eXtended Finite Element Method (XFEM) in Abaqus

Zhen-zhong Du

SIMULIA

Overview

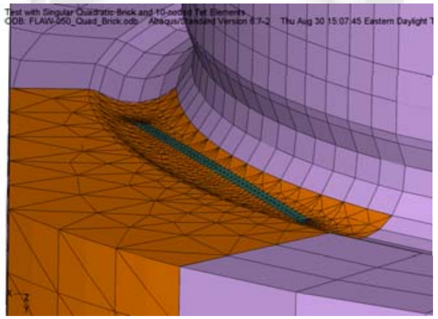
- **Introduction**
- **Basic XFEM Concepts**
- **Modeling Approaches**
  - **Stationary cracks**
    - **Contour integral calculation**
  - **Propagation cracks**
    - **Cohesive segments approach**
    - **Linear elastic fracture mechanics approach**
- **XFEM simultaneously used with other Fracture and Failure Techniques**
  - Bulk material failure and interfacial delamination
- **Analysis Procedures**
  - **Static**
  - **Implicit dynamic**
  - **Low cycle fatigue**
- **XFEM used with other Analysis Techniques**
  - **Global/local modeling approach**
  - **Co-Simulation**
- **Elements, Outputs and others**
- **Demonstration**

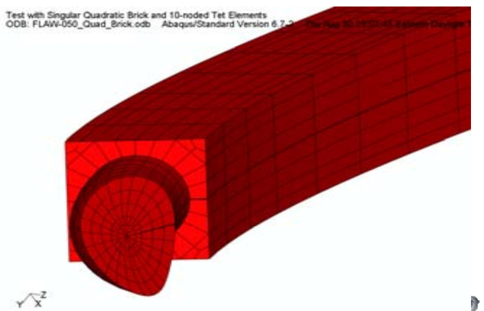
SIMULIA

Introduction

Introduction

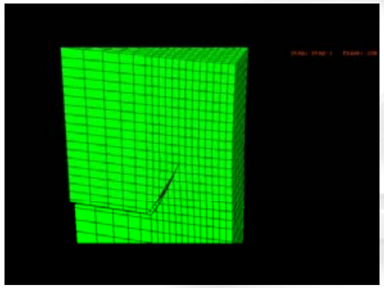
- **Strong technology exists in Abaqus:**
  - Interfacial cracks with VCCT and cohesive element techniques
  - Smeared crack approach to continuum damage initiation and evolution in the bulk materials
- **Some difficulties exist:**
  - Modeling and analysis of stationary 3-D curved surface cracks
  - Progressive crack growth simulations for arbitrary 3-D cracks
- **eXtended Finite Element Method (XFEM) becomes relatively mature to be commercialized since it was 1st introduced by Belyschko and Black in 1999.**

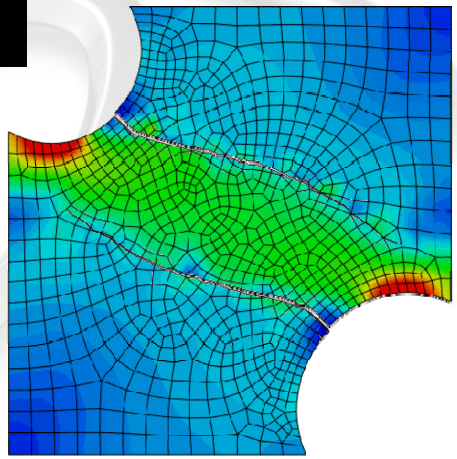


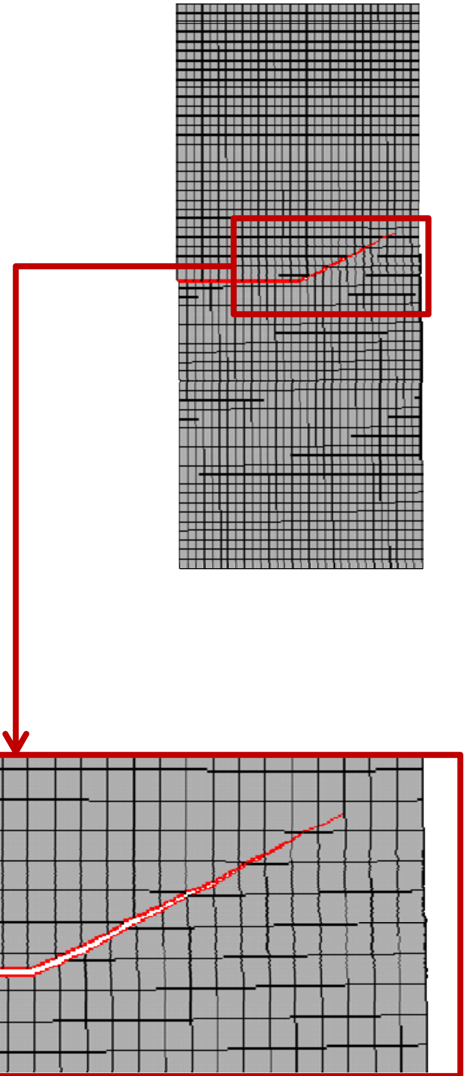


Introduction

- **Makes modeling of cracks easier and accurate**
  - Allows crack to be modeled independent of the mesh
  - Allows simulation of initiation and propagation of a discrete crack along an arbitrary, solution-dependent path without the requirement of remeshing
  - Supports contour integral evaluation for a stationary







Basic XFEM Concepts

SIMULIA

Basic XFEM Concepts

- is an extension of the conventional finite element method based on the concept of partition of unity;
- allows the presence of discontinuities in an element by enriching degrees of freedom with special displacement functions

Nodal displacement vectors

Displacement vector

Jump function

Nodal enriched degree of freedom vector

$$\mathbf{u} = \sum_{I=1}^{N} N_I(x)\left[\mathbf{u}_I + H(x)\mathbf{a}_I + \sum_{\alpha=1}^{4} F_\alpha(x)\mathbf{b}_I^\alpha\right]$$

Shape functions

Nodal enriched degree of freedom vector

Asymptotic crack-tip functions

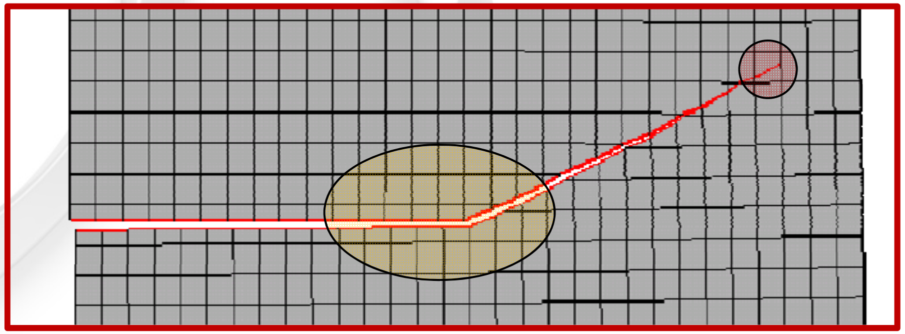
Basic XFEM Concepts

Applies to all nodes in the model

Applies to nodes whose shape function support is cut by the crack tip

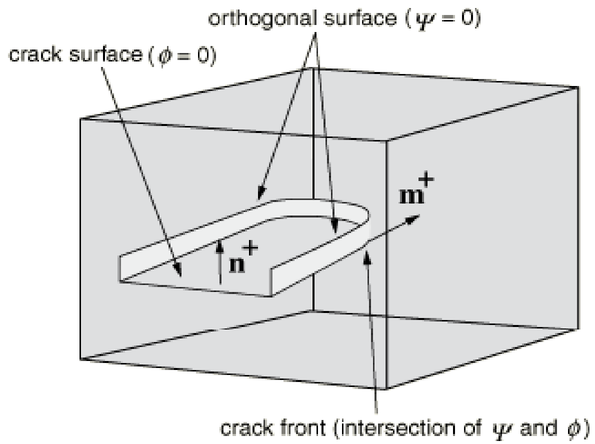
$$\mathbf{u} = \sum_{I=1}^{N} N_I(x)\left[\mathbf{u}_I + H(x)\mathbf{a}_I + \sum_{\alpha=1}^{4} F_\alpha(x)\mathbf{b}_I^\alpha\right]$$

Applies to nodes whose shape function support is cut by the crack interior



Basic XFEM Concepts
Level set method

- Is a numerical technique for describing a crack and tracking the motion of the crack
- Couples naturally with XFEM and makes possible the modeling of 3D arbitrary crack growth without remeshing
- Requires two level sets for a crack:
  - The first describes the crack surface, Φ (phi)
  - The second, Ψ (psi), is constructed so that the intersection of two level sets gives the crack front

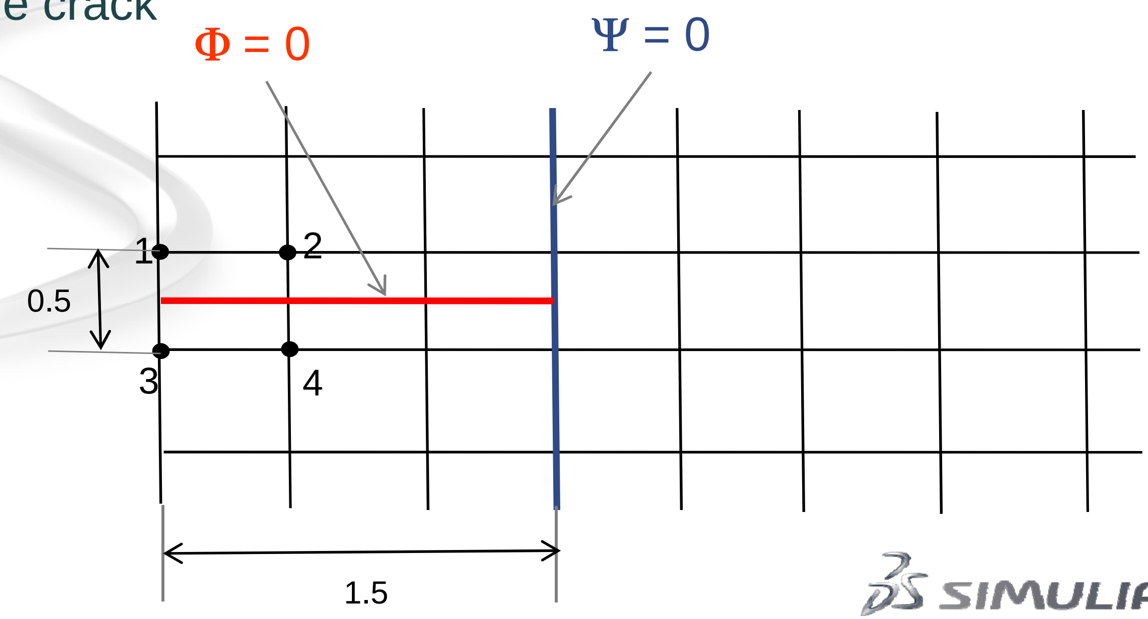


- Uses signed distance functions to describe the crack geometry
- No explicit representation of the crack is needed and the crack is entirely described by nodal data

