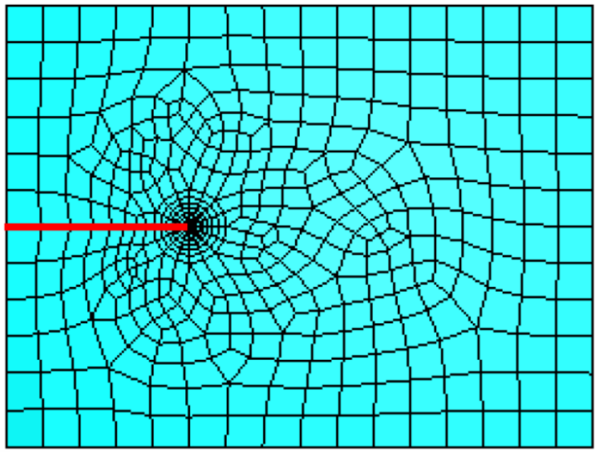
دانشگاه صنعتی اصفهان
دانشکده مکانیک

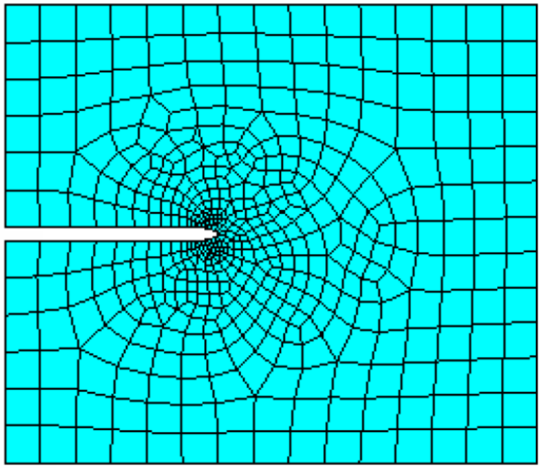
Modeling Fracture with Abaqus

- Modeling Cracks
- Calculation of Contour Integrals
- Creating an XFEM Fracture Model

Crack Modeling Overview

- A crack can be modeled as either
  - Sharp
    - Small-strain analysis
    - Singular behavior at the crack tip
      - Requires special attention
    - In Abaqus, a sharp crack is modeled using *seam* geometry
  - Blunted
    - Finite-strain analysis
    - Non-singular behavior at crack tip
    - In Abaqus, a blunted crack is modeled using *open* geometry
      - For example, a notch





Crack Modeling Overview

•The crack-tip singularity in small-strain analysis

- For mesh convergence in a small-strain analysis, the singularity at the crack tip must be considered.
  - J values are more accurate if some singularity is included in the mesh at the crack tip than if no singularity is included.
  - The stress and strain fields local to the crack tip will be modeled more accurately if singularities are considered.

•In small-strain analysis, the strain singularity is:

•Linear elasticity $\varepsilon \propto \frac{1}{\sqrt{r}}$

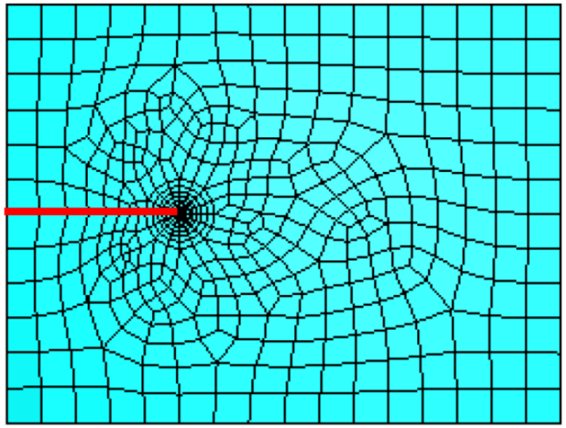
•Perfect plasticity $\varepsilon \propto \frac{1}{r}$

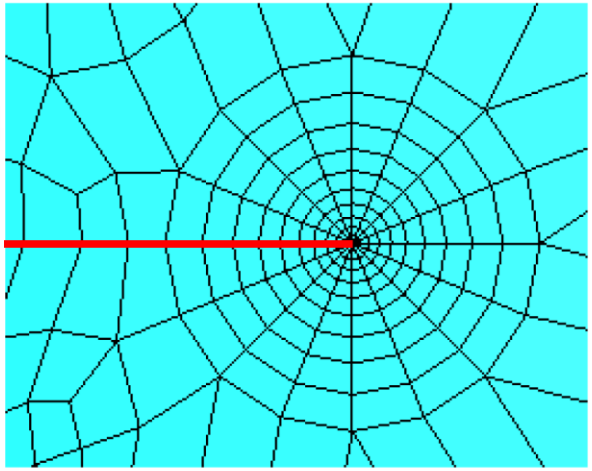
•Power-law hardening $\varepsilon \propto \frac{1}{r^{\frac{n}{n+1}}}$

Modeling Sharp Cracks in Two Dimensions

In two dimensions…

- The crack is modeled as an internal edge partition embedded (partially or wholly) inside a face.
    - This is called a *seam* crack
    - The edge along the seam will have duplicate nodes such that the elements on the opposite sides of the edge will not share nodes.
- Typically, the entire 2D part is filled with a quad or quad-dominated mesh.
    - At the crack tip, a ring of triangles are inserted along with concentric layers of structured quads.
    - All triangles in the contour domains must be represented as degenerated quads.

