Abaqus Technology Brief

TB-04-FMCAE-1

Revised: April 2007

Fracture Mechanics Study of a Compact Tension Specimen Using Abaqus/CAE

Summary

Abaqus/CAE includes modeling and postprocessing capabilities for fracture mechanics analyses. These features provide interactive access to the contour integral fracture mechanics technology in Abaqus/Standard. Several fracture-specific tools are available, such as those for creating seam cracks, defining singularities, selecting the crack front and crack tip, defining q-vectors or normals to the crack front, and creating focused meshes. With these tools models can be created to estimate ***J***-integrals, stress intensity factors, and crack propagation directions. In this technology brief a standardized compact tension specimen is modeled, and ***J***-integral results are compared with those generated from applicable American Society for Testing and Materials (ASTM) standards and from a laboratory testing method. It is shown that Abaqus results are in very close conformance with the experimental results.

Background

The compact tension (CT) specimen has been standardized by the ASTM for use in the experimental determination of the fracture toughness of metallic materials.

A schematic diagram of a CT specimen testing apparatus is shown in Figure 1. A clevis and pin arrangement is used to hold the specimen. The precracked specimen is loaded at a controlled rate, and the resulting load-displacement data are recorded. Analysis of the experimental data allows the material fracture toughness to be determined in terms of the stress intensity factor K or the ***J***-integral.

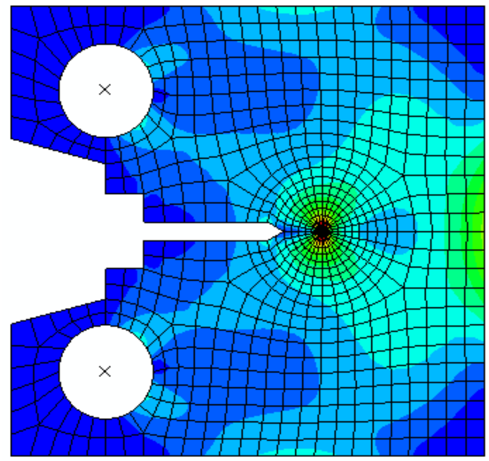
Prior to Version 6.5 the fracture mechanics capabilities of Abaqus/Standard were not supported interactively; the inclusion of fracture-specific tools into Abaqus/CAE allows for more efficient development of fracture mechanics models. In this technology brief it is shown how a fracture mechanics analysis of a low alloy ferritic steel CT specimen will be conducted with Abaqus/CAE and Abaqus/Standard. ***J***-integral values are computed and compared to those calculated with standard analytical methods.

Finite Element Analysis

The dimensions of the specimen under consideration are shown in Figure 2. The initial crack length (not shown) is 5 mm.

The elastic modulus of the specimen material is 213 GPa, and Poisson's ratio is 0.3. The yield stress is approximately 715 MPa, and the true stress versus logarithmic strain curve for this material is plotted as in Figure 3.

A two-dimensional plane-strain model is analyzed in Abaqus. The loading pins are modeled as rigid bodies. The specimen is loaded by applying a displacement to the pins in the vertical direction; all other motions of the pin are restrained. Surface-to-surface contact with a finite-sliding formulation is defined between the pins and the specimen. Two analysis steps are used. In the first step contact is established between the pins and the specimen by applying a small displacement (1×10^{-5} mm) in the vertical direction. In the second step controlled displacement loading of the pins is applied.



Key Abaqus Feaftures and Benefits

- For two- and three-dimensional models interactive definition of seam cracks, crack fronts, and **q**-vectors.
- Automatic generation of focused meshes, with interactive control of element degeneracy and midside node positioning to facilitate crack-tip singularity modeling.
- Estimation of ***J***-integrals, C_t-integrals, T-stress, stress intensity factors (K_I, K_{II}, K_{III}), and crack propagation direction.
- Visualization of node sets used in every contour integral and history plots of results for every contour.

