ABAQUS/CAE User's Manual

20.1 Modeling fracture mechanics

You can study the onset of cracking in a quasi-static model by selecting regions from your model that will be used to compute contour integral estimates. During the analysis ABAQUS/Standard writes the values of the contour integrals to the output database as history output. For a detailed description of contour integrals, see "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual. This section describes how you define a crack in ABAQUS/CAE. The following topics are covered:

- "An overview of contour integral analysis," Section 20.1.1
- "Defining the crack front," Section 20.1.2
- "Defining the crack tip or crack line," Section 20.1.3
- "Defining the crack extension direction," Section 20.1.4
- "Controlling the singularity at the crack tip for a small-strain analysis," Section 20.1.5
- "Contour integral output," Section 20.1.6
- "Meshing the crack region and assigning elements," Section 20.1.7
- "What is a seam?," Section 20.1.8
- "Creating a seam," Section 20.1.9
- "Controlling the singularity at the crack tip," Section 20.1.10
- "Defining data for contour integrals," Section 20.1.11
- "Modifying data for contour integrals," Section 20.1.12
- "Requesting contour integral output," Section 20.1.13

ABAQUS/CAE User's Manual

20.1.1 An overview of contour integral analysis

You can study the onset of cracking in quasi-static problems using contour integral estimates; however, a contour integral estimate does not predict how a crack will propagate. You can compute contour integrals for two- or three-dimensional models, and you can choose to perform one of the following types of contour integral calculations:

- J-integral
- C_t-integral (for creep)
- T-stress (for linear materials)
- Stress intensity factors for linear homogeneous materials and for interfacial cracks lying on the interface between two linear homogeneous materials

For more information, see "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

You define cracks in the Interaction module. To perform a contour integral analysis, you must select the crack front, the crack tip or crack line, and the crack extension direction, as described in the following sections. The entities that you can select depend on whether the part is two- or three-dimensional and whether you are

defining the part using geometry from an ABAQUS native part or using elements and nodes from an orphan mesh part. In some cases the crack tip or the crack line is the same as the crack front that you selected, and ABAQUS/CAE selects the crack tip or crack line for you.

A crack in a two-dimensional model is a region containing edges that are free to move apart. A crack in a three-dimensional model is a region containing faces that are free to move apart. The simplest approach to performing a contour integral analysis uses a region that already contains edges or faces that are free to move apart as the crack separates. Alternatively, you can model the crack as a line embedded in a face in a two-dimensional model or as a face embedded in a cell in a three-dimensional model. The embedded line or face is called a seam, and you can perform a contour integral analysis using a seam. When you mesh the model, ABAQUS/CAE creates duplicate overlapping nodes on the seam; these coincident nodes are free to move apart as the seam separates. Seams are described in more detail in "What is a seam?," Section 20.1.8. Figure 20–1 shows a two-dimensional model with existing edges that are free to move apart and a similar model with an embedded seam that is free to move apart.

Figure 20–1 You can define a crack using an existing region or using a seam.
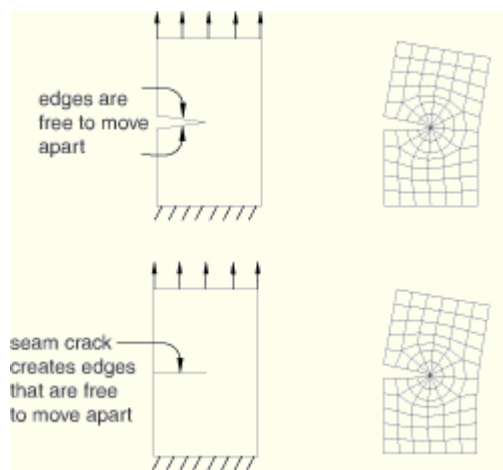
You can specify that the crack front is defined on a symmetry plane, in which case you need to model only half of the structure. ABAQUS doubles the values calculated by the contour integral to arrive at the correct values. In most cases you need a refined mesh surrounding the crack. After you create a crack, you must use the **History Output Request** editor to request that ABAQUS write contour integral information to the output database. For more information, see "Contour integral output," Section 20.1.6.

For detailed instructions, see "Defining data for contour integrals," Section 20.1.11.

ABAQUS/CAE User's Manual

20.1.2 Defining the crack front

The first step in the procedure to configure a contour integral is to define the crack front by selecting entities from the assembly. The crack front is the forward part of the crack. ABAQUS uses the crack front to compute the first contour integral using all of the elements inside the crack front and one layer of elements outside the crack front. You can request output for more than one contour integral, in which case ABAQUS/CAE adds a

single layer of elements to the group of elements that were used to calculate the previous contour integral. Figure 20-2 illustrates how ABAQUS/CAE computes successive contour integrals for a two-dimensional model by adding layers of elements.