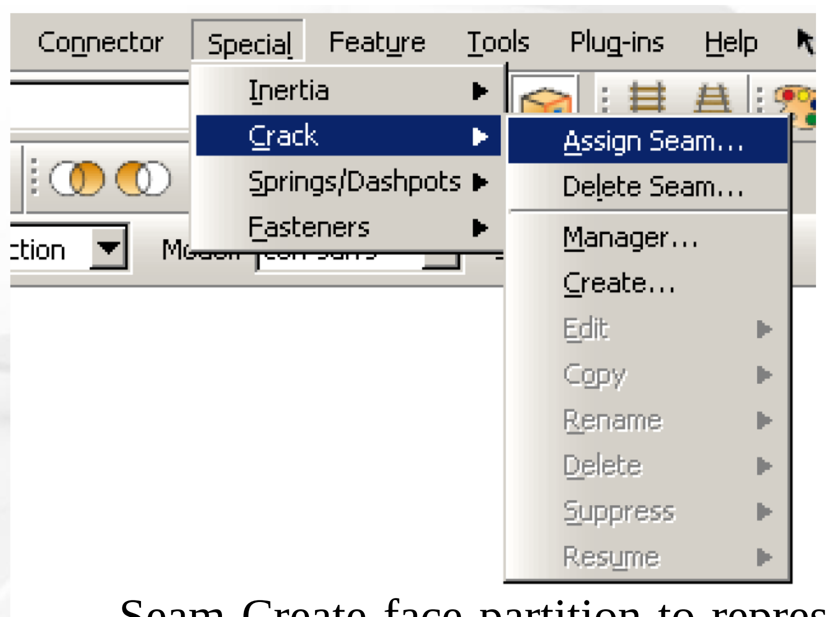




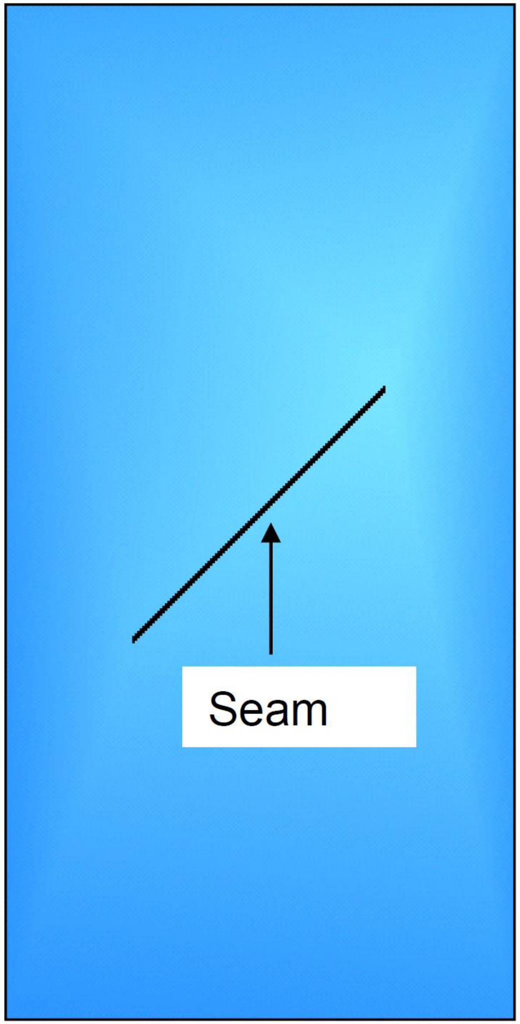
Modeling Sharp Cracks in Two Dimensions

Example: Slanted crack in a plate

- In Abaqus/CAE a seam is defined by through the **Crack** option underneath the **Special** menu of the Interaction module.
    - The seam will generate duplicate nodes along the edge.

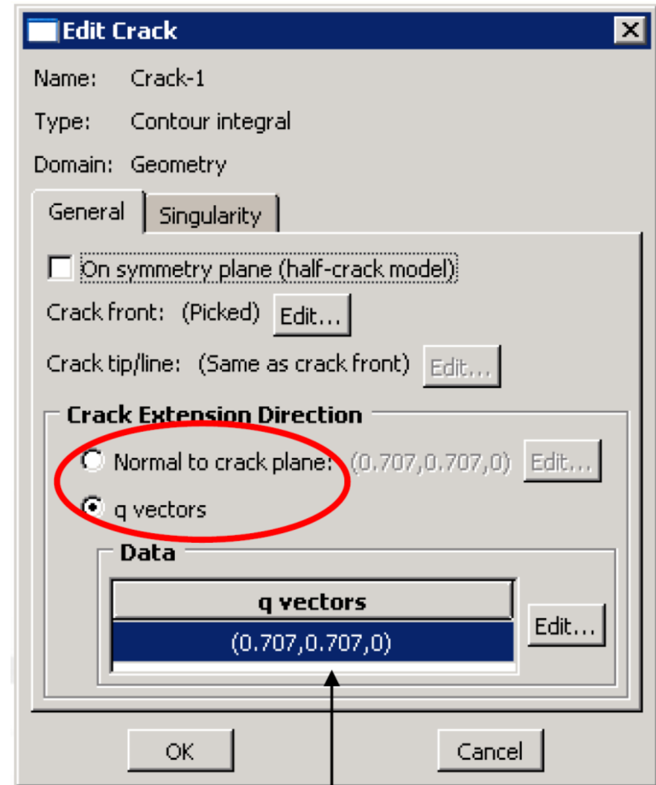


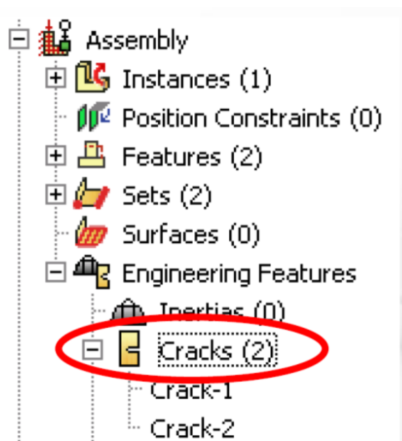
Seam Create face partition to represent the seam; assign a seam to the partition.



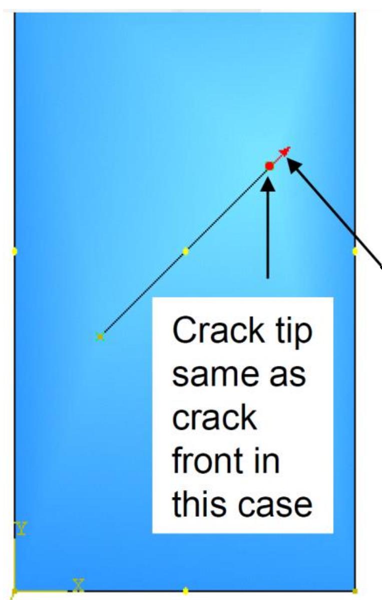
Modeling Sharp Cracks in Two Dimensions

- To define the crack, you must specify
  - Crack front and the crack-tip
  - Normal to the crack plane or the direction of crack advance
    - ❖ The crack advance direction is called the **q** vector.