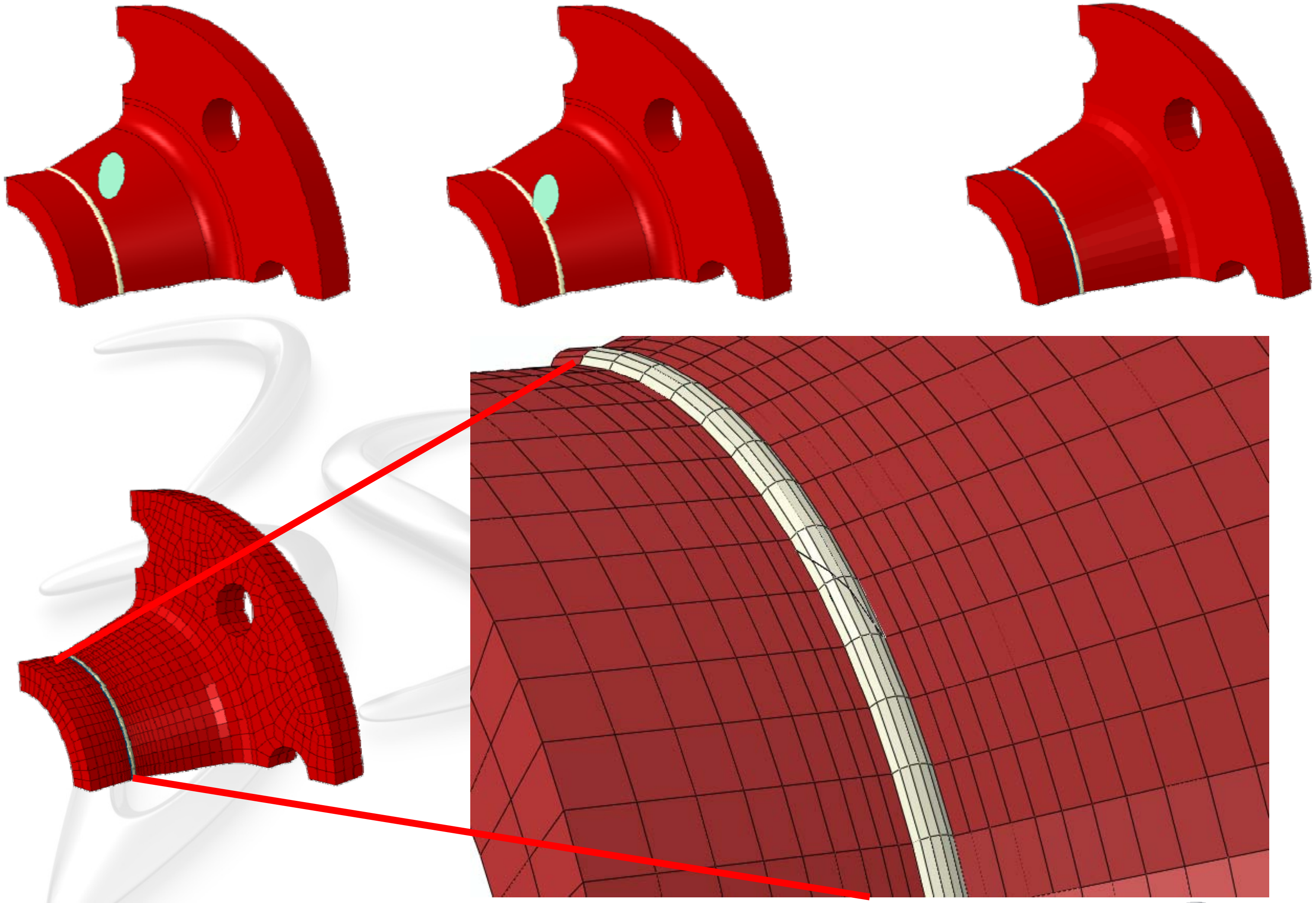
Basic XFEM Concepts

- **Calculating Φ and Ψ**
  - The nodal value of the function Φ is the *signed* distance of the node from the crack face
    - Positive value on one side of the crack face, negative on the other
  - The nodal value of the function Ψ is the *signed* distance of the node from an almost-orthogonal surface passing through the crack front
    - The function Ψ has zero value on this surface and is negative on the side towards the crack

Node	Φ	Ψ
1	+0.25	–1.5
2	+0.25	–1.0
3	–0.25	–1.5
4	–0.25	–1.0



Basic XFEM Concepts



Modeling approaches

Modeling stationary cracks

SIMULIA

Stationary Cracks

- **Full enrichment is used**
- **Different forms of asymptotic crack-tip functions are needed depending on crack location and the extent of the inelastic material deformation**
  - Currently only asymptotic crack-tip fields corresponding to an isotropic elastic material are considered
  - Can be extended

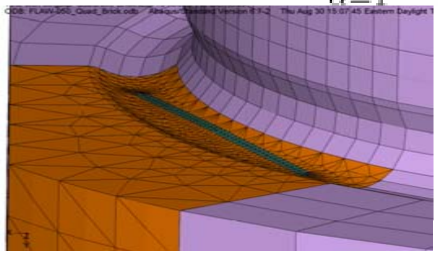
$$\mathbf{u} = \sum_{I=1}^{N} N_I(x)[\mathbf{u}_I + H(x)\mathbf{a}_I + \sum_{\alpha=1}^{4} F_\alpha(x)\mathbf{b}_I^\alpha]$$

For isotropic elasticity:

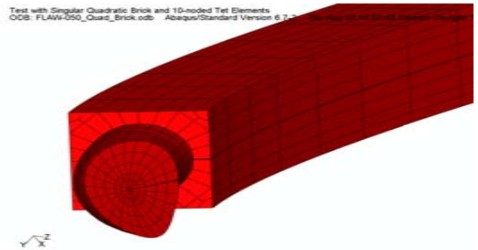
$$[F_\alpha(x), \alpha = 1\text{-}4] = [\sqrt{r}\sin\frac{\theta}{2}, \sqrt{r}\cos\frac{\theta}{2}, \sqrt{r}\sin\theta\sin\frac{\theta}{2}, \sqrt{r}\sin\theta\cos\frac{\theta}{2}]$$

Stationary Cracks

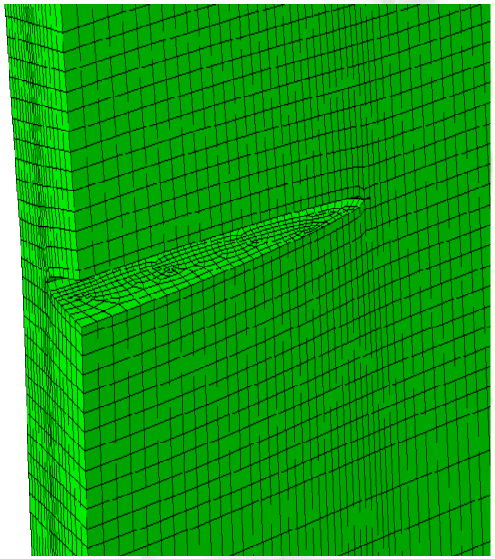
$$\mathbf{u} = \sum_{I=1}^{N} N_I(x)\left[\mathbf{u}_I + H(x)\mathbf{a}_I + \sum_{\alpha=1}^{4} F_\alpha(x)\mathbf{b}_I^\alpha\right]$$

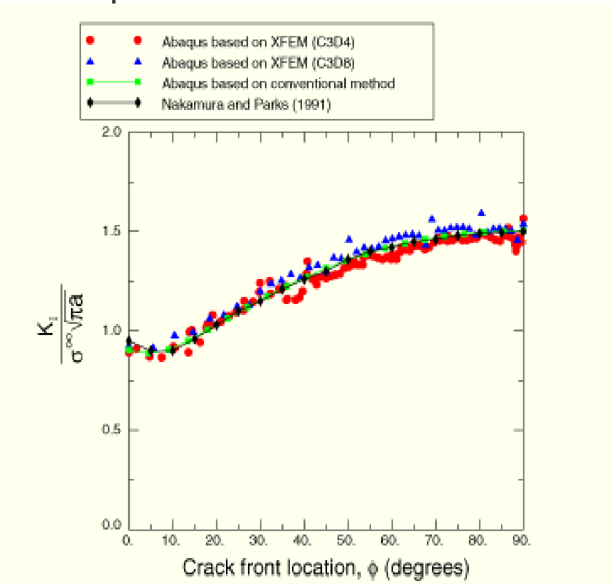


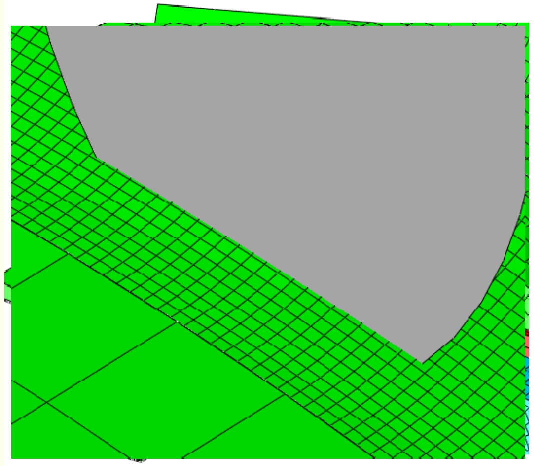
- Support contour integral evaluations for an arbitrary stationary surface crack without the need to conform the mesh to the geometry of the discontinuities.
- Support only 1st order brick and 1st and 2nd order tetrahedron elements with isotropic elastic materials and small deformation in a stationary crack.
  - Semi-elliptical crack in a plate



Contour integral mesh with Conventional method







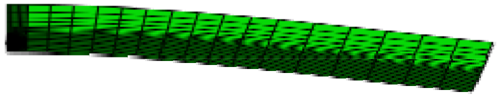
Contour integral mesh with XFEM

Contour integral with residual stress field

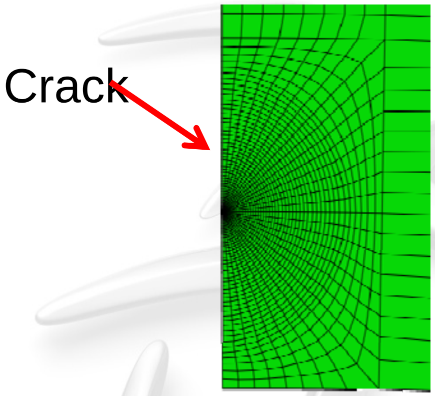
- Functionality
  - The residual stress field can now be taken into account based on either the conventional finite element method or XFEM
- Use cases/drivers
  - A residual stress field can be resulted from service loads that produce plasticity, a metal forming process in the absence of an anneal treatment, thermal effects, or swelling effects.
  - The standard definition of the contour integral may lead to a path-dependent value when the residual stresses are significant.
  - An additional term due to the residual stress field is now included to ensure the path independence of contour integral.
- Usage
  - Input File Usage: *CONTOUR INTEGRAL, RESIDUAL STRESS STEP=*n*, TYPE=J
    - The user can take into account the final stress from any previous step by using the STEP parameter.
  - STEP=0 means initial stresses defined on *initial conditions, type=stress are used.
  - With XFEM only STEP=0 is currently allowed
- Theory

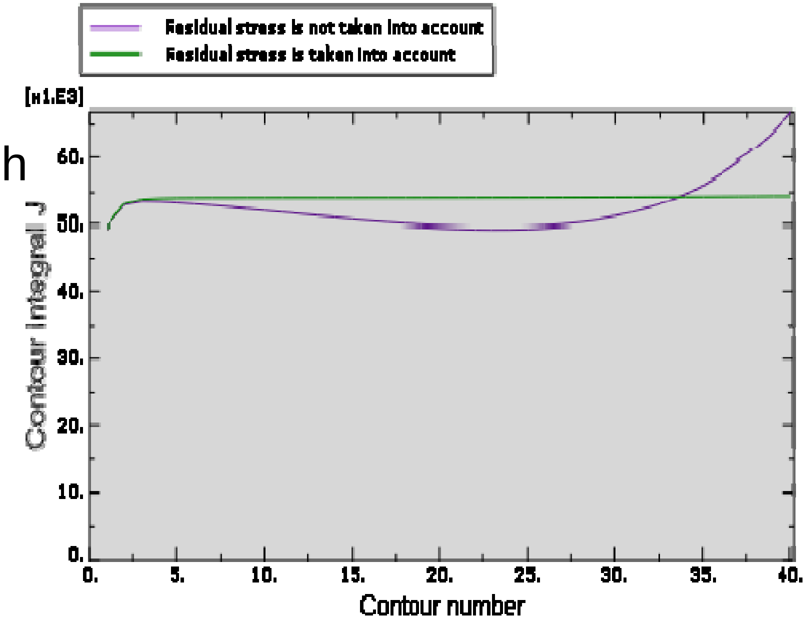
$$\bar{J} = \int_A \lambda(s)\mathrm{n} \bullet (w\mathrm{I} - \sigma \bullet \frac{\partial \mathrm{u}}{\partial \mathrm{x}}) \bullet \mathrm{q}d\mathrm{A} + \int_V \sigma : \frac{\partial \varepsilon^o}{\partial \mathrm{x}} \bullet \mathrm{q}dV$$