Figure 20-2 Successive contour integrals are calculated by adding a layer of elements.
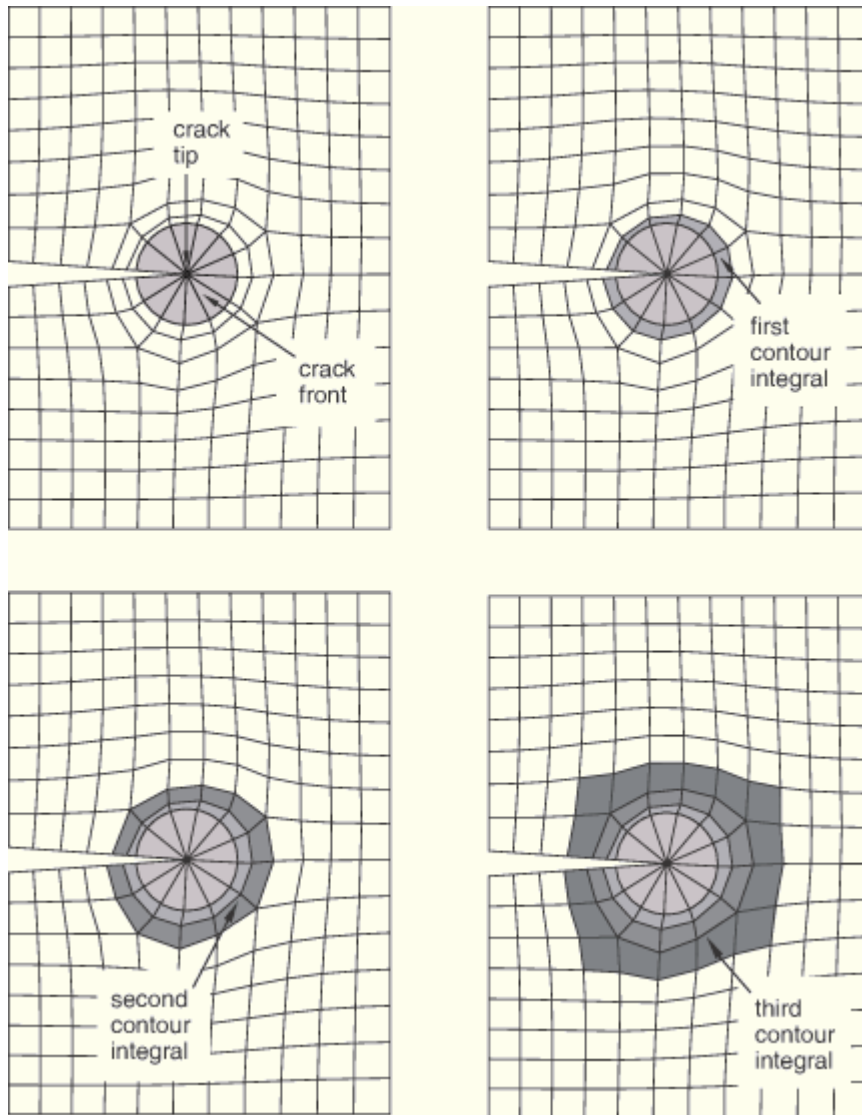
If your part is three-dimensional, ABAQUS computes contour integrals at each node along the crack line, as shown in Figure 20-3. For more information, see "Defining the crack front" in "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

Figure 20-3 ABAQUS computes the contour integral at each node along the crack line.
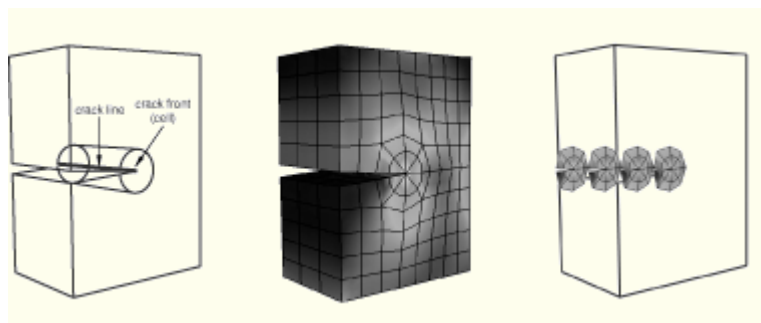
The entities from which you can select depend on whether the part instance is an ABAQUS native part or an orphan mesh part and on the modeling space of the part.

ABAQUS native part instance

When you are defining the crack front on an ABAQUS/CAE native part, the entities that you can select depend on the modeling space of the part.
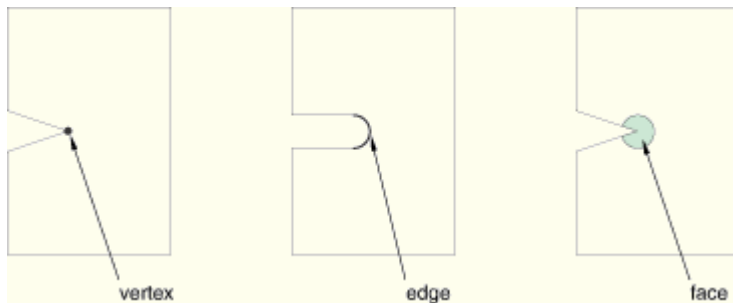
Two-dimensional part instance

If you are defining the crack front on a two-dimensional ABAQUS/CAE native part, you can select the following:

- A single vertex
- Connected edges
- Conected faces

[Figure 20–4](#) shows the entities from which you can select when defining a crack front on a two-dimensional native part.

Figure 20-4 Selecting the crack front from a two-dimensional native part.

Three-dimensional part instance

If you are defining the crack front on a three-dimensional ABAQUS/CAE native part, you can select the following:

- Connected edges
- Connected faces
- Connected cells

[Figure 20–5](#) shows the entities from which you can select when defining a crack front on a three-dimensional native part.

Figure 20-5 Selecting the crack front from a three-dimensional native part.

Orphan mesh part instance

When you are defining the crack front on an orphan mesh part, you can select the elements or element edges or faces that define the crack front. Alternatively, you can select the nodes from the corresponding region. When you are defining the crack front on an orphan mesh part, the entities that you can select depend on the modeling space of the part.
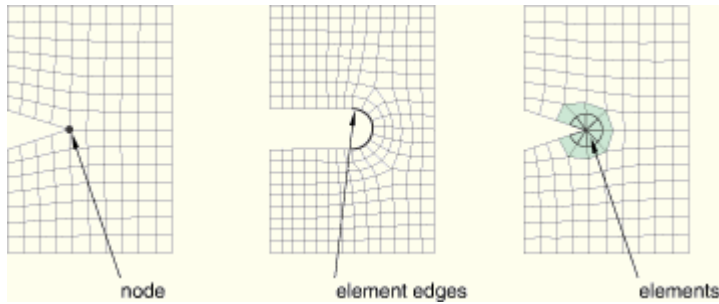
Two-dimensional part instance

If you are defining the crack front on a two-dimensional orphan mesh part, you can select the following:

- A single node
- Connected element edges
- Connected elements

Figure 20–6 shows the entities from which you can select when defining a crack front on a two-dimensional orphan mesh part.