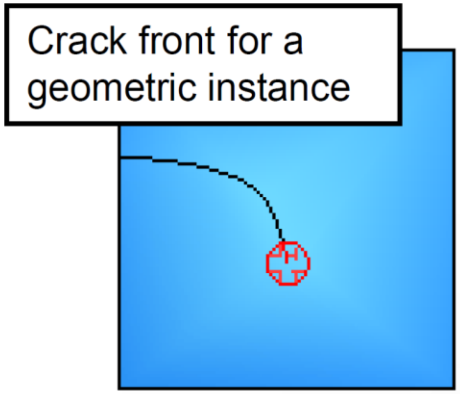
Select the vertex at either end as the crack front. (Repeat for the other end.)

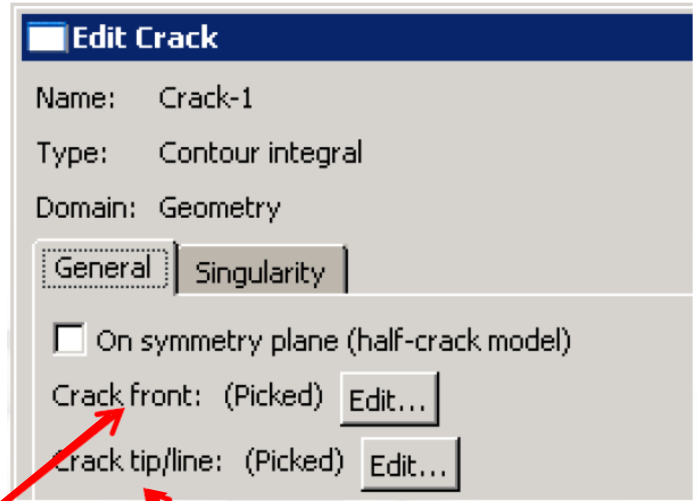


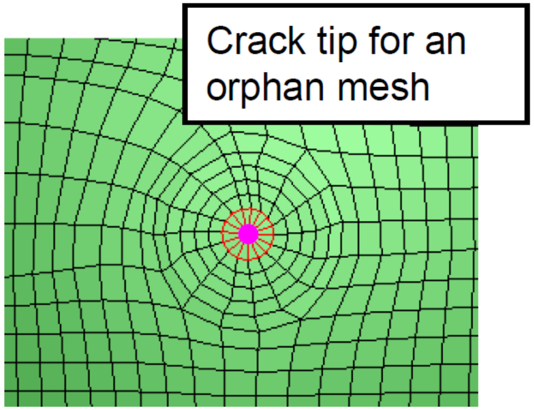
The crack extension direction (**q** vector) defines the direction in which the crack would extend if it were growing. It is used for contour integral calculations.

Modeling Sharp Cracks in Two Dimensions

- Other options for defining the crack front and crack tip







Crack front may be:

Vertex/Node

Edges/Element edges

Faces/Elements

Geometric Instances — Orphan Mesh

Crack tip may be:

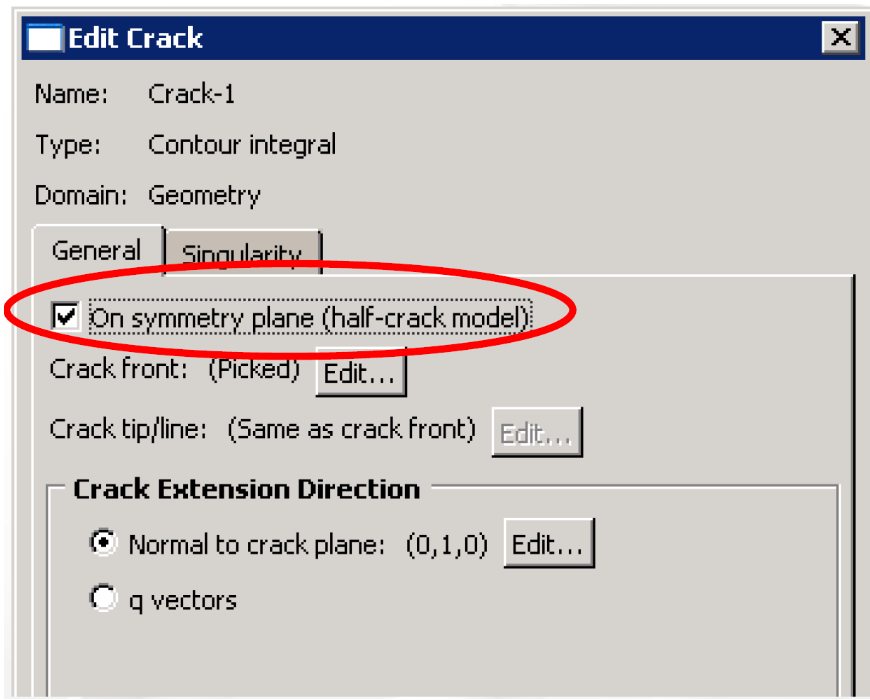
Vertex/Node

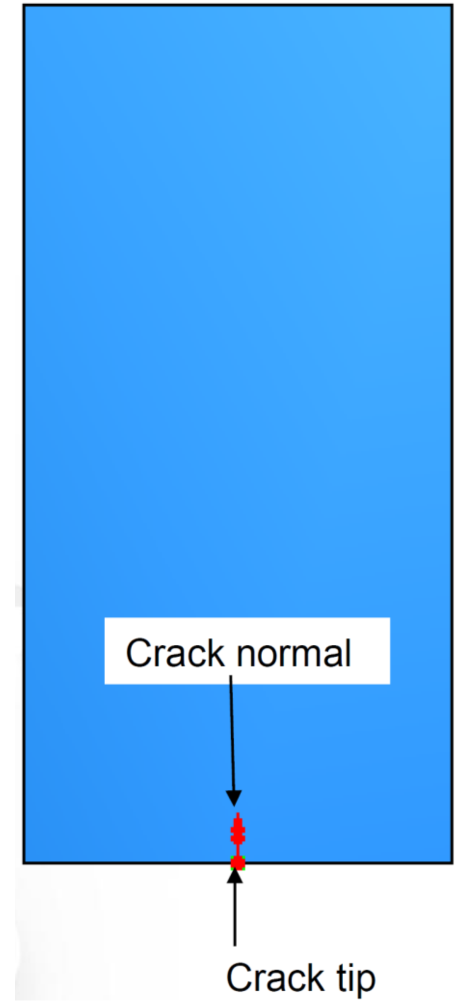
Geometric Instances — Orphan Mesh

Modeling Sharp Cracks in Two Dimensions

Example: crack on a symmetry plane

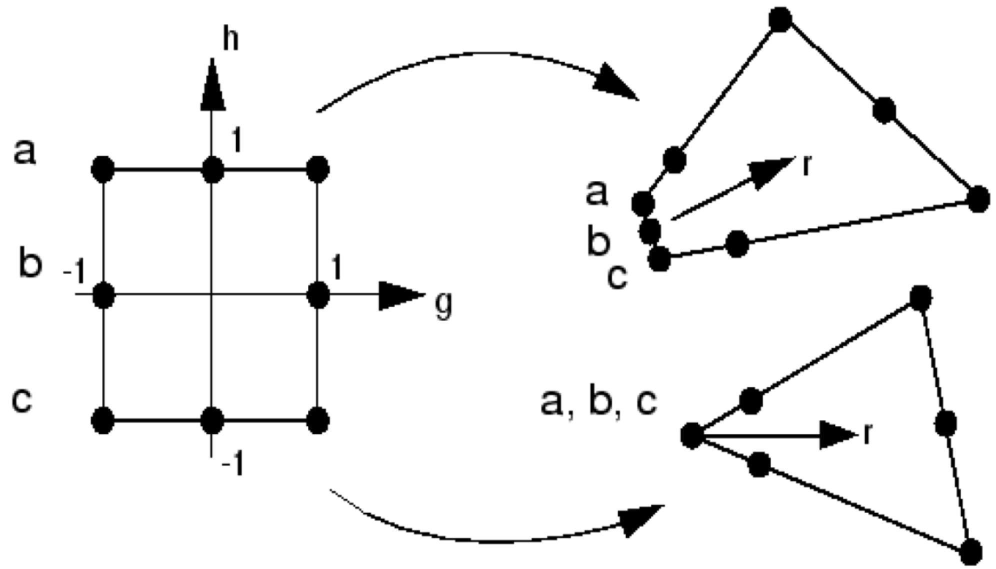
- If the crack is on a symmetry plane, you do not need to define a seam.
  - This feature can be used only for Mode I fracture.





Modeling Sharp Cracks in Two Dimensions

- Modeling the crack-tip singularity with second-order quad elements
  - To capture the singularity in an 8-node isoparametric element:
    - Collapse one side (e.g., the side made up by nodes a, b, and c) so that all three nodes have the same geometric location at the crack tip.
    - Move the midside nodes on the sides connected to the crack tip to the ¼ point nearest the crack tip.



Modeling Sharp Cracks in Two Dimensions

- If nodes *a*, *b*, and *c* are free to move independently, then

$$\varepsilon \to \frac{A}{r} + \frac{B}{\sqrt{r}} \text{ as } r \to 0$$

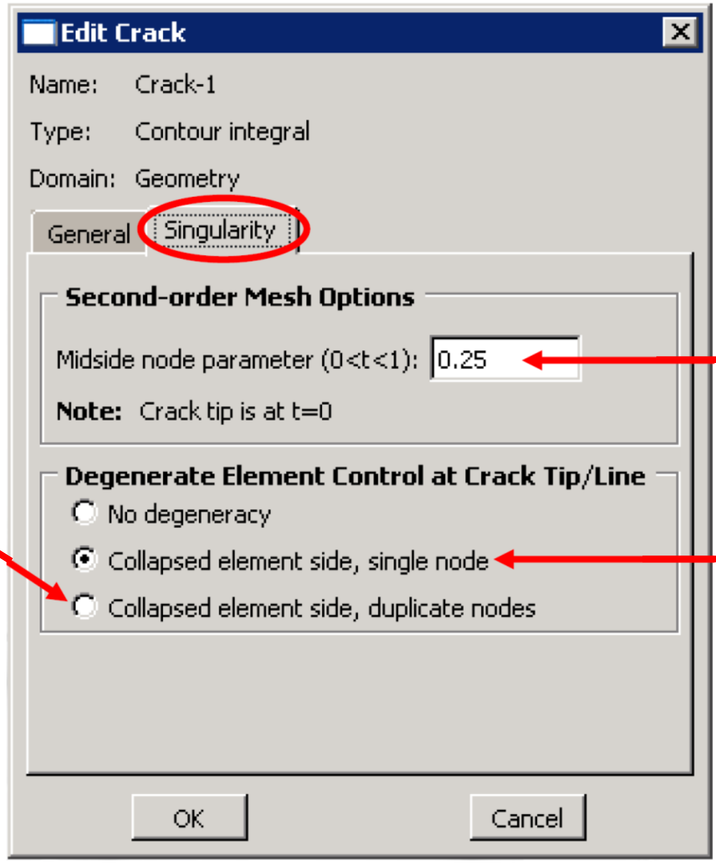
everywhere in the collapsed element.

  - If nodes *a*, *b*, and *c* are constrained to move together, $A = 0$:
    - The strains and stresses are square-root singular (suitable for linear elasticity).
  - If nodes *a*, *b*, and *c* are free to move independently and the midside nodes remain at the midsides, $B = 0$:
    - The singularity in strain is correct for the perfectly plastic case.
- For materials in between linear elastic and perfectly plastic (most metals), it is better to have a stronger singularity than necessary.
  - The numerics will force the coefficient of this singularity to be small.