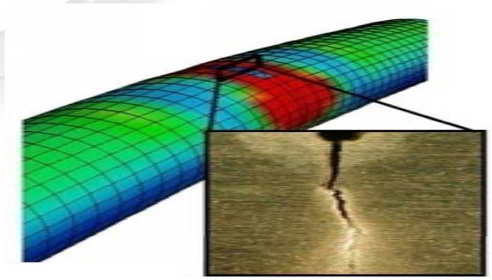
Contour integral with residual stress



Half geometry of a single edge notch under three point bending







SIMULIA

Modeling propagation cracks

SIMULIA

Propagation cracks

$$\mathbf{u} = \sum_{I=1}^{N} N_I(x)[\mathbf{u}_I + H(x)\mathbf{a}_I + \sum_{\alpha=1}^{4} \cancel{F_\alpha(x)}\mathbf{b}_I^\alpha]$$

- **Assumptions**
  - Near-tip asymptotic singularity is not considered
  - Crack has to propagate across an entire element at a time to avoid the need to model the stress singularity
  - Effective engineering approach
- **Two distinct types of damage modeling within an XFEM framework**
  - Cohesive segments approach
  - Linear elastic fracture mechanics (LEFM) approach
- **Cohesive segment approach**
  - Uses traction-separation laws
  - Follows the general framework for surface based cohesive behavior
  - Damage properties are specified as part of the bulk material definition
- **LEFM-based approach**
  - Uses the virtual crack closure technique (VCCT)
  - VCCT for XFEM uses the same principles as in VCCT for interfacial debonding
  - Damage properties are specified via an interaction property assigned to the XFEM crack

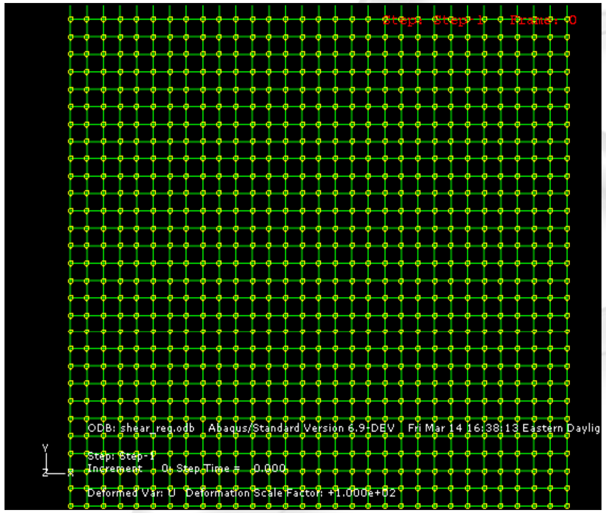
Cohesive segments approach

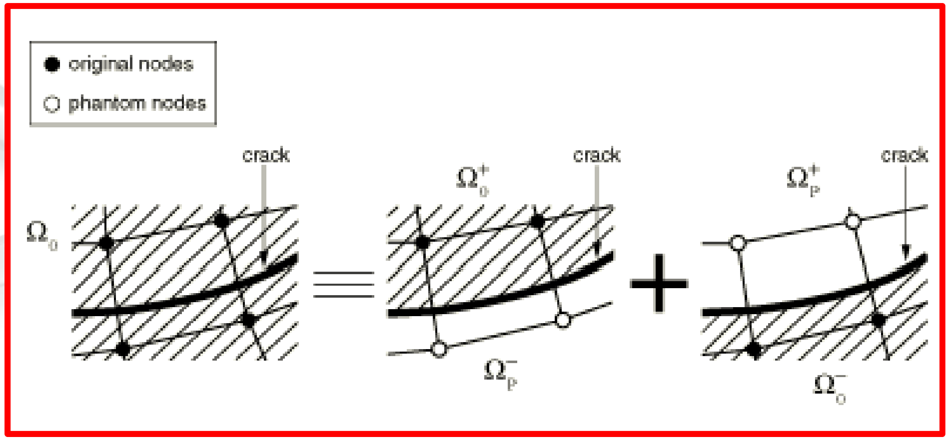
SIMULIA

Propagation Cracks

$$\mathbf{u} = \sum_{I=1}^{N} N_I(x)[\mathbf{u}_I + H(x)\mathbf{a}_I + \sum_{\alpha=1}^{4} F_\alpha \cancel{(x)} \mathbf{b}_I^\alpha]$$

- **"Phantom nodes" and cohesive segments**
  - Can be used for brittle or ductile fracture
  - Pressure-overclosure relationship governs the behavior when the crack is "closed"
  - Cohesive behavior contributes to the contact normal stress when the crack is "open"





Cohesive segments approach

- **Cohesive Damage Initiation Criteria**
  - Three stress-based and three strain-based damage initiation criteria are readily available
    - Maximum principal stress (MAXPS) and maximum principal strain (MAXPE)
    - Maximum nominal stress (MAXS) and maximum nominal strain (MAXE)
    - Quadratic nominal stress (QUADS) and quadratic nominal strain (QUADE)
  - In addition, a user-defined damage initiation criterion can be specified in user subroutine UDMGINI
  - Crack initiation bases on the stress/strain value at the center of enriched elements

Cohesive segments approach

- **Maximum principal stress (MAXPS) and maximum principal strain (MAXPE) criteria**
  - Initiation occurs when the maximum principal stress or strain reaches a critical value ($f = 1$)

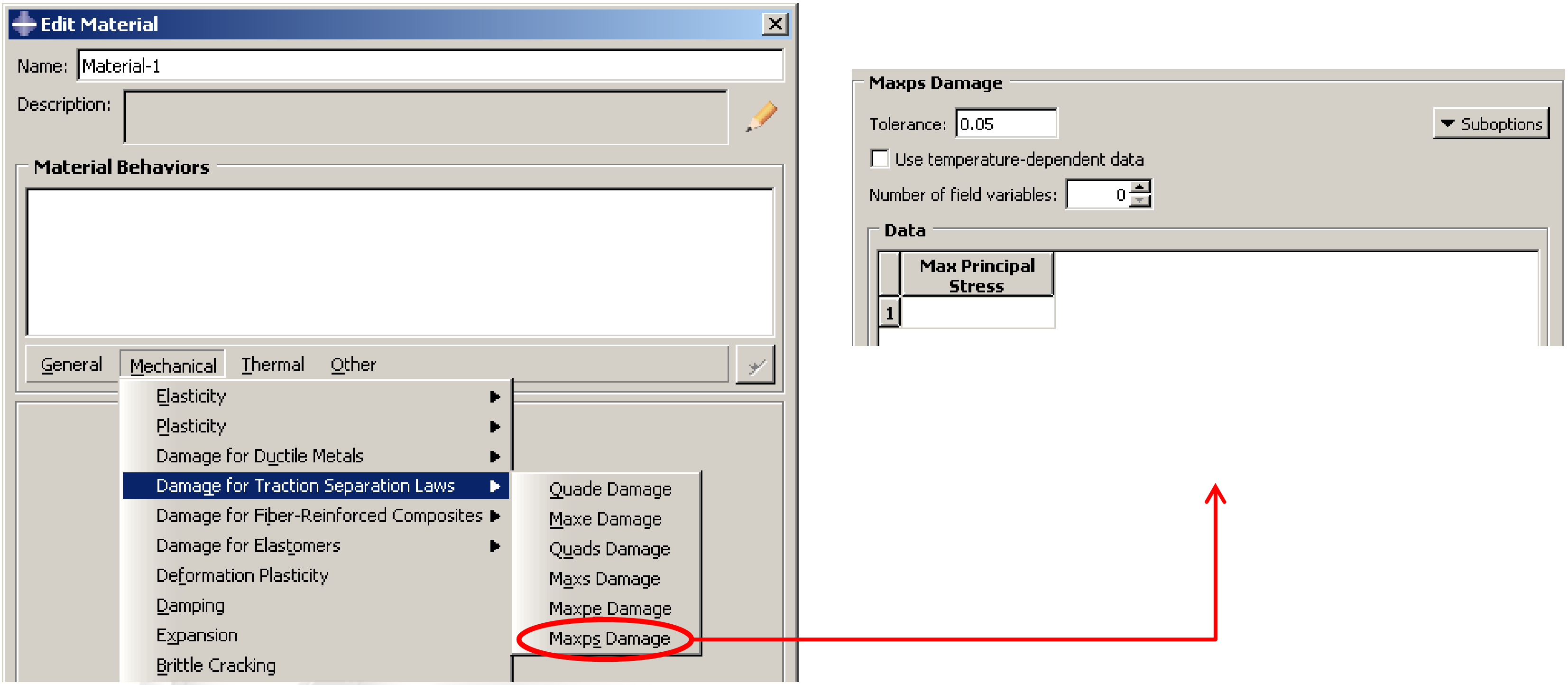
MAXPS $$f = \frac{\langle \sigma_n \rangle}{\sigma_{\max}^0}$$

MAXPE $$f = \frac{\langle \varepsilon_n \rangle}{\varepsilon_{\max}^0}$$

  - Crack plane is solution-dependent
    - Perpendicular to the direction of the maximum principal stress (or strain)
    - Can handle a changing crack plane and crack propagation direction

Cohesive segments approach

- **Maximum principal stress (MAXPS) and maximum principal strain (MAXPE) criteria (cont’d)**




```
* DAMAGE INITIATION, CRITERION = { MAXPS, MAXPE },
TOLERANCE = {value}
```

Cohesive segments approach

- **Maximum nominal stress (MAXS) and maximum nominal strain (MAXE) criteria**
  - Initiation occurs when the maximum nominal stress or strain reaches a critical value