Figure 20-6 Selecting the crack front from a two-dimensional orphan mesh part.


Three-dimensional part instance

If you are defining the crack front on a three-dimensional orphan mesh part, you can select the following:

- Connected element edges
- Connected element faces
- Connected elements

Figure 20–7 shows the entities from which you can select when defining a crack front on a three-dimensional orphan mesh part.

Figure 20-7 Selecting the crack front from a three-dimensional orphan mesh part.

ABAQUS/CAE User's Manual
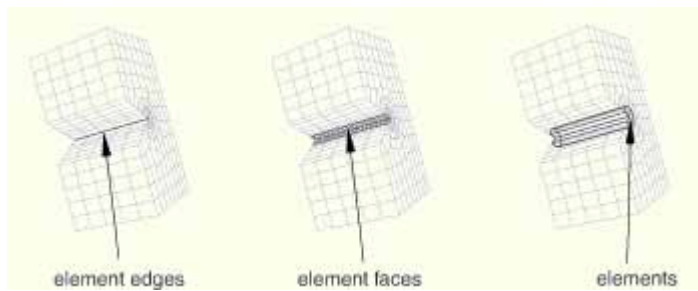
20.1.3 Defining the crack tip or crack line

After you have defined the crack front, the next step in the procedure to configure the contour integral is to define the crack tip or crack line by selecting entities from the assembly. The modeling space of your assembly governs whether you need to define a crack tip or a crack line for a contour integral analysis.

Two-dimensional

If your assembly is two-dimensional, you must define the crack tip by selecting a vertex or a node. The crack tip is the point on the crack front where you define the crack extension direction, q. In some cases the desired vertex or node will not exist, and you must create it by partitioning the crack front.

If you selected a vertex or a node to define the crack front, the same vertex or node defines the crack tip.

Three-dimensional

If your part is three-dimensional, you must define the crack line by selecting edges or element edges that form a continuous line. The crack line is a series of connected edges along the crack front where you define the crack extension direction, q. In some cases the desired edges will not exist, and you must create them by partitioning the crack front.

If you selected edges or element edges to define the crack front, the same edges define the crack line.

Selected edges (from an ABAQUS/CAE native part or from an orphan mesh) must be connected, must connect one side of the crack front to the other, and must be included in the crack front.

ABAQUS/CAE User's Manual

20.1.4 Defining the crack extension direction

After you have defined the crack front along with the crack tip or crack line, ABAQUS/CAE prompts you to specify the crack extension direction at the crack tip or along the crack line. You can specify the normal to the crack plane, n; or you can specify the crack extension direction, q, directly. For a detailed discussion of how ABAQUS uses n or q to calculate the crack extension direction, see "Specifying the virtual crack extension direction" in "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

Normal to crack plane

If you select **Normal to crack plane**, you can define the normal by selecting points from the model that represent the start and the end of the normal to the crack plane. The crack plane contains the q vector that ABAQUS needs to compute the contour integral. In many cases the crack plane represents the plane of symmetry of the crack.

You should define the normal to the crack plane only if the direction is the same at all points along the crack line. If the direction of the normal to the crack plane varies along the crack line, you cannot select a single normal that defines the crack extension direction at all points along the crack line.

To define the normal to the crack plane, you can select points from an ABAQUS native part (such as a vertex, datum point, or midpoint), or you can select nodes from an orphan mesh part. Alternatively, you can enter the coordinates of the points in the prompt area. If you select a point from an ABAQUS native part and subsequently

modify the part, ABAQUS/CAE regenerates the points and updates the normal accordingly. If you are working with an orphan mesh part, you must select nodes that represent the start and the end of the normal.

ABAQUS calculates a crack extension direction, q, that is orthogonal to the crack front tangent, t, and the normal, n.

q vectors

If you select **q vectors**, you can define the crack extension direction, q, directly by selecting points from the model that represent the start and end of the q vector. If you are working with an orphan mesh part, you must select nodes that represent the start and the end of the q vector. Alternatively, you can enter the coordinates of the points in the prompt area.

Figure 20–8 shows an example of a crack front formed by a circular arc along which the direction of the q vector is constantly varying. In contrast, the normal to the crack plane is constant. As a result, for this case you should define the crack extension direction by specifying the normal to the crack plane.

Figure 20-8 The direction of the q vector varies but the direction of the normal is constant.
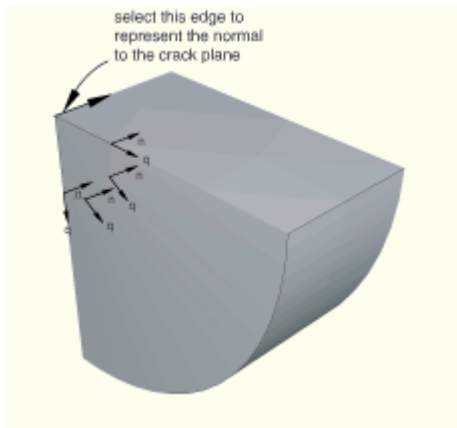
Figure 20–9 shows a crack front formed by the edge of a truncated cone along which the directions of both the q vector and the normal to the crack plane are varying.