Modeling Sharp Cracks in Two Dimensions

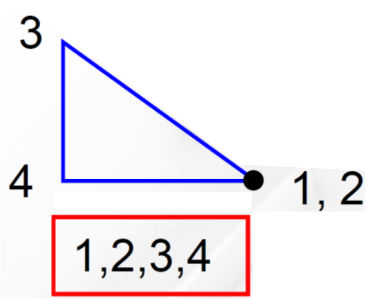
- Usage:

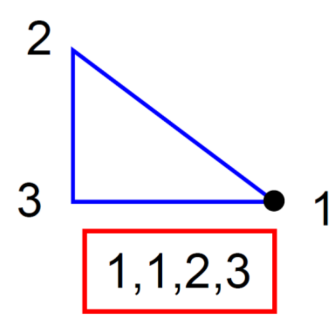
The crack tip nodes are independent: r $^{-1}$ singularity



Quarter-point midside nodes on the sides connected to the crack tip

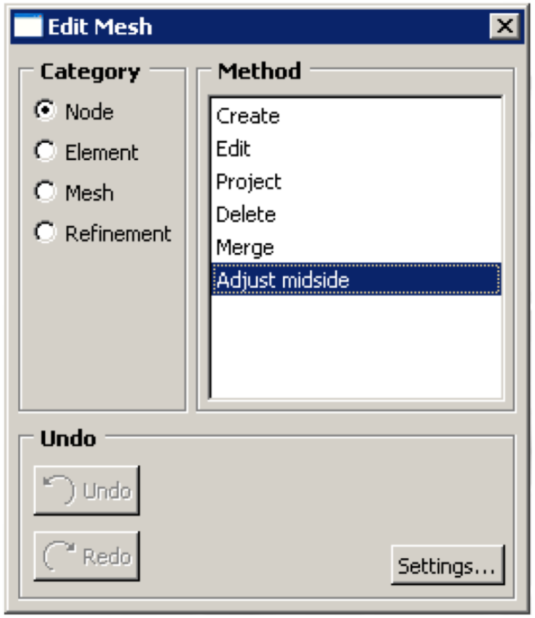
The crack tip nodes are constrained: $r^{-1/2}$ singularity

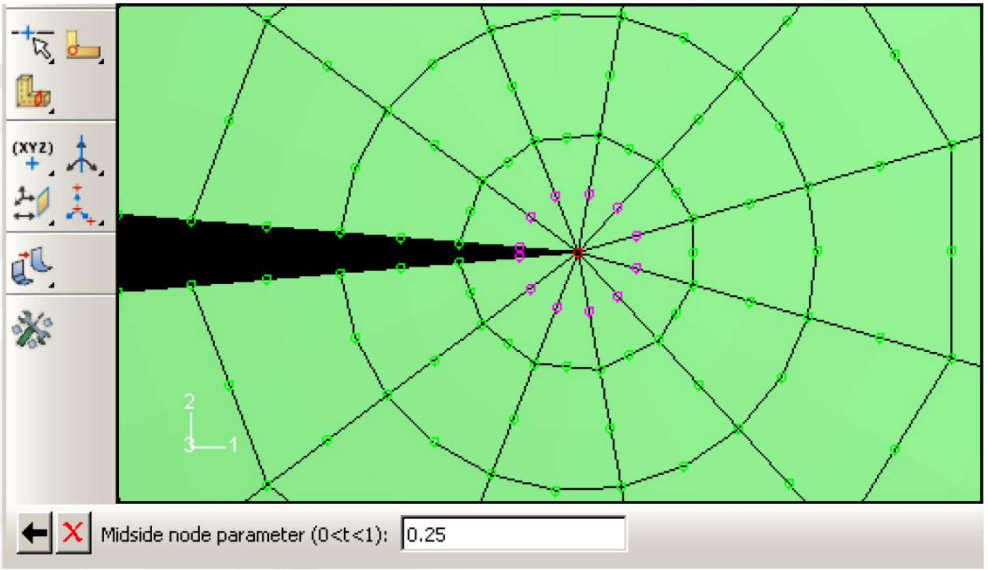




Modeling Sharp Cracks in Two Dimensions

- Aside: Controlling the position of midside nodes for orphan meshes
  - Singularity controls cannot be applied to orphan meshes.
  - Use the **Mesh Edit** tools to adjust their position.

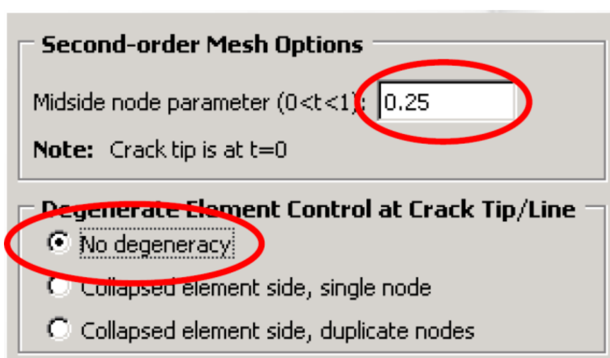


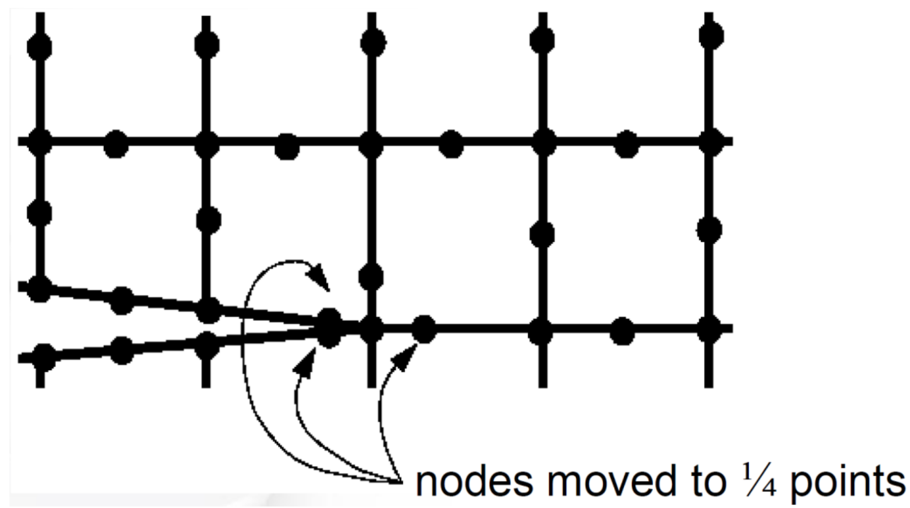


Modeling Sharp Cracks in Two Dimensions

• If the side of the element is **not collapsed** but the midside nodes on the sides of the element connected to the crack tip are moved to the ¼ point:

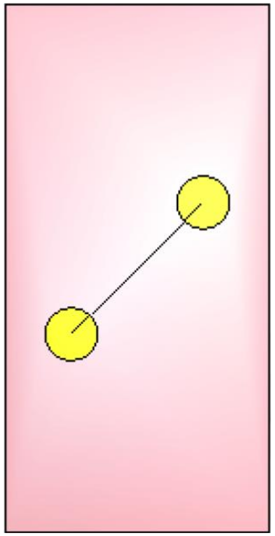
- The strain is square root singular along the element edges but not in the interior of the element.
- This is better than no singularity but not as good as the collapsed element.



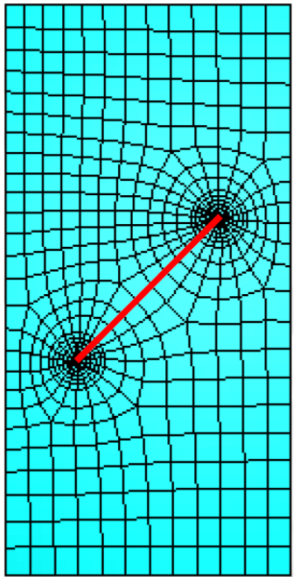


•Example: Slanted crack in a plate

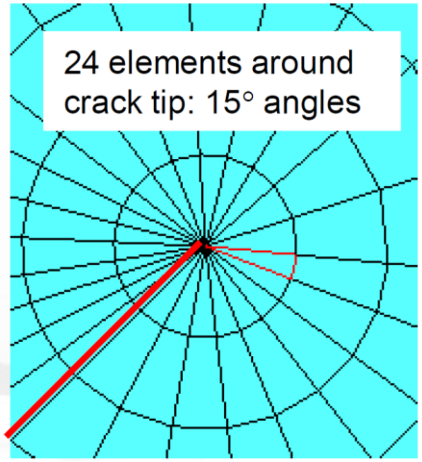
• To enable the creation of degenerate quads, you must create swept meshable regions around the crack tips (using partitions) and specify a quad-dominated mesh.

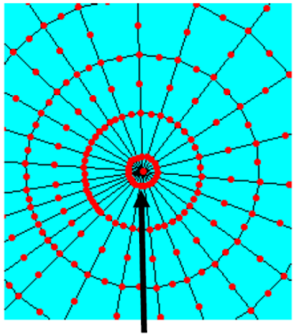


Quad-dominated mesh + swept technique for the circular regions surrounding the crack tips



Quadratic element type assigned to part





Quarter-point nodes

CPE8R elements; typical nodal connectivity shows repeated node at crack tip:

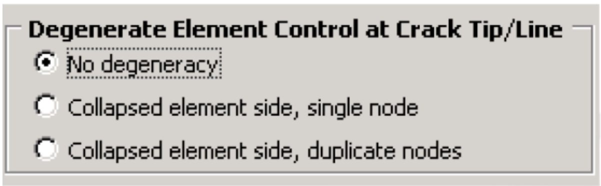
8, 8, 583, 588, 8, 1969, 1799, 1970

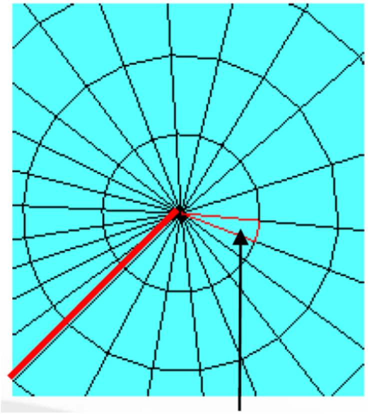
All crack-tip elements repeat node 8 in this example (nodes are constrained).

Modeling Sharp Cracks in Two Dimensions

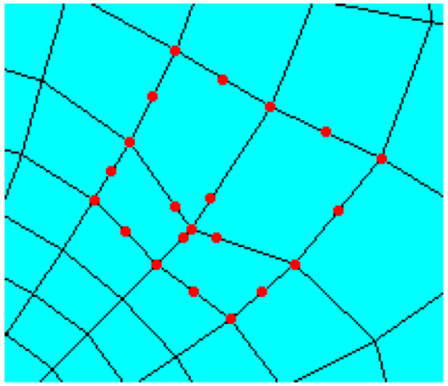
•Example: Slanted crack in a plate; Alternate meshes

- No degeneracy:



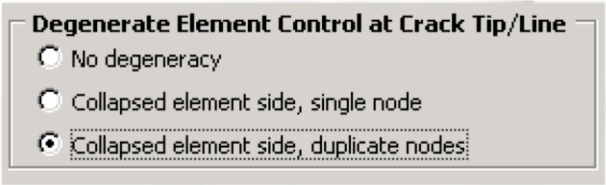


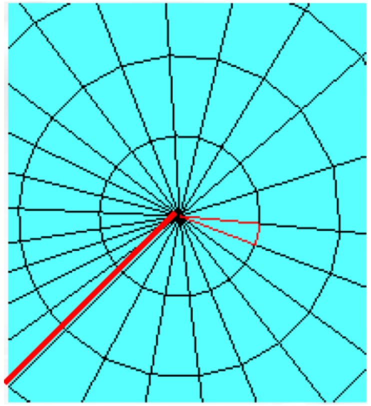
With swept meshable region: CPE6M elements at crack tip- cannot be used for fracture studies in Abaqus.



With arbitrary mesh, singularity only along edges connected to crack tip.