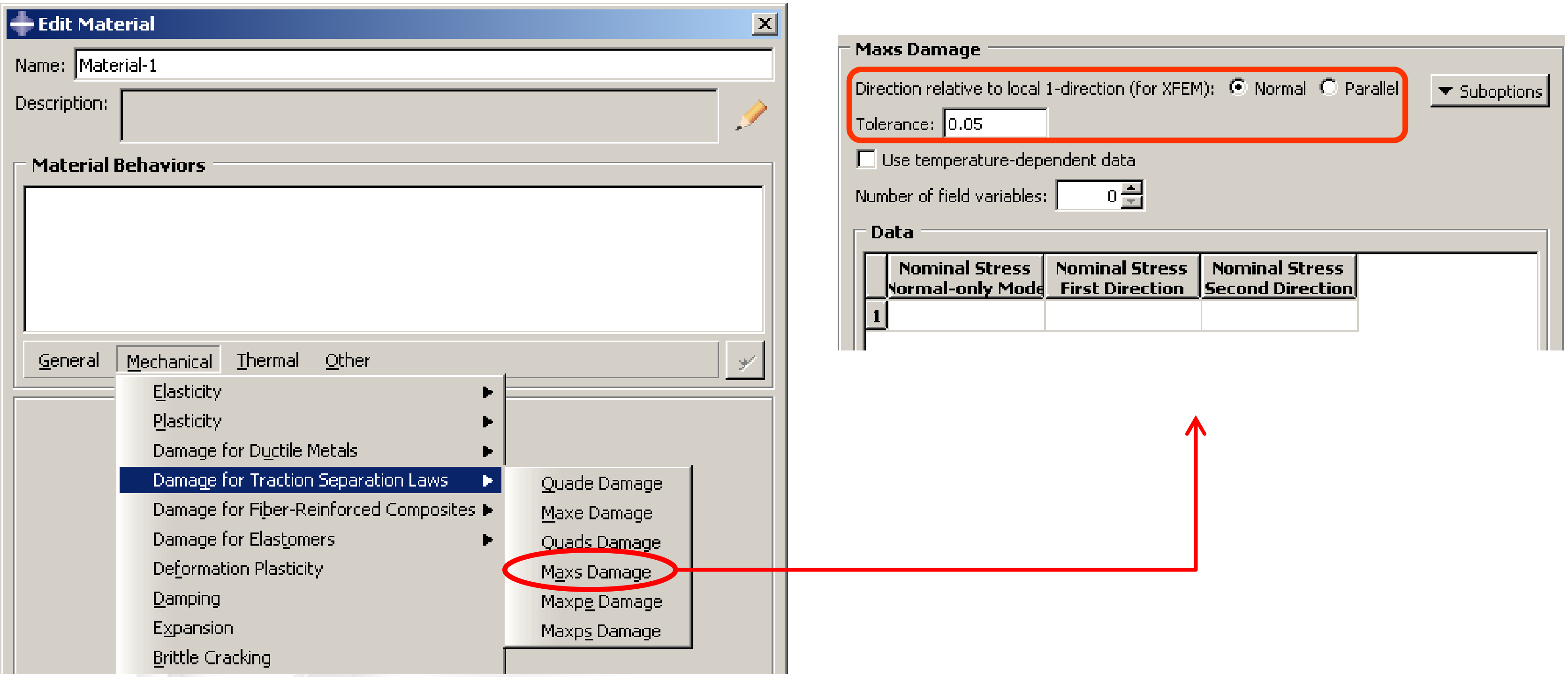
MAXS $$MAX\left\{\frac{\langle\sigma_n\rangle}{N_{\max}},\frac{\sigma_t}{T_{\max}},\frac{\sigma_s}{S_{\max}}\right\}=f \qquad \langle\sigma_n\rangle=\begin{cases}\sigma_n & \text{for } \sigma_n>0\\ 0 & \text{for } \sigma_n<0\end{cases}$$

MAXE $$MAX\left\{\frac{\langle\varepsilon_n\rangle}{\varepsilon_n^{\max}},\frac{\varepsilon_t}{\varepsilon_t^{\max}},\frac{\varepsilon_s}{\varepsilon_s^{\max}}\right\}=f \qquad \langle\varepsilon_n\rangle=\begin{cases}\varepsilon_n & \text{for } \varepsilon_n>0\\ 0 & \text{for } \varepsilon_n<0\end{cases}$$

  - The damage initiation criterion is satisfied when $1.0 \le f \le 1.0 + f_{tol}$ where f_{tol} is a user-specified tolerance value (default is 0.05)
  - Similar to the criterion used in conjunction with element-based cohesive behavior
  - User may specify a local material direction as the crack plane normal

Cohesive segments approach

- **Maximum nominal stress (MAXS) and maximum nominal strain (MAXE) criteria (cont'd)**




```
*DAMAGE INITIATION, CRITERION = { MAXS | MAXE },
NORMAL DIRECTION = {1 (default)| 2}, TOLERANCE = {0.05 (default)}
```

Cohesive segments approach

- **Quadratic nominal stress (QUADS) and quadratic nominal strain (QUADE)**

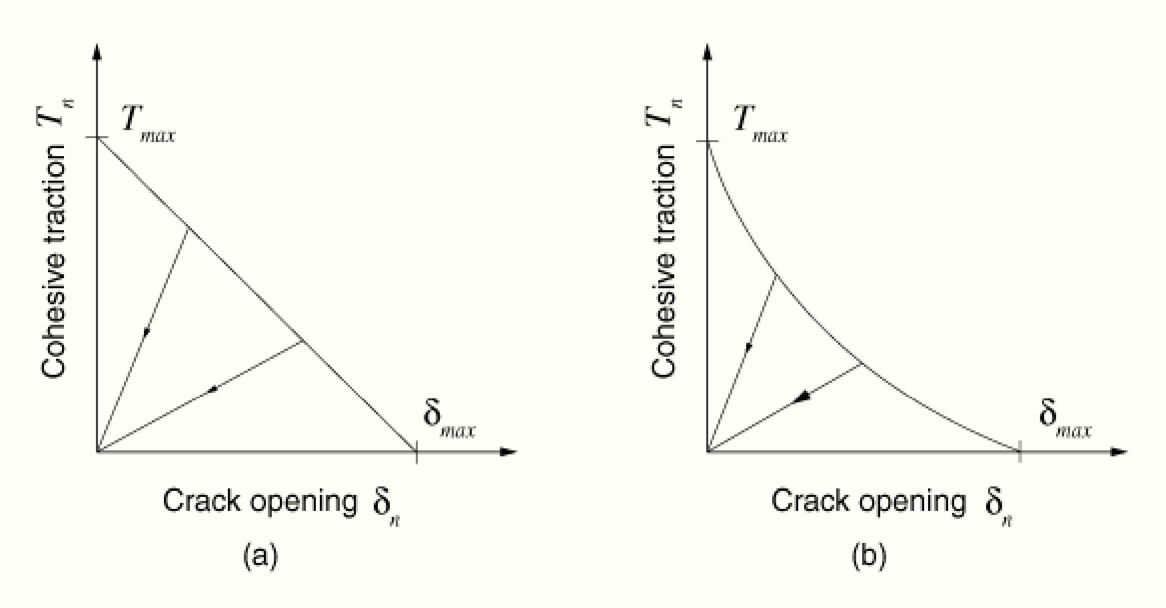
$$\left(\frac{\langle \sigma_n \rangle}{N_{\max}}\right)^2 + \left(\frac{\sigma_t}{T_{\max}}\right)^2 + \left(\frac{\sigma_s}{S_{\max}}\right)^2 = 1 \qquad \left(\frac{\langle \varepsilon_n \rangle}{\varepsilon_n^{\max}}\right)^2 + \left(\frac{\varepsilon_s}{\varepsilon_s^{\max}}\right)^2 + \left(\frac{\varepsilon_t}{\varepsilon_t^{\max}}\right)^2 = 1$$

  - Similarities with MAXS and MAXE
    - User selects the crack plane normal
    - User specifies critical values of normal and shear stresses (strains)
    - User interface in Abaqus/CAE similar to that of MAXS/MAXE

```
*DAMAGE INITIATION, CRITERION = { QUADS | QUADE },
NORMAL DIRECTION = {1 (default)| 2},
TOLERANCE = {0.05 (default)}
```

Cohesive segments approach

- **Damage evolution**
  - Any of the damage evolution models for traction-separation laws can be used: based on energy or displacement
  - However, it is not necessary to specify the undamaged traction-separation response



Cohesive segments approach

- **Damage stabilization**
  - Fracture makes the structural response nonlinear and non-smooth
    - Numerical methods have difficulty converging to a solution
  - Use viscous regularization helps with the convergence of the Newton method
  - The stabilization value must be chosen so that the problem definition does not change
    - A small value regularizes the analysis, helping with convergence while having a minimal effect on the response
    - Perform a parametric study to choose appropriate value for a class of problems

Cohesive segments approach

- **User defined damage initiation subroutine UDMGINI**

  ✓ Can be used to specify a user-defined damage initiation criterion.

  ✓ Allows the specification of more than one failure mechanisms in an element, with the most severe one governing the actual failure.

  ✓ Can be used in combination with several Abaqus built-in damage evolution models, with each model corresponding to a particular failure mechanism.

  ✓ Currently is only supported within the context of XFEM.

Keyword User interface when using UDMGINI

***ELEMENT, TYPE=C3D8, ELSET=ENRICHED ...**

***SOLID SECTION, MATERIAL=STEEL, ELSET=ENRICHED**

***ENRICHMENT, TYPE=PROPAGATION CRACK, ELSET=ENRICHED, NAME=ENRICHMENT**

***MATERIAL, NAME=STEEL**

***DAMAGE INITIATION, CRITERION=USER, PROPERTIES=NCONST, FAILURE MECHANISMS = NFAIL**

***DAMAGE EVOLUTION, FAILURE INDEX = 1**

…

***DAMAGE EVOLUTION, FAILURE INDEX = NFAIL**

User Subroutine Interface for UDMGINI

SUBROUTINE UDMGINI(FINDEX, NFINDEX, FNORMAL, NDI, NSHR, NTENS,PROPS, NPROPS, STATEV, NSTATEV, STRESS, STRAIN, STRAINEE, LXFEM, TIME, DTIME, TEMP, DTEMP, PREDEF, DPRED, NFIELD, COORDS, NOEL, NPT, LAYER, KSPT, KSTEP, KINC, KDIRCYC,KCYCLELCF, TIMECYC, SSE, SPD, SCD, SVD, SMD, JMAC, JMATYP, MATLAYO, LACCFLA, CELENT, DROT, ORI)

Variables to be defined

FINDEX(NFINDEX)

The vector defines the indices for all the failure mechanisms.

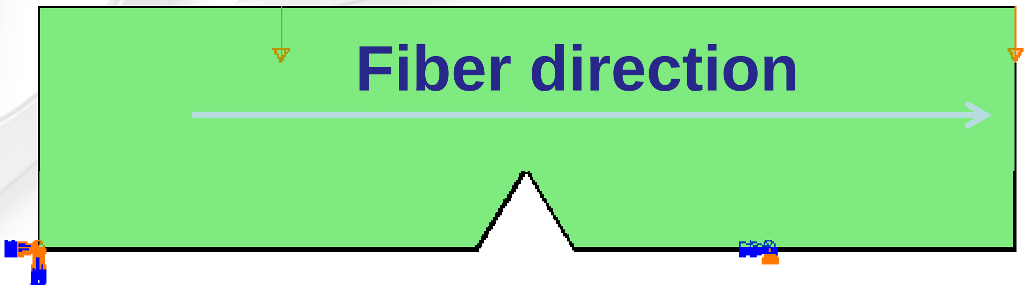
FNORMAL(NDI, NFINDEX)

The array defines the normal direction to the fracture plane (3D) or line (2D) for each failure mechanism.