Figure 20-9 The directions of both the q vector and the normal vary along the crack front.
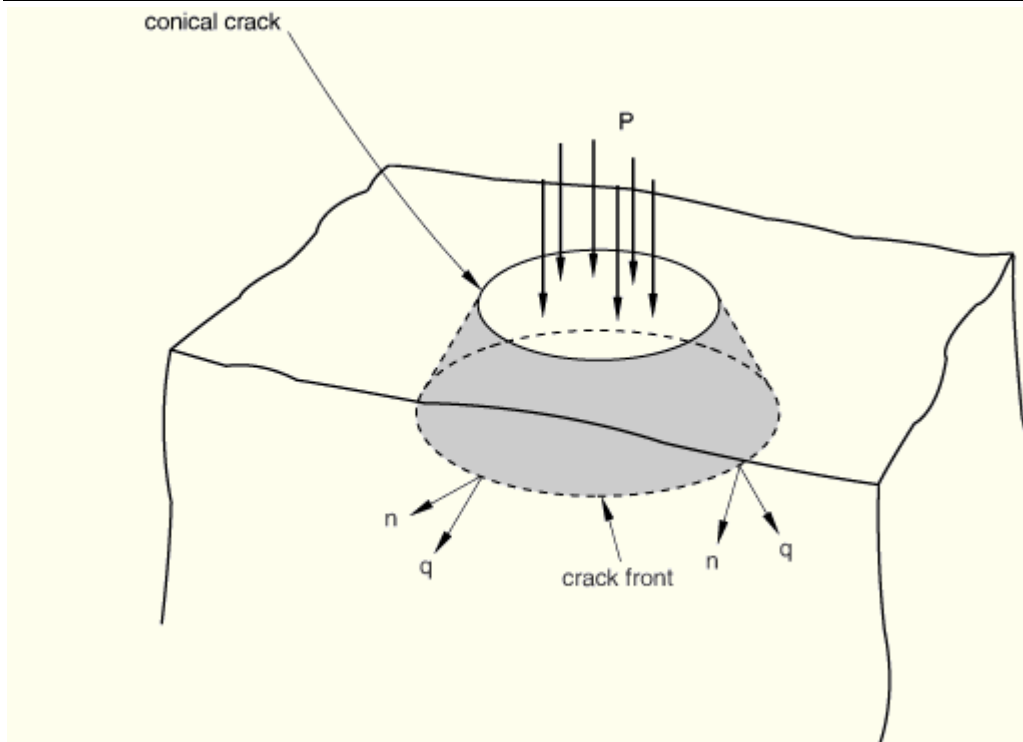
For this case you should do the following:

1. Define the contour integral, and use a single **q**-vector to specify the crack extension direction.
2. Mesh the part, create a job, and write the model to an input file.
3. Import the input file, which will create an orphan mesh representation of the part.
4. Edit the crack that was imported with the model. Each node along the crack front will have the crack extension direction defined by the **q**-vector that you specified in Step 1.
5. Use the Query toolset to determine the start and end coordinates of the **q**-vector at each node, and edit the data defining the **q**-vectors at each node along the crack front.

This technique is shown in .

Figure 20-10 Entering the **q**-vector at each node along the crack front.
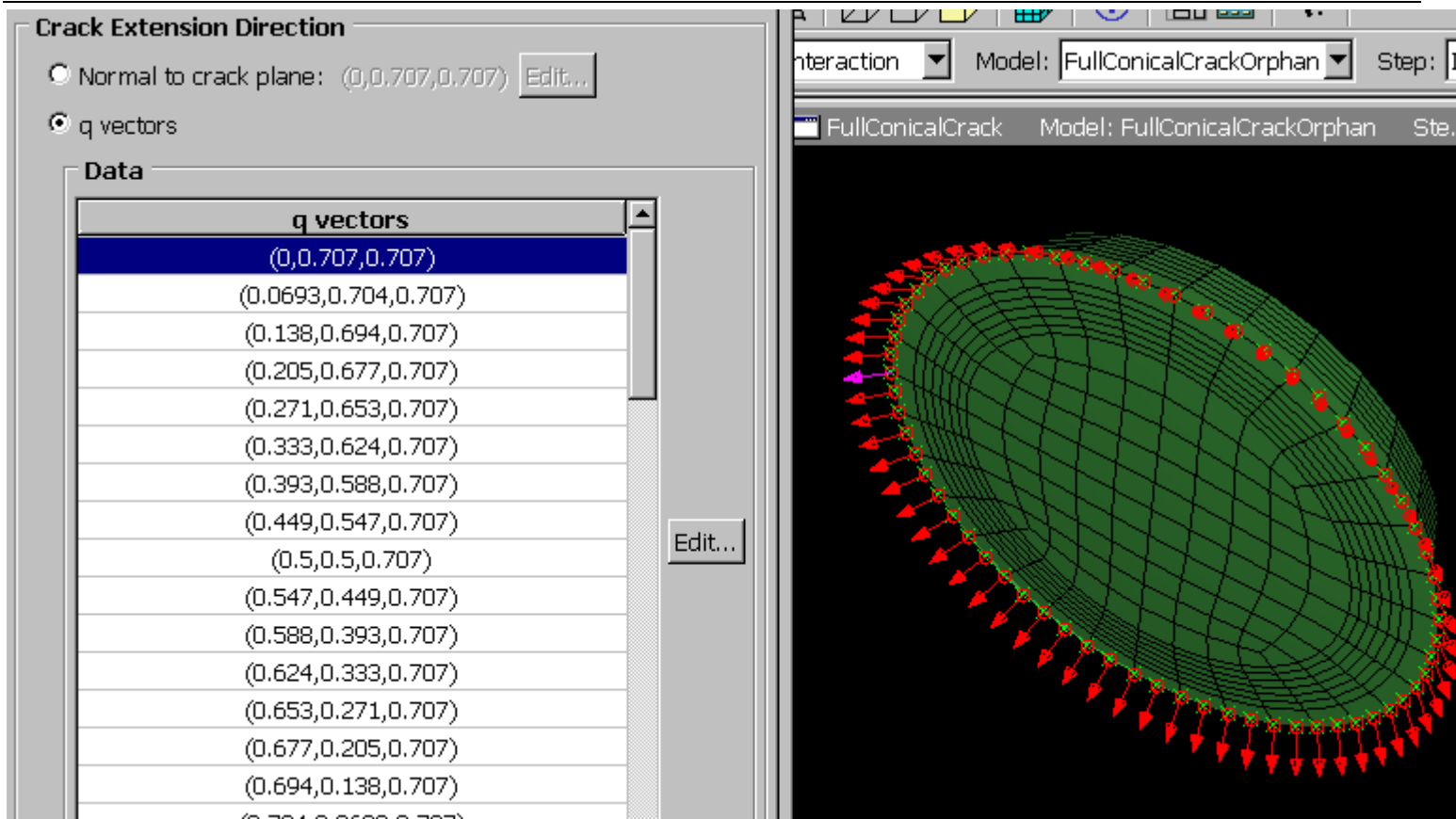
Figure 20–10 is taken from "Conical crack in a block modeled with and without submodeling," Section 1.4.2 of the ABAQUS Example Problems Manual. An ABAQUS Scripting Interface script is provided with this example that illustrates how you can enter the **q** vector at each node along the crack front.