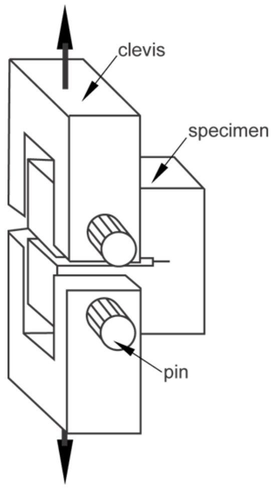


Figure 1: Apparatus for testing compact tension specimen (Ref. 1)

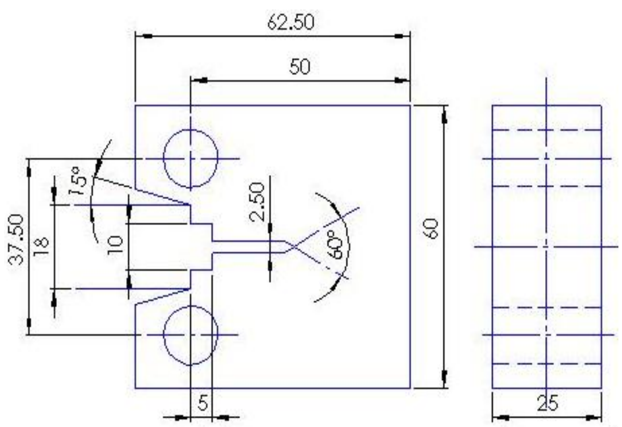


Figure 2: Compact tension specimen (all dimensions are in mm)

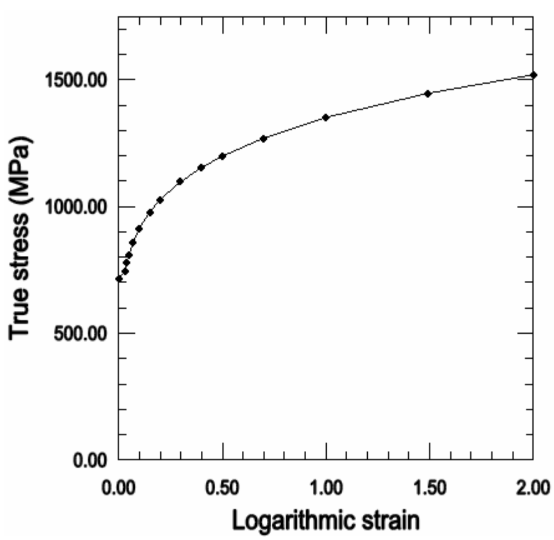


Figure 3: True stress–logarithmic strain curve for the specimen material (Ref. 2)

Definition of the model in Abaqus/CAE

The partitioned geometry of the model is shown in Figure 4. The load line displacement, which will be needed for postprocessing purposes, is evaluated at the points marked by yellow dots.

In Figure 4 the specimen crack is highlighted by a bold black line. From the Interaction module of Abaqus/CAE the crack is introduced into the model with a seam. A seam is defined in the model as an edge (in a two-dimensional part) or a face (in a solid part) that is originally closed but can open during an analysis. Abaqus/CAE places overlapping duplicate nodes along a seam when the mesh is generated.

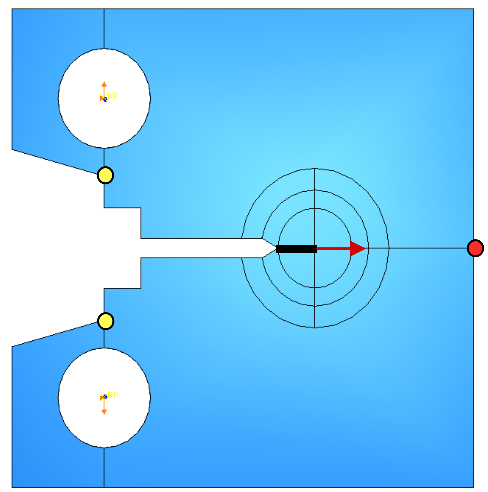


Figure 4: Partitioned two-dimensional compact tension specimen

From the crack editor, shown in Figure 5, the virtual crack extension direction is specified with the **q**-vector. In the present model it is defined with the starting point at the crack tip and the end point at the red dot in Figure 4; the resulting **q**-vector is shown in red. The crack extension direction can also be specified in terms of the normal to the crack plane.

For a sharp crack the strain field becomes singular at the crack tip. Including the singularity at the crack tip for a small-strain analysis improves the accuracy of the ***J***-integral, stress intensity factors, and the stress and strain calculations.