Example of using UDMGINI

Mode II Fracture of Cortical Bone

- **Objective**
  - Demonstrate how composite fracture criteria could be applied to predict cortical bone fracture using XFEM and user defined damage initiation criteria
- **FEA model**
  - 3D model of a notched bone under asymmetric four point bending
  - Assuming fiber runs along the axis of the specimen
- **XFEM**
  - Two damage initiation criteria analyzed
    - Built-in maximum principal stress criterion
    - Composite fracture criterion with two competing failure mechanisms



Example of using UDMGINI

Mode II Fracture of Cortical Bone

- **Fiber failure mechanism**
  - The fiber direction is based on the material orientation
  - Fiber failure stresses:

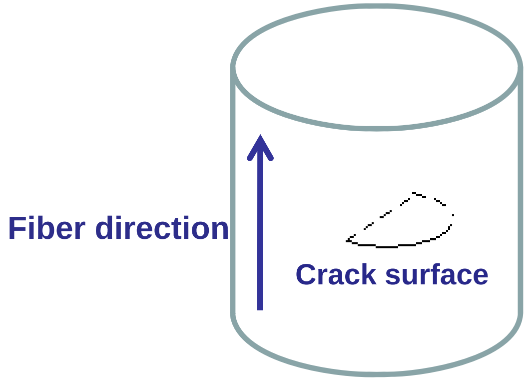
    $\sigma_f = 133$ MPa

    $\tau_f = 68$ MPa

  - Fiber failure criterion

$$\bar{\sigma} = \sqrt[2]{\frac{\sigma_{11}^{\;2}}{\sigma_f^{\;2}} + \frac{\tau_{12}^2}{\tau_f^{\;2}} + \frac{\tau_{13}^2}{\tau_f^{\;2}}} \geq 1$$

  - Crack surface is always perpendicular to the fiber direction



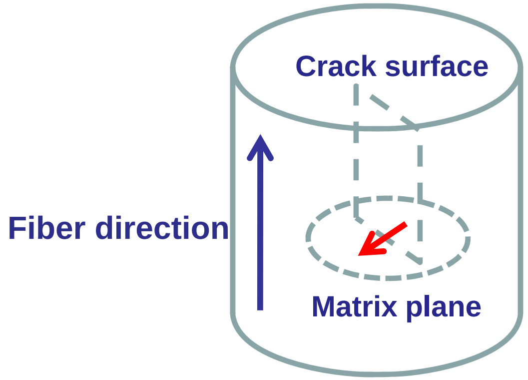
Example of using UDMGINI

Mode II Fracture of Cortical Bone

- **Matrix failure mechanism**
  - Matrix plane: any plane perpendicular to the fiber direction
  - The in-plane maximum principal stress and its direction within the matrix plane is used to determine the crack initiation and propagation direction
  - Matrix failure stress:

$$\sigma_f = 51 \text{ MPa}$$

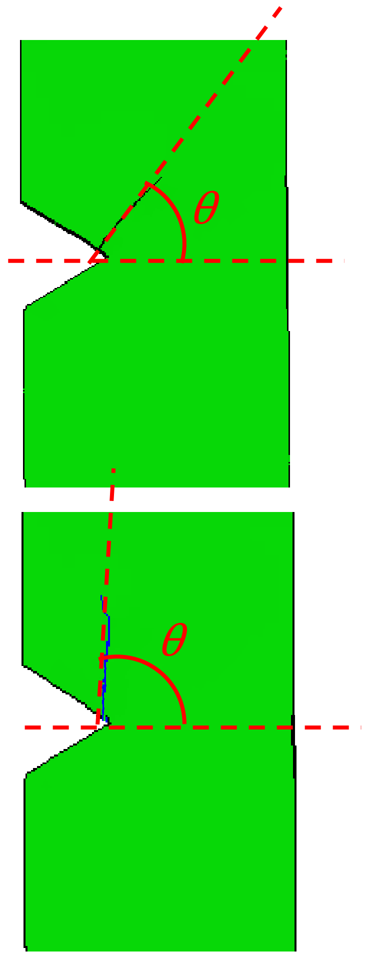
  - Crack surface is the plane that perpendicular to the maximum in-plane principal direction and parallel to the fiber direction



Example of using UDMGINI

Mode II Fracture of Cortical Bone

- **Built-in maximum principal stress criterion**
  - Crack deflection angle $\theta \sim 65°$
    - Close to analytical solution
- **Composite fracture criterion**
  - Crack deflection angle $\theta \sim 90°$
    - Similar results were observed in experimental study on human cortical bone fracture[1]



Qin, Liu and Du (6th world congress on biomechanics, 2010)

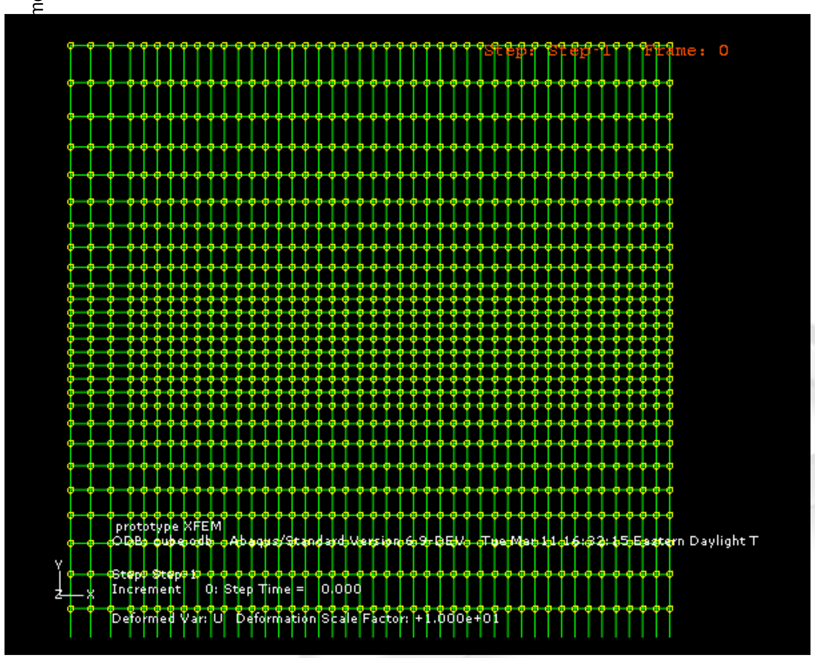
1. Zimmermann, et al. "Mixed-mode Fracture of Human Cortical Bone, Biomaterial, 2009, 30: 5877-5884

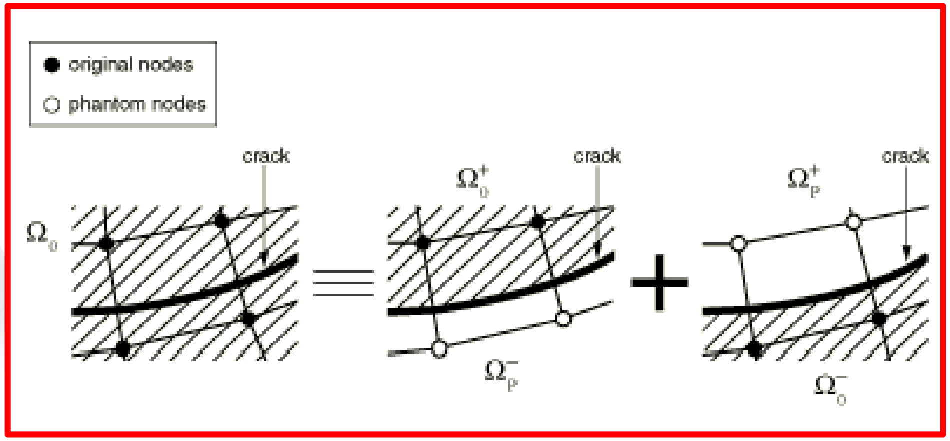
Linear Elastic Fracture Mechanics Approach (LEFM)

$$\mathbf{u} = \sum_{I=1}^{N} N_I(x)[\mathbf{u}_I + H(x)\mathbf{a}_I + \sum_{\alpha=1}^{4} F_\alpha \cancel{(x)} \mathbf{b}_I^\alpha]$$

Propagation cracks

- **Phantom nodes and Linear elastic fracture mechanics**
  - More appropriate for brittle fracture
  - Strain energy release rate at the crack tip is calculated based on the modified Virtual Crack Closure Technique (VCCT)





LEFM approach

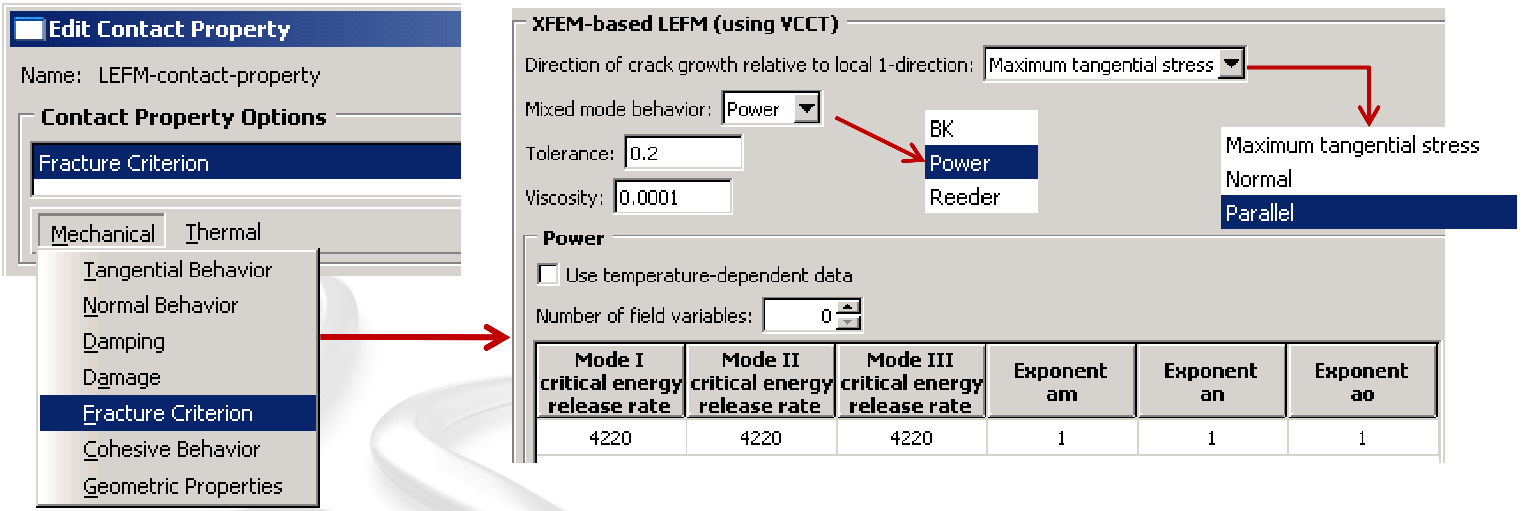
- **Linear elastic fracture mechanics in an XFEM framework**
  - A critical strain energy release rate criterion based on the Virtual Crack Closure Technique (VCCT)
    - Specified as an interaction property in association with an XFEM crack
    - Three mode-mix formulae available: the BK law, the power law, and the Reeder law models
  - User can specify the crack plane normal direction
    - The maximum tangential stress (MTS) direction is used as the default normal direction for the crack plane
    - Can choose local 1- or 2- directions

LEFM approach

- **Linear elastic fracture mechanics in an XFEM framework (cont'd)**
  - Although VCCT requires a crack to calculate the energy release rate, the LEFM approach can be used when no initial crack is present
    - Specify damage initiation in the material property definition
    - VCCT becomes active when damage initiation criteria are met

LEFM approach

- **User interface**




```
*SURFACE INTERACTION, NAME=LEFM-contact-property
*SURFACE BEHAVIOR
*FRACTURE CRITERION, TYPE = VCCT, MIXED MODE BEHAVIOR = POWER,
NORMAL DIRECTION = MTS, VISCOSITY = 0.0001
4220.,4220.,4220.,1.,1.,1.
```

LEFM approach

- **Plate with a hole example**
  - ➢ With pre-existing crack

    *SURFACE BEHVIOR

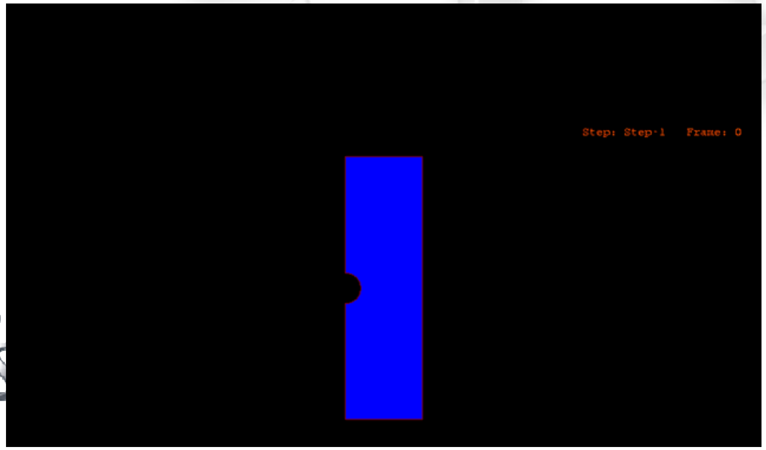
    *FRACTURE CRITERION, TYPE=VCCT, NORMAL DIRECTION=(MTS, 1, 2), VISCOSITY=
  - ➢ Without pre-existing crack