ABAQUS/CAE User's Manual

20.1.5 Controlling the singularity at the crack tip for a small-strain analysis

If the geometry of the crack region defines a sharp crack, the strain field becomes singular at the crack tip, as described in "Constructing a fracture mechanics mesh for small-strain analysis" in "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual. Including the singularity in your model for a small-strain analysis improves the accuracy of the contour integral and the stress and strain calculations.

To include the singularity in your contour integral estimates, click the **Singularity** tab in the **Crack** editor and choose the desired position of the midside node and degenerate element control. For detailed instructions, see "Controlling the singularity at the crack tip," Section 20.1.10.

In addition, you must do the following in the Mesh module:

- If the assembly or part is two-dimensional, you must model the crack front with a ring of triangles and assign quadrilateral elements to the remainder of the contour integral region.
- If the assembly or part is three-dimensional, you must model the crack front with a ring of wedges and assign hexahedral elements to the remainder of the contour integral region.

When you mesh the crack front, ABAQUS does the following:

- Converts the elements in the crack front to collapsed quadrilateral or hexahedral elements.
- If you modeled the crack front with a ring of second-order triangles or wedges, ABAQUS moves the midside nodes to the specified position along the element edges that radiate out from the crack tip or crack line. (If you modeled the crack front with a ring of first-order triangles or wedges, ABAQUS ignores the position that you specified for the midside nodes.)

An instance of an orphan mesh part is always a dependent instance. As a result, you cannot adjust the midside nodes of instances of orphan mesh parts in the assembly. For more information, see "What is the difference between editing an orphan mesh part, a meshed part, and a meshed part instance in the assembly?," Section 41.2. You must display the original orphan mesh part and use the Edit Mesh toolset to adjust the position of the midsize nodes. For more information, see "Adjusting the position of midside nodes," Section 41.5.5. In addition, you cannot create collapsed elements from triangular and wedge elements of an orphan mesh.

*ABAQUS/CAE User's Manual*

20.1.6 Contour integral output

After you have defined the crack, you must use the **History Output Request** editor in the Step module to include the contour integral data in the output database generated by the analysis. The editor allows you to configure the following:

- The output frequency.
- The type of contour integral calculation to perform (J-integral, C_t-integral, stress intensity factors, or T-stress).
- The number of contours to evaluate.

ABAQUS writes a history output variable to the output database for each contour integral that it calculates. Figure 20–11 shows the format of the name of an output variable representing a contour integral.

Figure 20-11 The format of a contour integral name in the output database.
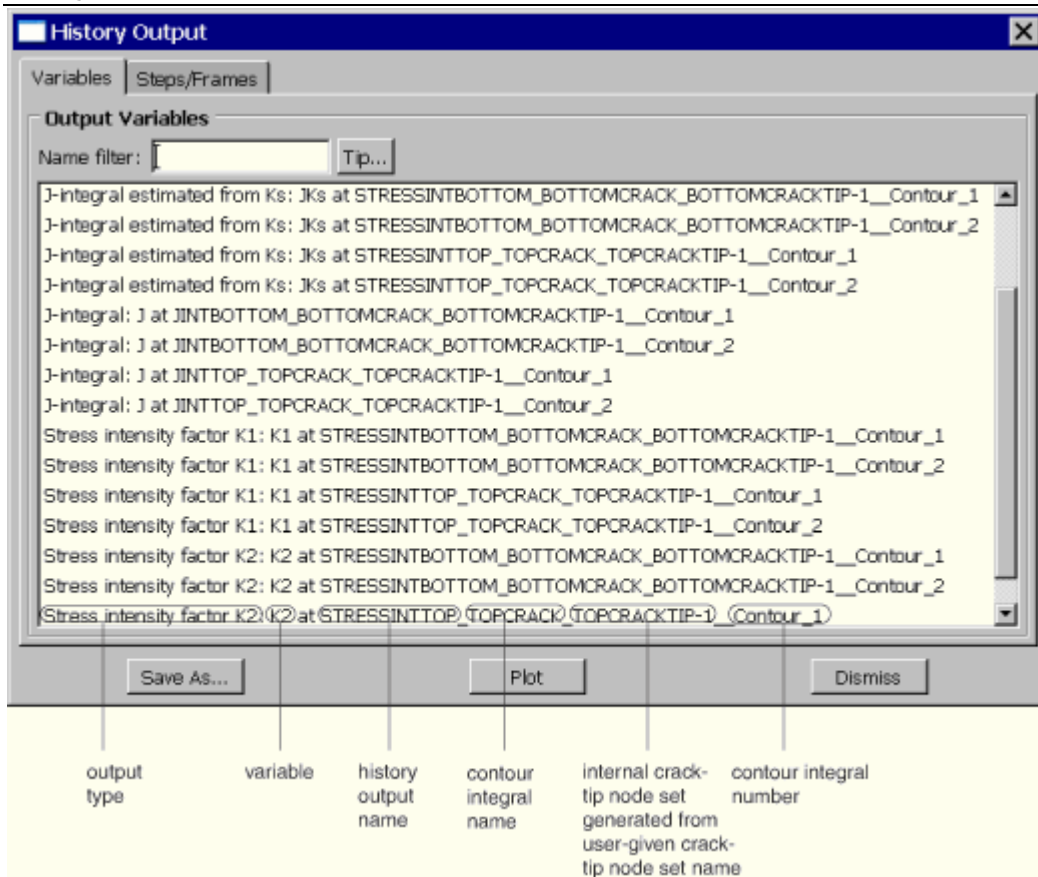
For detailed instructions, see "Requesting contour integral output," Section 20.1.13. For more information, see "Modifying history output requests," Section 14.11.3, and "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