The partitioning of the geometry is defined by the circular lines centered on the crack tip (Figure 4); this partitioning strategy facilitates the generation of a focused mesh. The crack tip is meshed using a ring of collapsed quadratic quadrilateral elements. Second-order elements are generally used to obtain a mesh singularity at the crack tip. As shown in Figure 6, the crack editor is also used to specify the definition of the singularity.

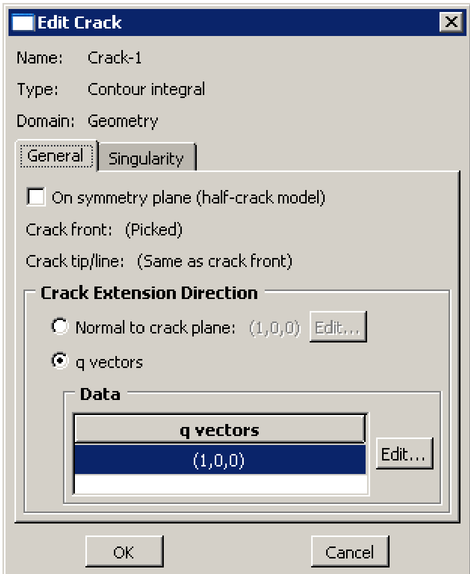


Figure 5: Defining the crack extension direction

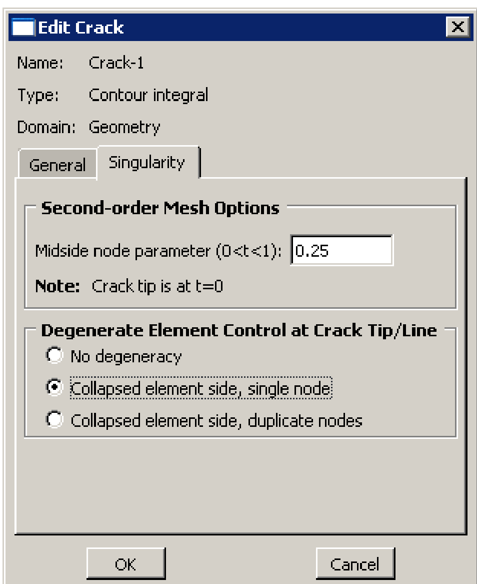


Figure 6: Defining the crack-tip singularity parameters

Specification of the midside node parameter and the crack-tip element degeneracy allows different singularity types to be defined. In the present analysis a value of 0.25 is used for the midside node parameter. This definition moves the midside nodes on the element sides adjoining the collapsed edge to the 1/4 points of the elements. At the crack tip the element sides are collapsed with single-node-type degenerate element control. These settings combine to create a $1/\sqrt{r}$ singularity in strain.

The circular partitioned areas are meshed using the "swept meshing" technique; this method allows the mesh to be regular and focused. The inclusion of the seam and singularity definition causes Abaqus/CAE to create automatically collapsed elements with correct connectivity definitions.

The remaining portion of the model is free meshed using the "medial axis" meshing algorithm. The edge-based tools for specifying mesh seeding facilitate the development of a focused mesh around the crack tip.

The mesh used in the present analyses is shown in Figure 7. As specified in the crack editor, the "single node" degeneracy method was used for the elements at the crack tip. In this approach the collapsed element edges are defined by repeating nodes in the element connectivity.

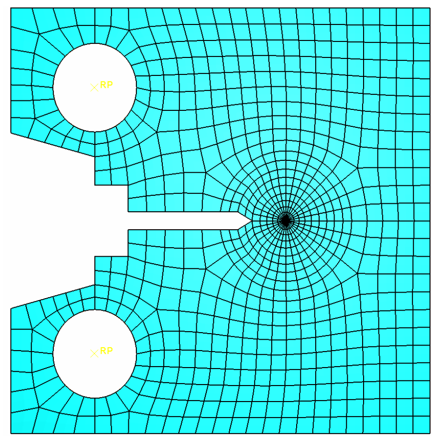


Figure 7: A focused mesh with elements having collapsed edges around the crack tip

Results and Conclusions

In the second analysis step contour integral output is requested for 10 contours. When the crack is defined in the Interaction module, the crack front and the crack tip must be specified. All the elements within the crack front and one layer of elements outside the crack front are used to determine the first contour integral. In computing the additional contour integrals, Abaqus automatically adds a single layer of elements to the group of elements that were used to calculate the previous contour integral.