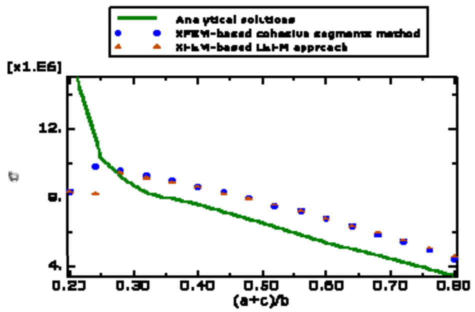
    *MATERIAL

    *DAMAGE INITIATION

    *SURFACE BEHVIOR

    *FRACTURE CRITERION, TYPE=VCCT, NORMAL DIRECTION=(MTS, 1, 2), VISCOSITY=





SIMULIA

XFEM simultaneously used with other Fracture and Failure Techniques

SIMULIA

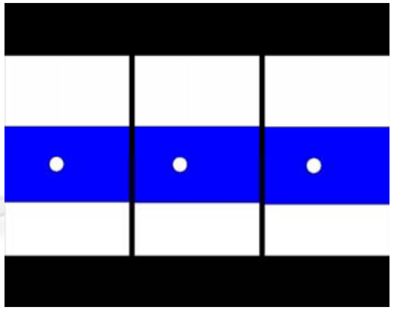
Bulk material failure and interfacial delamination

- **XFEM for matrix cracking and surface-based cohesive approach or VCCT for interfacial delamination**

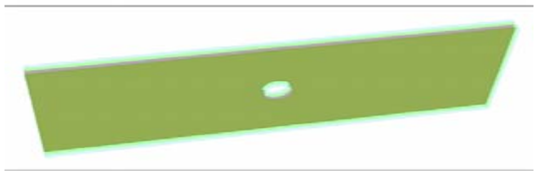
*DAMAGE INITIATION, Criterion=(*QUADE, QUADS, MAXS, MAXE*, MAXPE, MAXPS), NORMAL DIRECTION=(1, 2)

$$\mathrm{Max}\left(\left\{ \frac{\langle t_n \rangle}{t_n^0} \right\}, \left\{ \frac{t_s}{t_s^0} \right\}, \left\{ \frac{t_t}{t_t^0} \right\} \right) = 1$$

$$\left\{ \frac{\langle t_n \rangle}{t_n^0} \right\}^2 + \left\{ \frac{t_s}{t_s^0} \right\}^2 + \left\{ \frac{t_t}{t_t^0} \right\}^2 = 1$$



45/-45/90/0



(Courtesy: Bristol University)

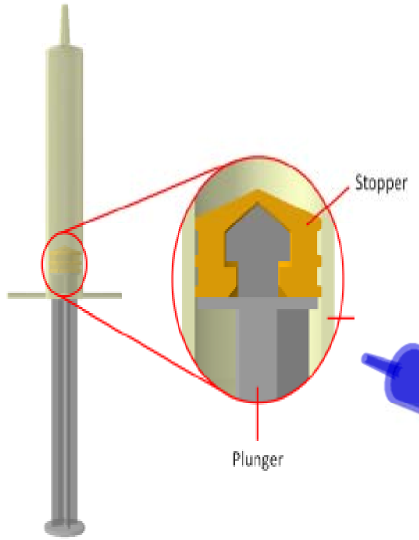
- **Can also be used with UMAT to include fiber failure**

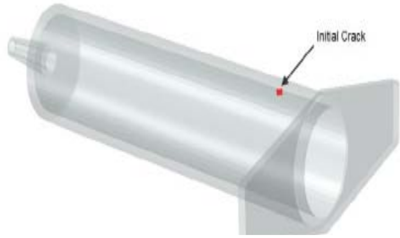
Analysis procedures

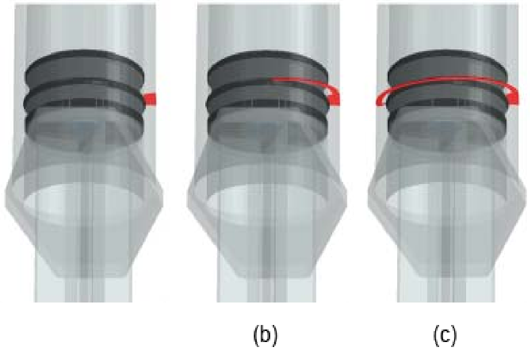
SIMULIA

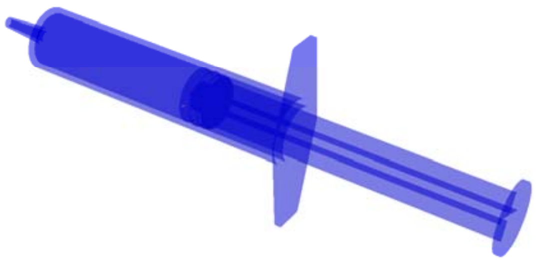
Static

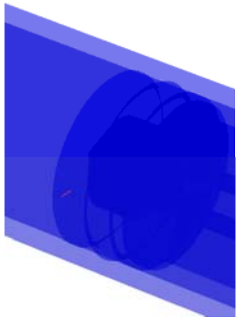
Prefilled syringe failure analysis (Abaqus Technology Brief)

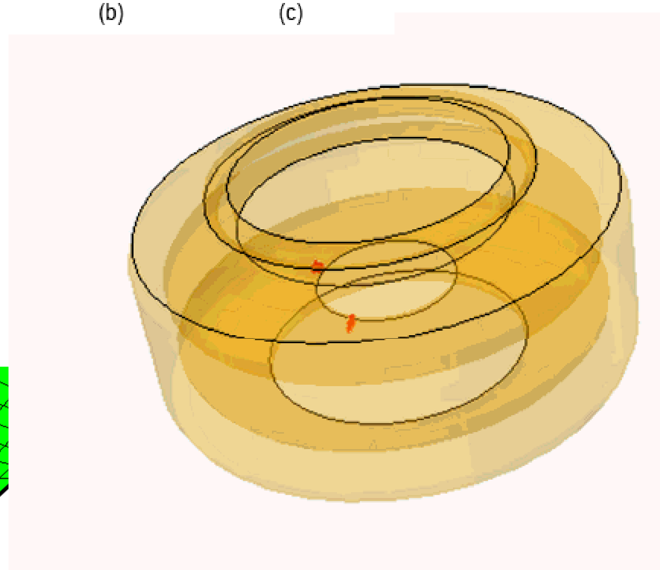


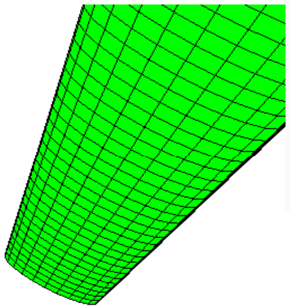


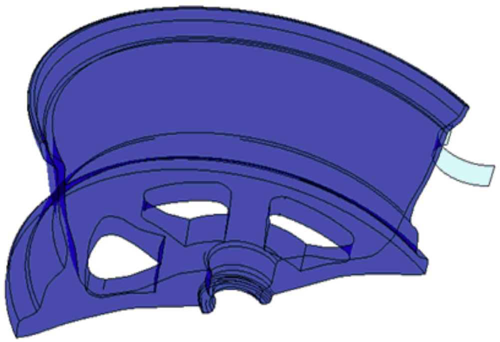






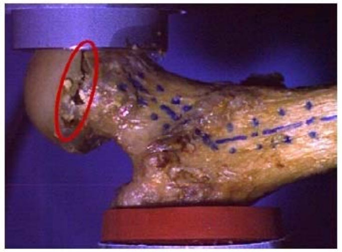


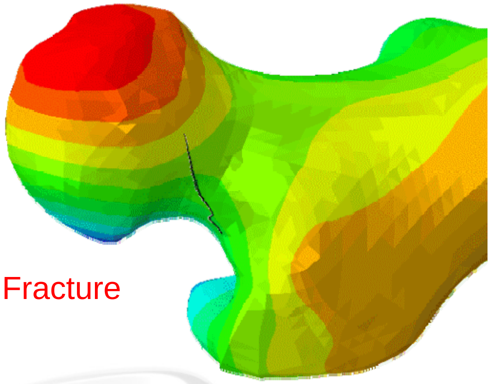




Implicit dynamic

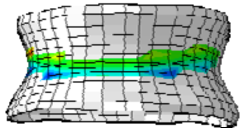
- Sports injuries



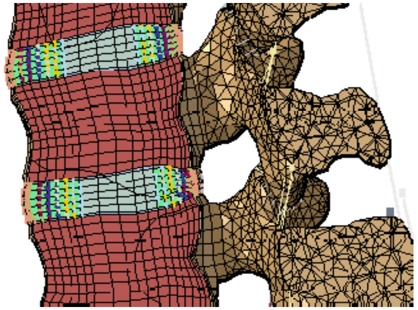


Fracture

(Courtesy: Mayo Clinic)

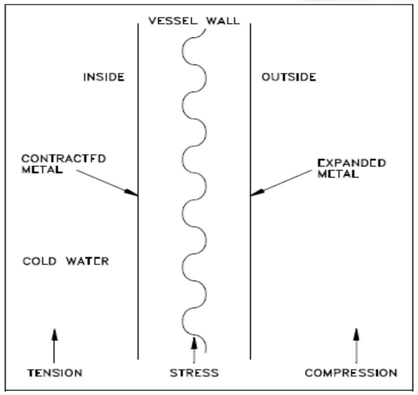


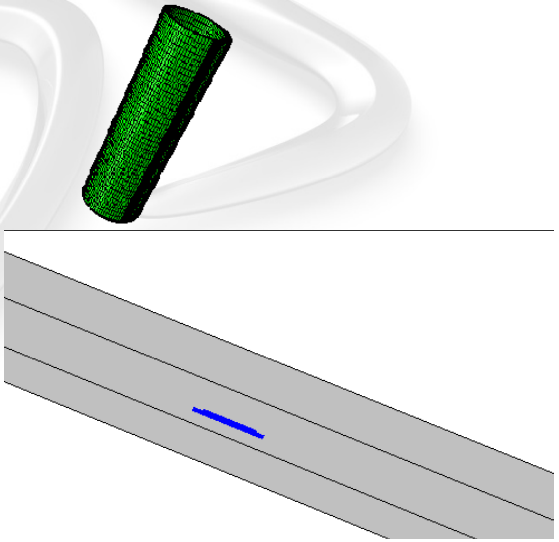
Spine fracture

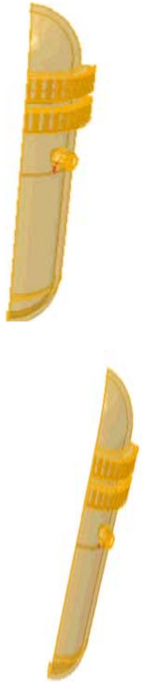


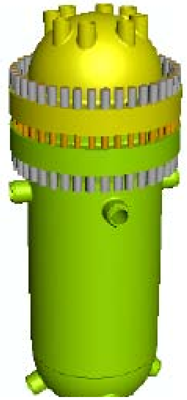
Applications:
➢A dynamic impact followed by a static loading.
➢A dynamic event (Sports injuries, Pressurized thermal shock).
➢An inertia term to stabilize a quasi-static analysis.