ABAQUS/CAE User's Manual

20.1.7 Meshing the crack region and assigning elements

Large stress concentrations occur at the crack tip. As a result, you should create a refined mesh around the crack tip to get accurate results for stresses and strains. In contrast, because the J-integral is an energy measure, you can obtain accurate J-values with a relatively coarse mesh. However, if the material becomes more nonlinear, you must create a finer mesh at the crack tip to maintain the accuracy of the J-values. You can control the density of the mesh at the crack tip by partitioning the region and by assigning mesh seeds to the resulting edges. For more information, see "Understanding seeding," Section 17.4.

For a large-strain analysis during which ABAQUS will allow for nonlinear geometry, you should mesh the contour integral region with quadrilateral or hexahedral elements. For more information, see "Constructing a fracture mechanics mesh for finite-strain analysis" in "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

However, for a small-strain analysis that does not allow for nonlinear geometry, you must allow for the singularity at the crack tip or crack line by meshing the region that defines the crack front with a ring of triangles or wedges. For more information, see "Controlling the singularity at the crack tip for a small-strain analysis," Section 20.1.5.

You must use the swept meshing technique to create wedge elements; however, there are limitations on the regions that ABAQUS/CAE can mesh using the swept meshing technique, as described in "Swept meshing of three-dimensional solids," Section 17.10.3. As a result, if you cannot use the swept meshing technique, you cannot create wedge elements, and you cannot allow for the singularity at the crack line. In most cases you can ignore the singularity if your mesh is sufficiently refined to model the deformation around the crack tip or crack line and the resulting high strain gradients. You can also ignore the singularity if you are interested in only the contour integral output.
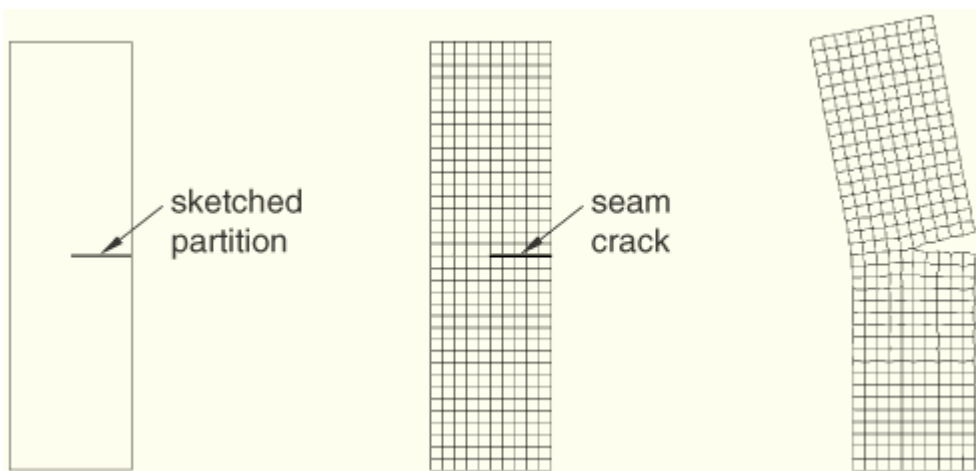
ABAQUS/CAE User's Manual

20.1.8 What is a seam?

A seam defines an edge or a face in your model that is originally closed but can open during an analysis. ABAQUS/CAE places overlapping duplicate nodes along a seam when the mesh is generated. A seam cannot extend along the boundaries of a part and must be embedded within a face of a two-dimensional part or within a cell of a solid part. After you create a seam, you can determine its crack properties using a contour integral analysis. Because a seam modifies the mesh, you cannot create a seam on a dependent part instance. A seam is not a surface, and you cannot specify which side of the seam is positive or negative. As a result, you cannot apply a load or a boundary condition to a seam.

Figure 20-12 shows a seam on the face of a planar part and the effect of applying a tensile load to the model. The duplicate nodes along the seam are independent of each other and are free to move.

Figure 20-12 A seam embedded in a face.

Figure 20–13 shows a similar analysis of a seam embedded in a solid part. The seam was created by partitioning the solid with a sketch drawn on a datum plane.

Figure 20-13 A seam embedded in a cell.
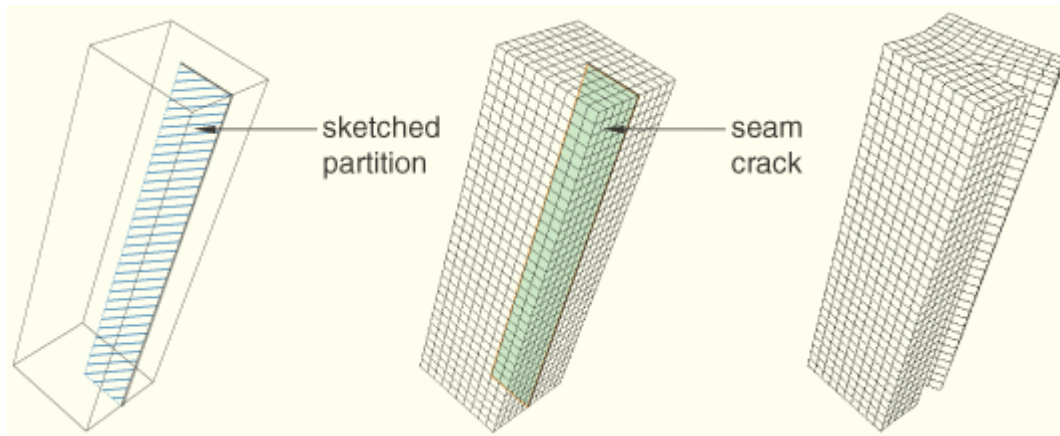
For detailed instructions, see "Creating a seam," Section 20.1.9, .