The results obtained from Abaqus for the ***J***-integral are compared with the results computed by ASTM standard methods and with the laboratory testing method used in Ref. 1.

Both of the latter methods require the history of pin reaction force versus load line displacement. The area under this curve must be calculated for every increment of recorded load line displacement. The reaction force versus load line displacement curves and area calculations can be done easily using the *X*–*Y* data tools in the Visualization module of Abaqus/CAE.

ASTM calculations of J

The ***J***-integral estimates from Abaqus are first compared with the results obtained using the method outlined in ASTM standards E1737-96 and E399-90 (Refs. 3 and 4). The formulae used in these standards are outlined in Appendix A.

In Figure 8 the pin reaction force is plotted against load line displacement. Figure 9 is generated by calculating the total area under the curve in Figure 8 for each value of load line displacement; ten values are computed. The curve in Figure 9 plots the total work done in opening the crack; this quantity is the sum of the elastic and plastic work.

The plastic work is required for the ASTM calculation and can be found by subtracting the elastic work from the total work.

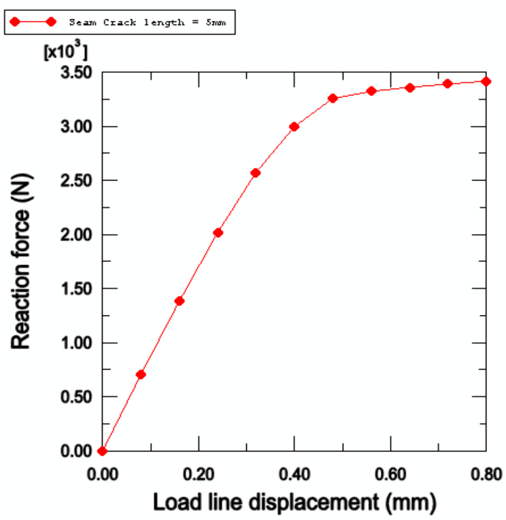


Figure 8: Pin reaction force versus load line displacement for a seam crack 5 mm in length

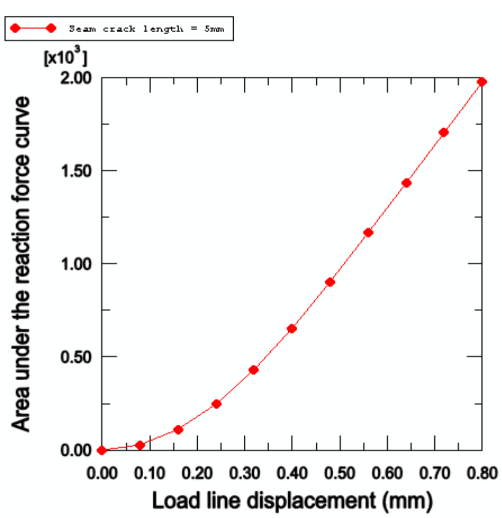


Figure 9: Area under the reaction force curve plotted for a seam crack of 5 mm

Laboratory measurement of J for small strain

This technique requires the Abaqus analysis to be repeated for different length seam cracks. In the present study the analysis is repeated for additional crack lengths of 3 and 7 mm.

For each of the models (3, 5, and 7 mm crack lengths) the area under the reaction force/load line displacement curve is found and plotted for every increment of load line displacement; these curves are shown in Figure 10. The ***J*** -integral values are then found by differentiating the curves at a seam crack length of 5 mm. The *X–Y* data operations in the Visualization module of Abaqus/CAE are used for these computations.

