANALYSIS=DISCONTINUOUS keyword option permits 8 equilibrium iterations (without severe discontinuities) before checking whether the residuals are increasing in two consecutive iterations and 10 before using a logarithmic rate of convergence check. The *CONTROLS, PARAMETERS=TIME INCREMENTATION specification allows 20 attempts for an increment.

2.9.2. Unsymmetric Matrix Storage and Solution Scheme

The stiffness matrix for enriched elements is unsymmetric; therefore, unsymmetric matrix storage and solution scheme is recommended for XFEM analysis to improve convergence behavior.

2.9.3. Damage Initiation Tolerance for the Cohesive Segments Method

There can be convergence difficulties resulting from the initiation of damage. For instance, if in a particular increment the calculated stress/strain in one of the enriched elements is significantly greater than the critical value you specified, Abaqus/Standard will reduce the increment size. Since it is typically not possible to reach the exact initiation stress/strain in an iterative procedure, we prescribe a damage tolerance f_{tol} so that damage is initiated if the calculated value of the stress or strain is greater than the critical value, but does not exceed it by the tolerance we specify. If $f = 1$ represents exact satisfaction of the damage initiation criteria, including a tolerance can be expressed as:

$$1.0 \leq f \leq 1.0 + f_{tol}$$

A larger value for f_{tol} allows for a quick solution but with lower accuracy. At the same time, too small a tolerance will require more iterations or cutbacks and may lead to poor convergence rates. The default tolerance is set at 0.05 (5%). If you encounter convergence difficulties, increasing the tolerance by a small amount can help.

To improve performance, a separate tolerance can be specified to control the growth of an existing crack. Typically, a tight tolerance should be used for the crack nucleation so that the crack will only nucleate at the true hot spots, and the growth tolerance can be slightly higher than the tolerance to improve performance.

Input File Usage:

```
*DAMAGE INITIATION, TOLERANCE=f_tol, GROWTH TOLERANCE=f_tol^g
```

For an unstable crack growth problem, an unstable growth tolerance may be used to improve performance of further extension of an existing crack. When f is much larger than $1.0 + f_{tol}$, excessive cut backs during iteration will occur. To improve performance, an unstable growth tolerance is introduced. After the initial crack nucleates, if $1.0 \leq f \leq 1.0 + f_{tol}$, instead of cutting back the increment size, the time increment is reduced automatically to a very small value, allowing more elements to fracture until $f < 1.0$ for all the elements ahead of the crack tip. After this, the time increment will automatically recover to a larger value. If no value is specified for the unstable growth tolerance, the default value is infinity.

Input File Usage:

```
*DAMAGE INITIATION, UNSTABLE GROWTH TOLERANCE=
*CONTROLS, TYPE=NO CUTBACK SCALING
```

$$*DAMAGE\ INITIATION,\ UNSTABLE\ GROWTH\ TOLERANCE = f_{tol}^{g}$$

2.9.4. Viscous Regularization

Crack propagation problems require progressive damage, which includes softening and loss of stiffness. This can cause severe convergence difficulties. Viscous regularization can be added to ensure that the tangent stiffness remains positive definite thereby improving the convergence rates. Viscous regularization is specified on the *DAMAGE STABILIZATION keyword option for the cohesive segments method and is specified by including the VISCOSITY parameter on the *FRACTURE CRITERION keyword option for the LEFM-based method. A small value regularizes the analysis, helping with convergence while having a minimal effect on the response.

It is important to note that excessive stabilization can negatively influence the solution and can even produce non-physical results. You must verify that this influence is within acceptable limits. The approximate amount of energy dissipated with damage stabilization over the whole model is available in output variable ALLVD and should be significantly smaller than the model's strain energy.

2.9.5. Static Stabilization

In many scenarios, predicting the load or displacement necessary for a crack to propagate completely through a specimen may not be possible. So if excessive load is prescribed, the structure may split into multiple regions and these regions may not be sufficiently constrained, leading to rigid body motion. The resulting rigid body modes will result in cutbacks and smaller time increments. Typically, the response beyond the point of complete failure may not be of interest. So to avoid unnecessary use of computational resources to obtain the response after complete failure, you can specify stabilization that will artificially damp the rigid body modes and facilitate faster convergence. The stabilization value can be specified using *STATIC, STABILIZE.

It is important to note that when stabilization is activated, it is acting on the whole model during the entire simulation; output variable ALLSD should be monitored to make sure that the damping is not negatively affecting the solution.

2.9.6. Enrichment Activation

After all existing cracks have completely propagated through the enrichment region, further crack initiation in the enriched region may lead to convergence problems. The

*ENRICHMENT ACTIVATION keyword option can be used to switch element enrichment during the step.

Input File Usage:

```
*ENRICHMENT ACTIVATION, ACTIVATE=AUTO OFF
```

This deactivates the enriched feature automatically once all the existing cracks have propagated through the enrichment region within the step.

3. Examples

Two sample input files are provided. The example is based on "*Crack propagation of a single-edge notch simulated using XFEM*" in the Abaqus Benchmark Guide. The sample input files illustrate the use of XFEM in Abaqus/Standard to predict crack initiation and propagation of a single-edge notch in a specimen using both the cohesive segments method and the LEFM method.

4. References

1. *Abaqus Analysis User's Guide*
2. *Abaqus Keywords Reference Guide*
3. *Lecture notes on Modeling Fracture and Failure with Aabqus*
4. QA0000000008202: Facilitating XFEM analysis convergence
5. QA00000042372: Status of Abaqus enriched elements
6. QA00000008204: Comparison of crack propagation and failure modeling techniques in Abaqus
7. QA00000046393: Contour integral evaluation using the Abaqus/Standard extended finite element method (XFEM)

5. Document History

Document Revision	Date	Revised By	Changes/Notes
V1.0	**4/16/2018**	**Ying CHEN, David PALMER**	**Original**

Our **3D**EXPERIENCE® platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes, the **3D**EXPERIENCE® Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 210,000 customers of all sizes in all industries in more than 140 countries. For more information, visit **www.3ds.com**.

3DEXPERIENCE®