ABAQUS/CAE User's Manual

20.1.9 Creating a seam

You can use the **Special** menu in the Interaction module to define a seam in your model. A seam defines a region in your model that can open during an analysis. ABAQUS/CAE places overlapping duplicate nodes along a seam when the mesh is generated. A seam cannot extend along the boundaries of a part and must be embedded within a face of a two-dimensional part or within a cell of a solid part. You can use the seam when you are selecting the crack front and the crack tip that will be used in a contour integral analysis. For more information, see "What is a seam?," Section 20.1.8.

To create a seam:

1. From the main menu bar in the Interaction module, select **Special**⟶**Crack**⟶**Create seam**.
2. From the model in the viewport, select the entities representing the seam. The entities must be embedded edges within a face of a two-dimensional part or embedded faces within a cell of a solid part; you cannot select any entities that lie on the boundary of the part.
3. Click mouse button 2 to indicate that you have finished selecting the seam.

   ABAQUS/CAE creates the seam.

4. To include the seam in a contour integral analysis, follow the procedure described in "Defining data for contour integrals," Section 20.1.11.

For information on related topics, click any of the following items:

- "What is a seam?," Section 20.1.8
- "Fracture mechanics," Section 7.10 of the ABAQUS Analysis User's Manual

ABAQUS/CAE User's Manual

20.1.9 Creating a seam

You can use the **Special** menu in the Interaction module to define a seam in your model. A seam defines a region in your model that can open during an analysis. ABAQUS/CAE places overlapping duplicate nodes along a seam when the mesh is generated. A seam cannot extend along the boundaries of a part and must be embedded within a face of a two-dimensional part or within a cell of a solid part. You can use the seam when you are selecting the crack front and the crack tip that will be used in a contour integral analysis. For more information, see "What is a seam?," Section 20.1.8.

To create a seam:

1. From the main menu bar in the Interaction module, select **Special**→**Crack**→**Create seam**.
2. From the model in the viewport, select the entities representing the seam. The entities must be embedded edges within a face of a two-dimensional part or embedded faces within a cell of a solid part; you cannot select any entities that lie on the boundary of the part.
3. Click mouse button 2 to indicate that you have finished selecting the seam.

   ABAQUS/CAE creates the seam.

4. To include the seam in a contour integral analysis, follow the procedure described in "Defining data for contour integrals," Section 20.1.11.

For information on related topics, click any of the following items:

- "What is a seam?," Section 20.1.8
- "Fracture mechanics," Section 7.10 of the ABAQUS Analysis User's Manual

ABAQUS/CAE User's Manual

20.1.10 Controlling the singularity at the crack tip

If the geometry of the crack region defines a sharp crack, the strain field becomes singular at the crack tip, as described in "Constructing a fracture mechanics mesh for small-strain analysis" in "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual. Including the singularity in your model for a small-strain analysis improves the accuracy of the contour integral and the stress and strain calculations.

To control the singularity at the crack tip:

1. Display the **Crack** editor using one of the following methods:
   - To configure a new contour integral, follow the instructions in "Defining data for contour integrals," Section 20.1.11.
   - To edit an existing contour integral, select **Special**→**Crack**→**Edit**→*crack name* from the main menu bar in the Interaction module.

2. To include the strain singularity in your contour integral estimates, click the **Singularity** tab in the **Crack** editor, and do one of the following:

- To create a $1/\sqrt{r}$ strain singularity for an elastic fracture mechanics application:

  a. From the **Second-order Mesh Options** field, enter a value of 0.25 for the **Midside node parameter** to move the midside nodes to the $1/4$ points.

  b. From the **Degenerate Element Control at Crack Tip/Line** field, choose **Collapsed element side, single node**.

- To create a $1/r$ strain singularity for a perfect plasticity fracture mechanics application:

  a. From the **Second-order Mesh Options** field, enter a value of 0.5 for the **Midside node parameter** to keep the midside nodes at the midside points.

  b. From the **Degenerate element control at Crack Tip/Line** field, choose **Collapsed element side, duplicate nodes**.

- To create a combined $1/\sqrt{r}$ and $1/r$ strain singularity for a power law hardening material:

  a. From the **Second-order Mesh Options** field, enter a value of 0.25 for the **Midside node parameter**.

  b. From the **Degenerate element control at Crack Tip/Line** field, choose **Collapsed element side, duplicate nodes**.

3. Click **OK** to include the singularity in the contour integral estimation and to close the editor.

Note: In most cases you must assign second-order triangle or wedge elements to the crack front region to include the singularity in your contour integral estimates. If you assign first-order elements, ABAQUS ignores the value of the midside node setting in the **Singularity** tabbed page. However, if you assign first-order elements and choose **Collapsed element side, duplicate nodes** from the **Degenerate Element Control at Crack Tip/Line** field, you will create a $1/r$ strain singularity.

ABAQUS/CAE User's Manual

20.1.11 Defining data for contour integrals

You can use the **Special** menu in the Interaction module to define the data for contour integrals. For more information, see "Fracture mechanics," Section 7.10 of the ABAQUS Analysis User's Manual.

The entities that you select to define the contour integral depend on whether the part is two- or three-dimensional and whether you are defining the part using geometry from an ABAQUS native part or using elements and nodes from an orphan mesh part.

To define the data for contour integrals:

1. From the main menu bar in the Interaction module, select **Special**⟶**Crack**⟶**Create**.

   The **Create Crack** dialog box appears.

2. Enter the name of the crack, and click **Continue** to close the dialog box.

3. From the model in the viewport, select the entities representing the crack front region. For a description of the entities to select, see "Defining the crack front," Section 20.1.2.