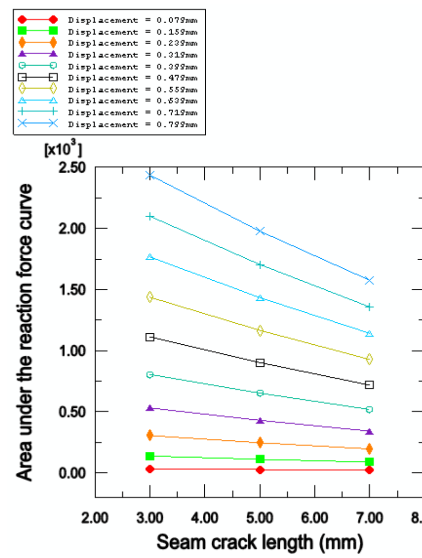


Figure 10: Area under the reaction force curve versus seam crack length for 10 load line displacement values

Comparison of Abaqus small-strain analysis results with those from ASTM standards and the laboratory testing method

As seen in Table 1, the results obtained from Abaqus are in very close conformance with the results computed using ASTM standards and the method in Ref. 1.

In Figure 11 the ***J***-integral values for all analysis methods are plotted. Usually the ***J***-integral for the first contour is ignored because of numerical inaccuracies in the stresses and strains at the crack tip. The effect of the inaccuracy is less pronounced in small-strain problems than in finite-strain problems.

The present analysis considers an elastic-plastic material. As shown in Figure 3, the material is not perfectly plastic but exhibits hardening. The nature of the crack-tip singularity for such a material is between that of a linear elastic material, which exhibits a $1/\sqrt{r}$ singularity, and that of a perfectly plastic material, which exhibits a $1/r$ singularity.

*Table 1: Comparison of **J**-integral values.*

Load line displacement (mm)	*J*-integral (N/mm)		
	Abaqus (5^{th} contour)	ASTM (E1737-96)	Anderson (Ref. 1)
0	0	0	0
0.0797	6.33	6.19	6.296
0.159	24.99	23.87	24.828
0.239	55.03	56.53	54.696
0.319	95.24	98.88	94.69
0.399	144.03	148.19	143.194
0.479	198.78	203.19	197.58
0.559	255.69	259.01	254.58
0.639	313.3	312.7	312.43
0.719	371.45	371.86	370.956
0.8	430.05	425.78	430.05

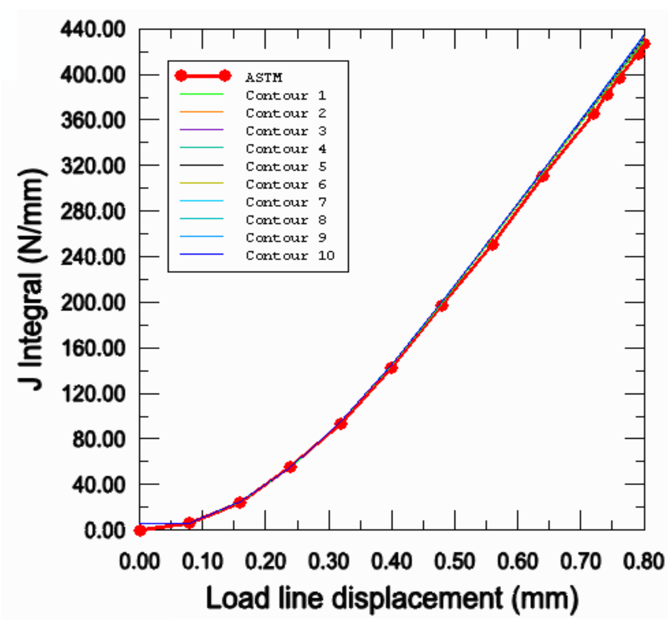


*Figure 11: **J**-integral results obtained for 10 contours using small-strain analysis*

Two-dimensional analyses are relatively insensitive to the strength of the singularity if a fine mesh is used around the crack tip and a sufficient number of contours are evaluated. An accurate far-field value of the ***J***-integral can still be obtained with a $1/\sqrt{r}$ singularity, as evidenced by the correlation between the Abaqus results and the analytical results.

Comparison of Abaqus finite-strain analysis results with those from ASTM standards

A second set of analyses was conducted in which finite strains were considered and the crack front region was selected to be larger than the plastic zone around the crack tip. This was accomplished by selecting the circular partitioned region immediately surrounding the crack tip as the first contour region when creating the crack in the Interaction module. The Abaqus results are compared only with the ASTM calculation, since these depend only on the load-displacement behavior of the specimen and not the strain magnitudes at the crack tip.