- Degenerate with duplicate nodes:



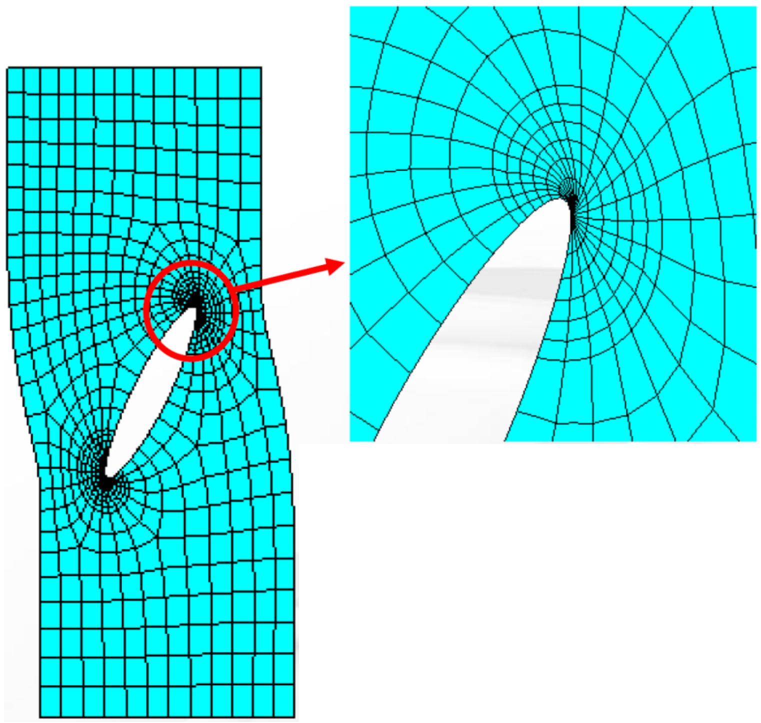


CPE8R elements at crack tip but no repeated nodes:1993, 1992, 583, 588, 2016, ...

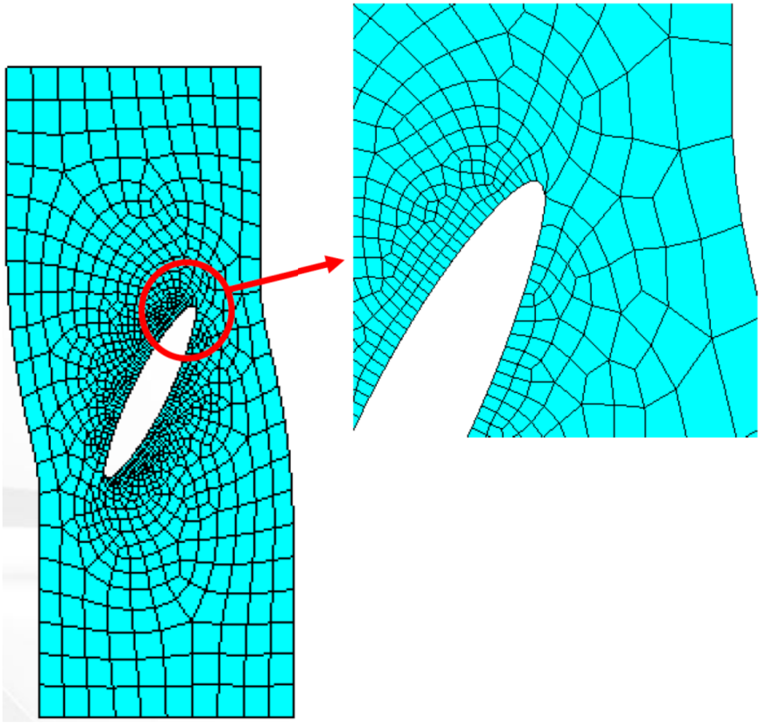
Coincident nodes located at crack tip

Modeling Sharp Cracks in Two Dimensions

•Example: Slanted crack in plate; Deformed shape



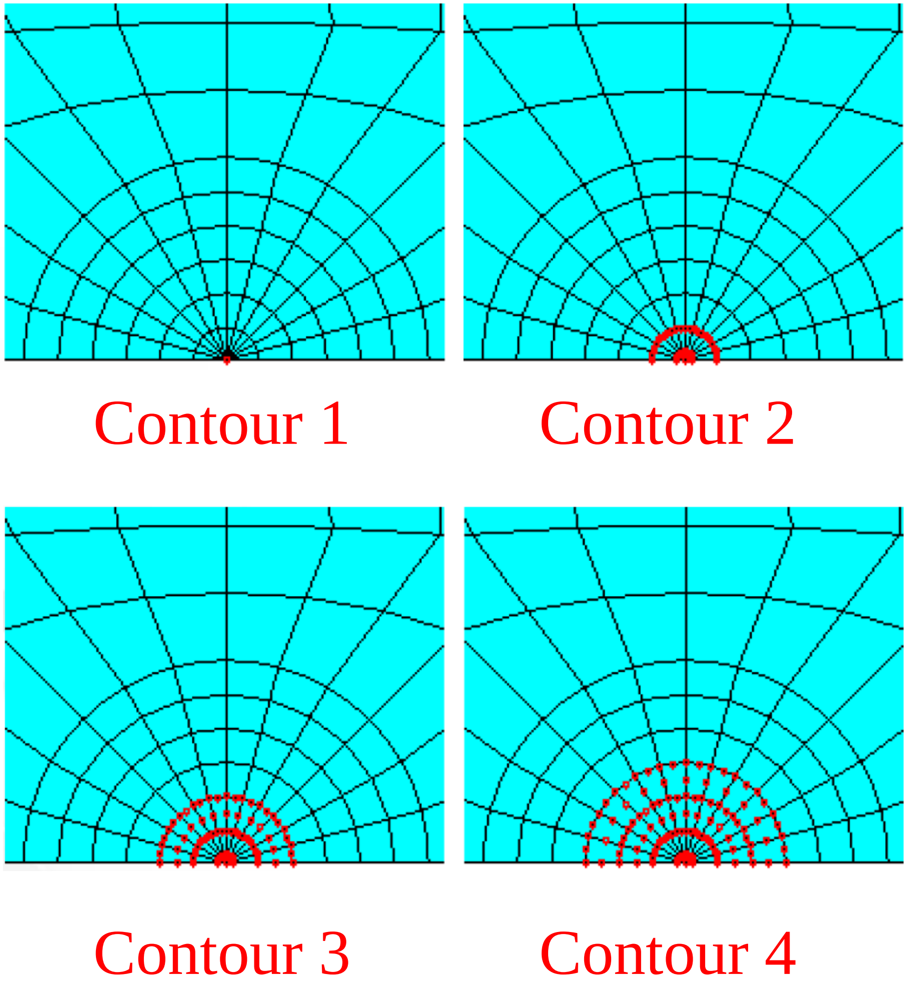
• Focused mesh



• Arbitrary mesh

- Different contours (domains) are created automatically by Abaqus.
  - The first contour consists of the crack front and one layer of elements surrounding it.