4. Click mouse button 2 to indicate that you have finished selecting the crack front region.

5. From the model in the viewport, select the entities representing the crack-tip region. In some cases, depending on the modeling space of your model and the entities that you selected to define the crack front, ABAQUS/CAE selects the crack tip for you and skips to Step 7. For more information, see "Defining the crack tip or crack line," Section 20.1.3.

   From the prompt line, toggle on **Select mesh entities** to select entities from an orphan mesh part.

6. Click mouse button 2 to indicate that you have finished selecting the crack-tip region.

7. From the prompt area, choose the method for defining the crack extension direction.

   Normal to crack plane

   Select **Normal to crack plane** to specify the normal to the crack plane, and select points representing the start and end of the normal.

   q vectors

   Select **q vectors** to specify the crack extension direction directly, and select points representing the start and end of the vector.

   For more information, see "Defining the crack extension direction," Section 20.1.4.

   ABAQUS/CAE displays a red arrow indicating the crack extension direction and displays the **Edit Crack** dialog box.

8. Use the **Edit Crack** dialog box to configure the parameters that control a contour integral analysis.
   - Click the **General** tab of the dialog box to do the following:
     ▪ Specify that the crack front is defined on a symmetry plane that models only half of the structure.
     ▪ Change the entities that define the crack front or crack tip/line.
     ▪ Change the method for defining the crack extension direction. You can also change the entities that define the crack extension direction.

     For more information, see "Modifying data for contour integrals," Section 20.1.12.

   - Click the **Singularity** tab of the dialog box to model a singularity of the strain field at the crack tip. For more information, see "Controlling the singularity at the crack tip," Section 20.1.10.

9. Click **OK** to configure the contour integral and to close the editor.

   ABAQUS displays green crosses on the region to represent the crack front.

   You must use the **History Output Request** editor in the Step module to include the contour integral data in the output database generated by the analysis. For more information, see "Contour integral output," Section 20.1.6, and "Modifying history output requests," Section 14.11.3.

For information on related topics, click any of the following items:

- "Modeling fracture mechanics," Section 20.1
- "Fracture mechanics," Section 7.10 of the ABAQUS Analysis User's Manual

ABAQUS/CAE User's Manual

20.1.12 Modifying data for contour integrals

If desired, you can modify the entities that you selected to define the crack front, the crack tip, and the crack extension direction that will be used to define a contour integral. You can also specify that the crack front is defined on a symmetry plane that models only half of the structure. For more information, see "Symmetry" in " Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

To define a crack to be used in a contour integral analysis:

1. Display the **Crack** editor using one of the following methods:
    - To configure a new contour integral, follow the instructions in "Defining data for contour integrals," Section 20.1.11.
    - To edit an existing contour integral, select **Special**→**Crack**→**Edit**→*crack name* from the main menu bar in the Interaction module.
2. Toggle on **On symmetry plane (half-crack model)** to specify that the crack front is defined on a symmetry plane that models only half of the structure. ABAQUS doubles the change in potential energy calculated from the virtual crack front advance to compute the correct contour integral values.
3. If desired, click **Edit** to modify your selection of entities that define the crack front, crack tip, or crack line. If ABAQUS/CAE selected the crack tip or crack line region to be the same as the crack front, you cannot edit the selection.
4. If desired, change the method for defining the crack extension direction. You can also click **Edit** to modify your selection of entities that define the crack extension direction.
5. Click **OK** to configure the contour integral and to close the editor.

    ABAQUS displays green crosses on the region to represent the crack front.

    You must use the **History Output Request** editor in the Step module to include the contour integral data in the output database generated by the analysis. For more information, see "Contour integral output," Section 20.1.6, and "Modifying history output requests," Section 14.11.3.

ABAQUS/CAE User's Manual

20.1.13 Requesting contour integral output

You use the **History Output Request** editor in the Step module to request output from the contour integral analysis. You must select **Contour integral** from the **Domain** field. The editor then allows you to select the output frequency

and the type of contour integral analysis to perform. For more information, see "Modifying history output requests," Section 14.11.3, and "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

To request contour integral output:

1. Enter the Step module.
2. From the main menu bar, select Output→History Output Request→Create.

   ABAQUS/CAE displays the Create History dialog box.

3. From the Create History dialog box, enter the name of the output request and the step in which it will be created. Click Continue to close the dialog box.

   ABAQUS/CAE displays the History Output Requests editor.

4. From the Domain field, select Contour integral, and select the desired contour integral.
5. From the fields that appear in the History Output Requests editor, enter the output frequency.
6. Select the type of contour integral calculation to perform. You can select from the following:

J-integral

You use the J-integral in rate-independent quasi-static fracture analysis to characterize the energy release associated with crack growth. The J-integral can be related to the stress-intensity factor if the material response is linear. For more information, see "The J-integral" in "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

C_t-integral

You use the C_t-integral for time-dependent creep behavior, where it characterizes creep crack deformation under certain creep conditions, including transient crack growth. For more information, see "The C_t-integral" in "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

T-stress

You use the T-stress component to represent a stress parallel to the crack front. For more information, see "The T-stress" in "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

Stress intensity factors

You use the stress intensity factors K_I, K_{II}, and K_{III} in linear elastic fracture mechanics to characterize the local crack-tip stress and displacement fields. For more information, see "The stress intensity factors" in "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

If you request a contour integral calculation using stress intensity factors, ABAQUS also computes the crack propagation direction at initiation. You must choose one of the following to indicate the crack initiation criteria:

- Maximum tangential stress
- Maximum energy release rate

- K11=0

For more information, see "The crack propagation direction" in "Contour integral evaluation," Section 7.10.2 of the ABAQUS Analysis User's Manual.

7. Enter the number of contours to evaluate. ABAQUS/CAE computes the next contour integral by adding a single layer of elements to the region defined by the previous domain, as described in Figure 20-2. Each contour generates a value or a group of values for the contour integral.
8. Click OK to configure the contour integral analysis and to close the editor.

For examples of how ABAQUS names history output variables in the output database for each contour integral that it calculates, see "Contour integral output," Section 20.1.6.