The results from the second analysis set are shown in Figure 12. It is clear that the Abaqus results are in very close conformance with the ASTM standard results. The inclusion of finite-strain effects only changes the results slightly for this analysis because the nonlinearity in the analysis is highly localized at the crack tip and does not affect the global behavior.

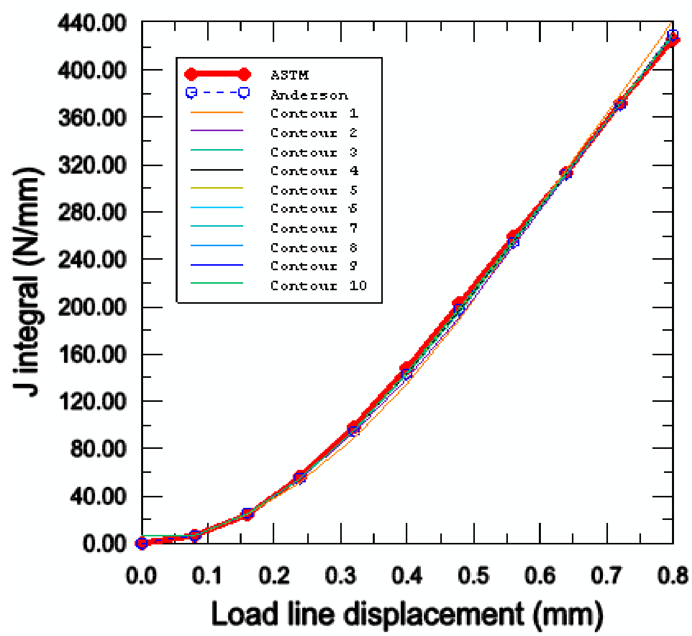


*Figure 12: **J**-integral results obtained for 10 contours using finite-strain analysis*

Appendix A

The calculations of the ***J***-integral are made from load and load-line displacement curves. Begin by writing ***J*** in terms of its elastic and plastic components:

$$J = J_{el} + J_{pl}$$

or

$$J = \frac{K^2(1-\nu^2)}{E} + \frac{\eta A_{pl}}{B(W-a)}$$

where

$$J_{el} = \frac{K^2(1-\nu^2)}{E} \quad \text{and} \quad J_{pl} = \frac{\eta A_{pl}}{B(W-a)}$$

with

$$K = \frac{P}{B\sqrt{W}} f(a/W)$$

For a compact tension specimen,

$$f(a/W) = \frac{2+\frac{a}{W}}{\left(1-\frac{a}{W}\right)^{3/2}} \left[\begin{array}{l} 0.886 + 4.64\left(\frac{a}{W}\right) - 13.32\left(\frac{a}{W}\right)^2 \\ +14.72\left(\frac{a}{W}\right)^3 - 5.60\left(\frac{a}{W}\right)^4 \end{array} \right]$$

and

$$\eta = 2 + 0.522 \times {}^{b}\!/_{W}$$

$$b = W - a$$

A_{pl} = plastic work, found from the load versus load line displacement curve

W = distance between the point of application of load to the point marked by a red dot in Figure 4

a = distance between the load line and the crack tip

B = thickness of the specimen

References

1. Anderson, T. L., "Fracture Mechanics: Fundamentals and Applications," CRC Press, 1991.
2. Brocks, W., and I. Scheider, "Numerical Aspects of the Path-Dependence of the ***J***-integral in Incremental Plasticity—How to calculate reliable ***J***-values in FE analyses," GKSS FORSCHUNGSZENTRUM internal report—GKSS/WMS/01/08, 2001.
3. ASTM Standards, "Test Method for ***J***-integral characterization of Fracture Toughness, ASTM E1737-96," American Society for Testing and Materials, Philadelphia, 1996.
4. ASTM Standards, "Standard Test Method for Plane-Strain Fracture Toughness of Metallic Materials, ASTM E399-90," American Society for Testing and Materials, Philadelphia, 1997.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, see the following Abaqus 6.11 documentation references:

- Abaqus/CAE User's Manual
  - "Fracture mechanics," Section 31
- Abaqus Analysis User's Manual
  - "Contour integral evaluation," Section 11.4.2