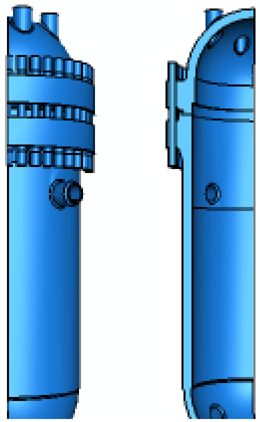
- Pressurized thermal-shock







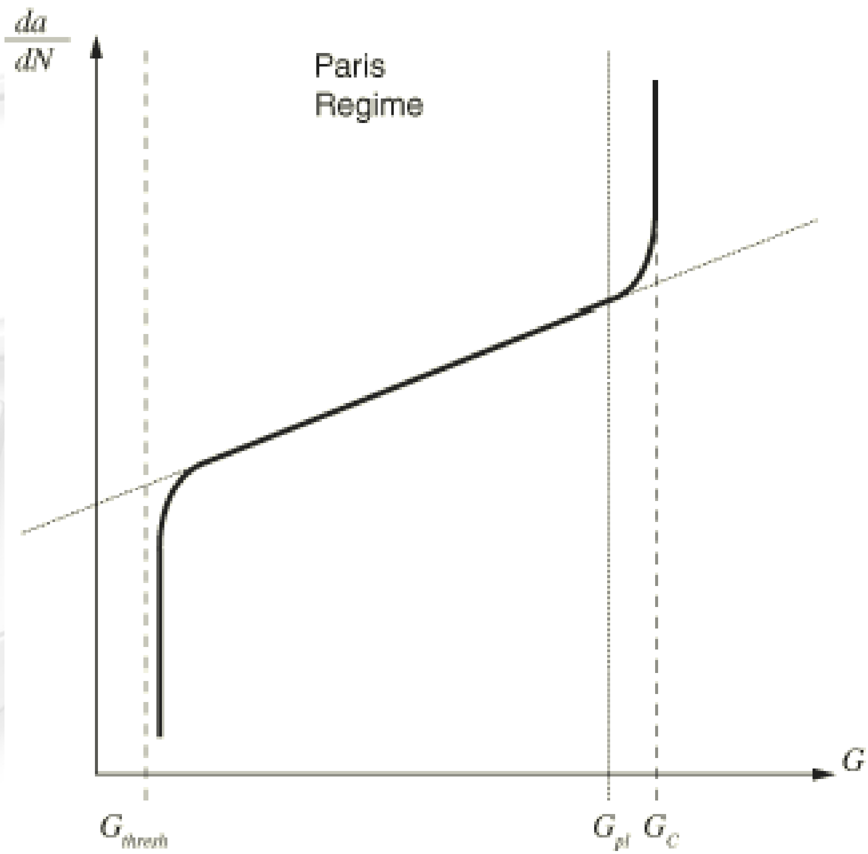




Low cycle fatigue

- Based on linear elastic fracture mechanics approach
- The onset and fatigue crack growth are characterized by using the Paris law

$$\frac{da}{dN} = c_3 \Delta G^{c_4}$$



SIMULIA

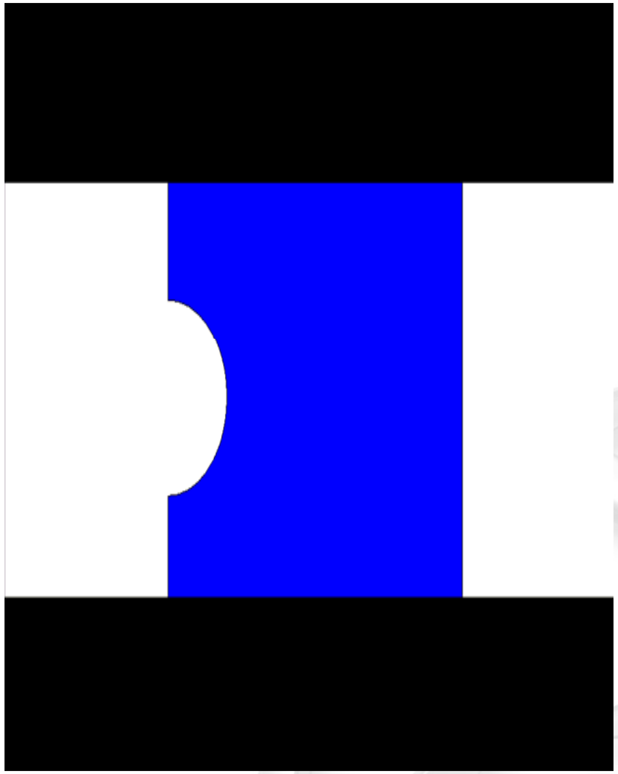
Low cycle fatigue

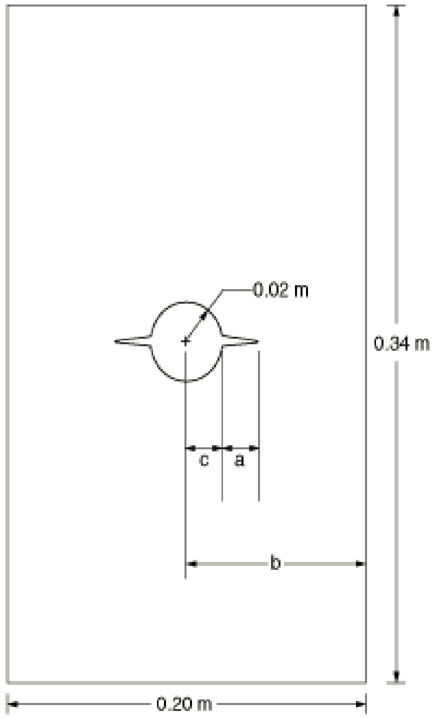
- Defined in the framework of direct cyclic procedure
- Assumes a pre-existing crack (follows aero-industry practice)
  - If you perform a fatigue analysis in a model without a pre-existing crack, you must precede the fatigue step with a static step that nucleates a crack
  - The crack can then grow along an arbitrary path under cyclic fatigue loading
- Usage (not currently supported by Abaqus/CAE)

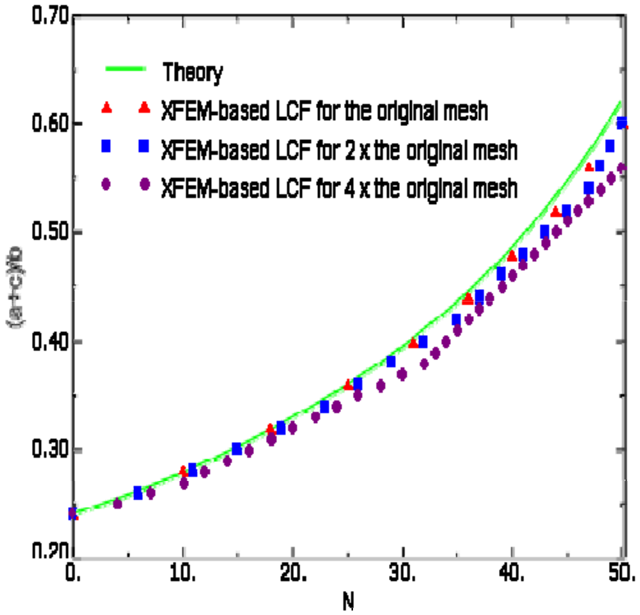
```
*SURFACE INTERACTION, NAME=LCF-contact-property
*SURFACE BEHAVIOR
:
*FRACTURE CRITERION, TYPE = FATIGUE,
MIXED MODE BEHAVIOR = BK | POWER | REEDER
```

Plate with a hole

A static loading to nucleate a crack followed by a low cycle fatigue loading







XFEM used with other Analysis Techniques

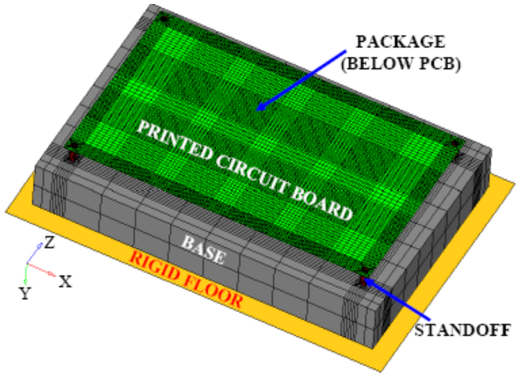
SIMULIA

Global/submodeling approach

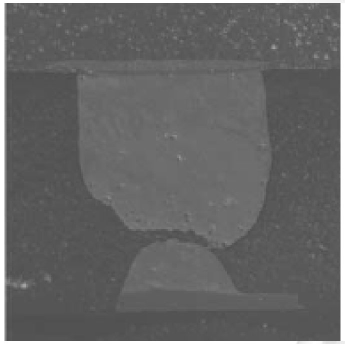
SIMULIA

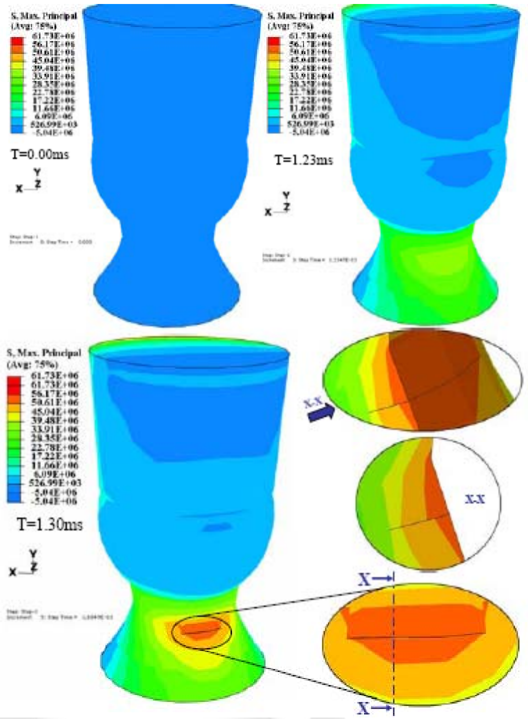
Global/local model approach with XFEM

XFEM for drop of PCB using Global-local approach



Explicit Global Model (Test Board –A)

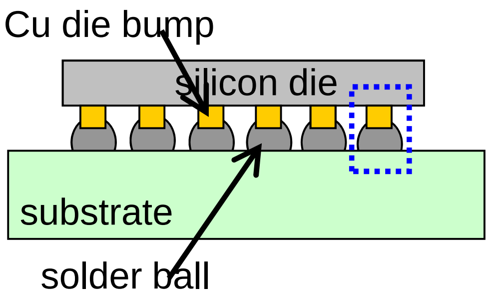


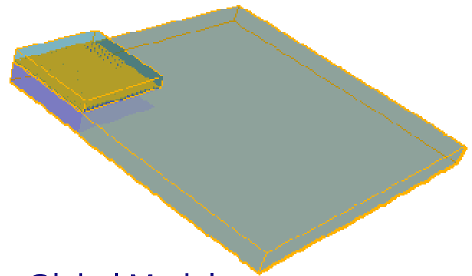


Crack Initiation and Propagation in CBGA Hi-Pb solder joints at different time intervals after the occurrence of impact using XFEM

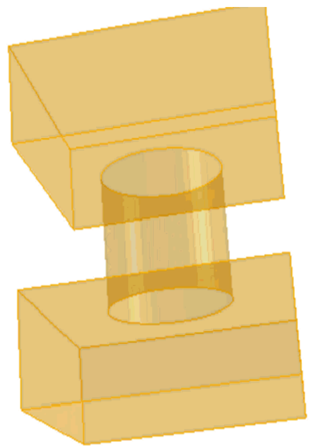
(Courtesy: Auburn University)

XFEM for silicon failure in a chip package subjected to thermal loading





Global Model



Local Model

Co-Simulation in Abaqus

SIMULIA

Brief Introduction to Co-Simulation

- The co-simulation technique is a multiphysics capability that provides several functions, available within Abaqus or as separate add-on analysis capabilities, for run-time coupling of Abaqus and another analysis program;
- Within the context of XFEM, co-simulation technique makes complex fracture analyses feasible by coupling Abaqus/Standard to Abaqus/Explicit;
  - Abaqus/Standard provides XFEM
  - Abaqus/Explicit is more efficient for solving complex contact interactions
- Identify an interface region using either node sets or surfaces when coupling Abaqus/Standard to Abaqus/Explicit;
- Time increments do not have to be the same-allow Abaqus to subcycle;
- For Abaqus/Standard to Abaqus/Explicit co-simulation, you do not define the fields exchanged; they are determined automatically according to the procedures and co-simulation parameters used.

	Import	Export
Implicit dynamic	CF, FV LUMPEDMASS PRESS, TEMP, U, V	CF, COORD PRESS, U, V
Explicit dynamic	CF, LUMPEDMASS U, V	CF, COORD V

Example 1— Notch specimen under three point bend impact loading

- **Identify the analysis program**

  *Co-simulation, name=Int-1, controls=Int-1_Ctrls, program=ABAQUS

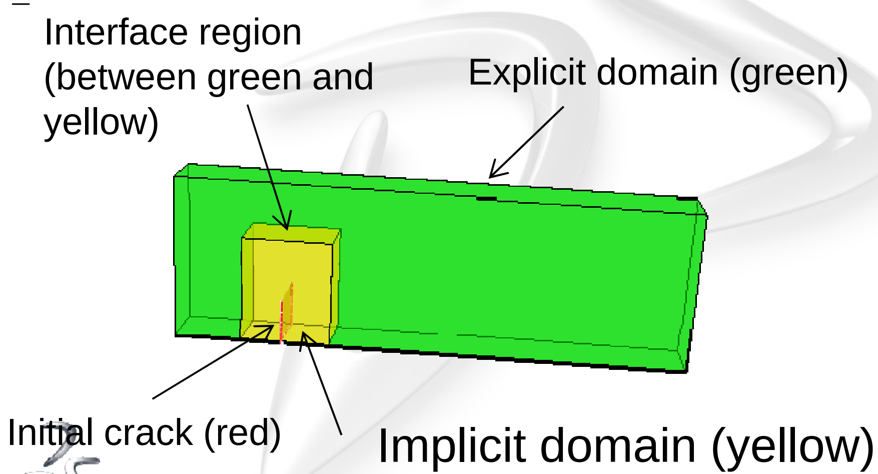
- **Identify the interface region in the model**

  *Co-simulation Region, type=SURFACE

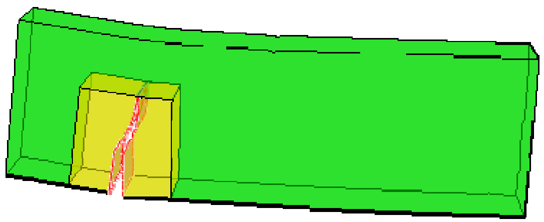
  crackRegion-1.crackRegionTieSurf

- **Define the time incrementation scheme**

  *Co-simulation Controls, name=Int-1_Ctrls, time incrementation=subcycle



Final deformed geometry

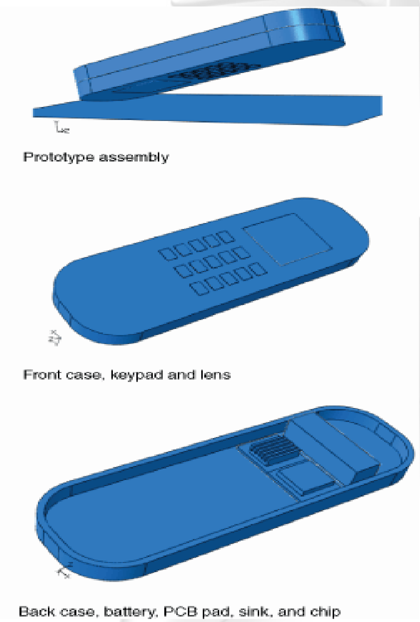


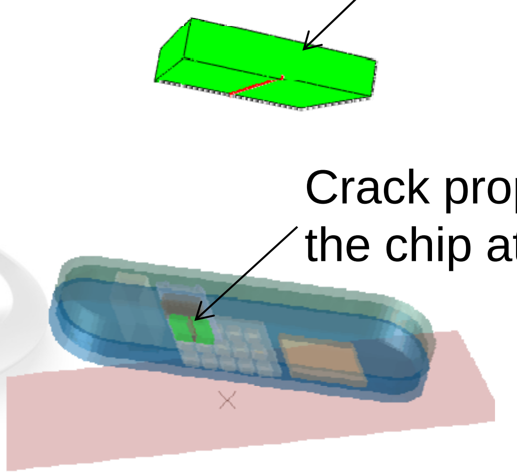
Abaqus/Standard and Abaqus/Explicit co-simulation execution

- **abaqus cosimulation <u>cosimjob</u>=*cosim-job-name* <u>job</u>=*comma-separated pair of job names* [<u>cpus</u>={*<u>number-of-cpus</u>* | *comma-separated pair of number-of-cpus*}]**

Example 2—Cellular phone drop

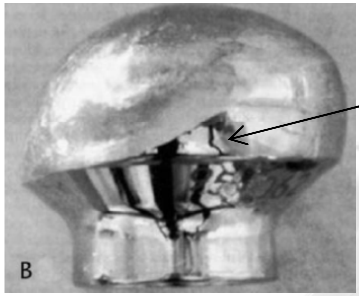
- The chip with an initial defect is modeled with Abaqus/Standard
- The rest of the body is modeled with Abaqus/Explicit—has advantages in handling complex contacts
- Co-simulation interactions using subcycling are defined on the chip and the PCB support surfaces



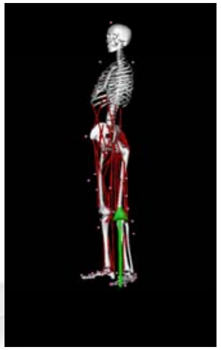


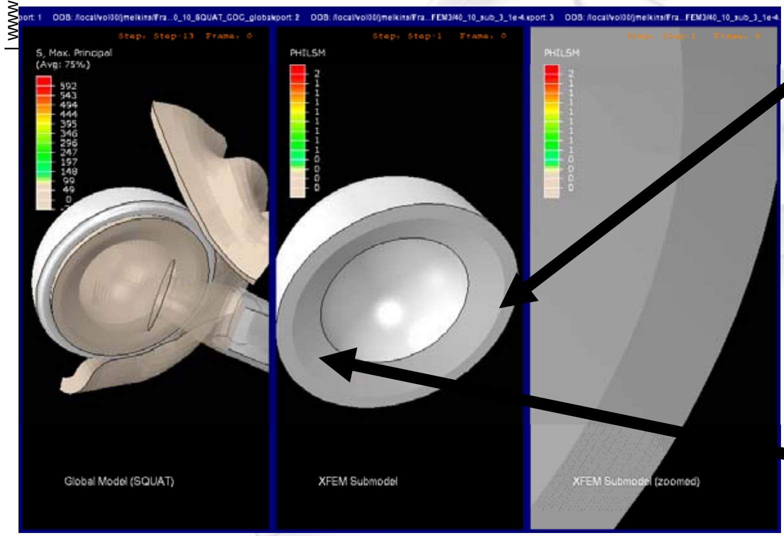
Customer examples with Co-Simulation and XFEM

- **Take advantage of XFEM technology in Implicit.**
- **Take advantage of Explicit in handling complex contacts with relatively short dynamic response times and for the analysis of extremely discontinuous events or processes.**
- **Co-simulation interactions using subcycling are defined on the interface.**



Crack



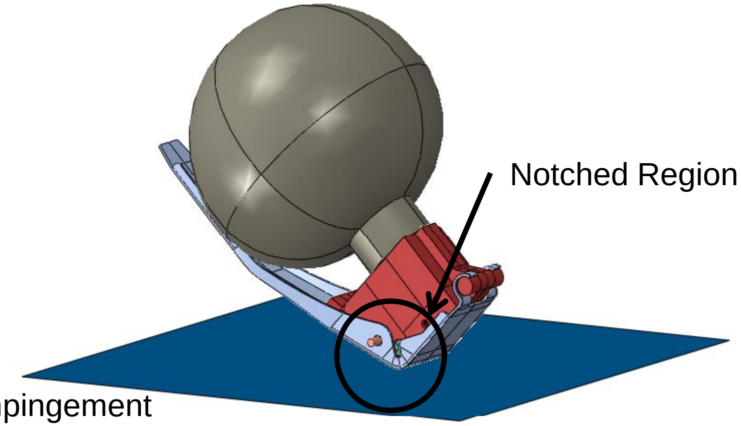


Neck Impingement

Impingement-prone Kinematics:
Sit-to-stand (STS)
Low STS
Supine roll-ove
Seated cross l
Stooping
Squatting
Shoe tie

Egress site

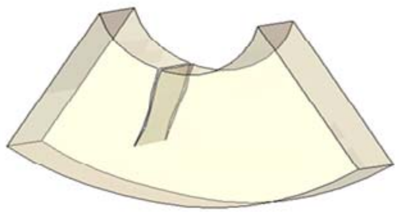
Ceramic Total Hip Bearing Fracture (Courtesy of Iowa University, 2011 SCC)



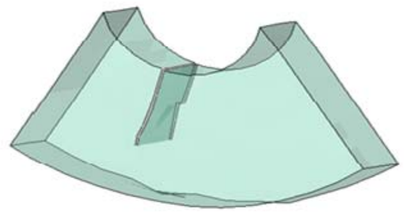
Notched Region

Interior View of Crack Growth with XFEM

Implicit Dynamics



Co-simulation



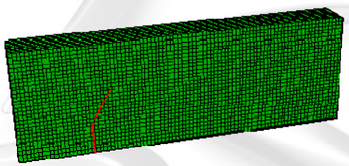
An M67 hand grenade drop test
(Courtesy of US ARMY, 2011 SCC)

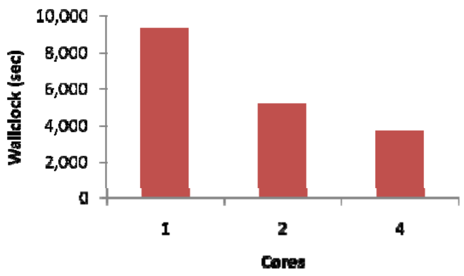
Elements, Outputs and Others

- **Elements supported for propagation cracks**
  - ✓ First-order 2D and 3D stress/displacement solid continuum elements, and second-order stress/displacement tetrahedron elements.
- **Elements supported for stationary cracks**
  - ✓ First-order 3D stress/displacement solid continuum elements, and second-order stress/displacement tetrahedron elements.

- **Support element loop parallel**
  - Crack propagation simulations can be relatively expensive, especially when small increment sizes are needed.
  - Parallelization allows faster turnarounds.

160,000 dofs





- **Output variables**
  - ✓ PHILSM----Signed distance function to describe the crack surface.
  - ✓ PSILSM --- Signed distance function to describe the initial crack front.
  - ✓ STATUSXFEM----Status of the enriched element. (The status of an enriched element is 1.0 if the element is completely cracked and 0.0 if the element contains no crack. If the element is partially cracked, the value of STATUSXFEM lies between 1.0 and 0.0.)
  - ✓ ENRRTXFEM---All components of strain energy release rate when linear elastic fracture mechanics with the extended finite element method is used.

Demonstration

- The direct link to the Introduction to the Abaqus 6.9 web-based training site is: http://www.simulia.com/services/training/wbtAbaqus69
- Customers access the site via SIMULIA Answer 4177

click - **XFEM fracture modeling** from the menu to the left to go straight to the ~10 min XFEM demo.