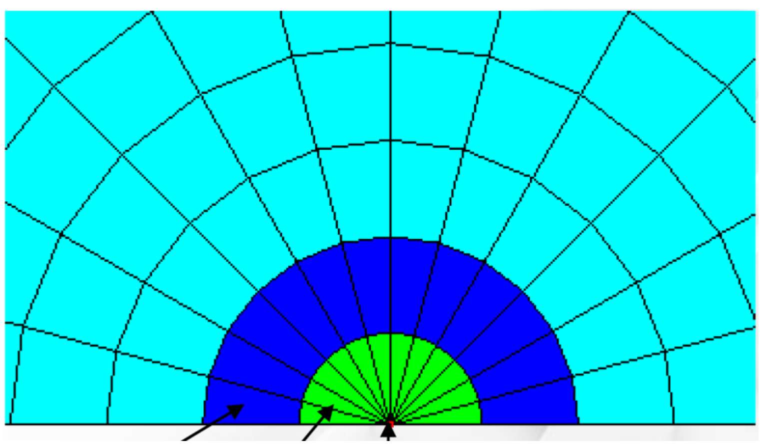
    Ring of elements from one crack surface to the other (or the symmetry plane).
  - The next contour consists of the ring of elements in contact with the first contour as well as the elements in the first contour.
  - Each subsequent contour is defined by adding the next ring of elements in contact with the previous contour.

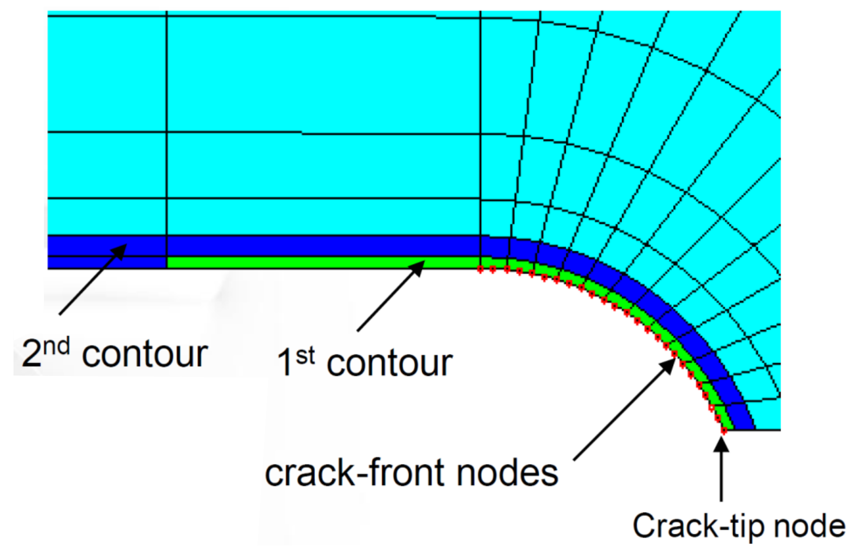


Calculation of Contour Integrals

- The J-integral and the C_t-integral at steady-state creep should be path (domain) independent.
    - The value for the first contour is generally ignored.

- Examples of contour domains:



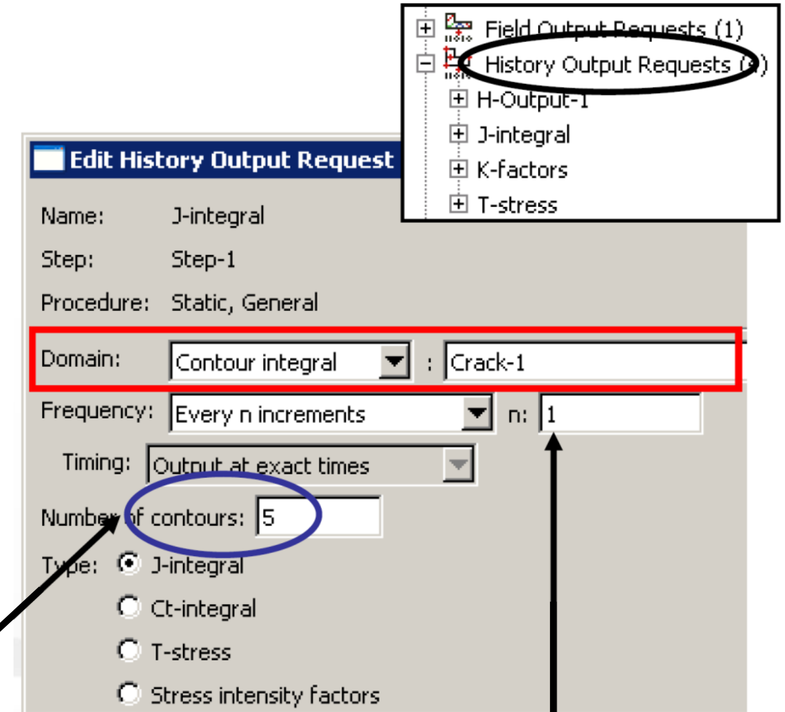


- Each subsequent contour is defined by adding the next ring of elements in contact with the previous contour.

Calculation of Contour Integrals

•Usage:

```
*CONTOUR INTEGRAL, CONTOURS=n,
TYPE={J, C, T STRESS, K FACTORS}
DIRECTION = {MTS, MERR, KII0}
```



Specifies the number of contours (domains) on which the contour integral will be calculated

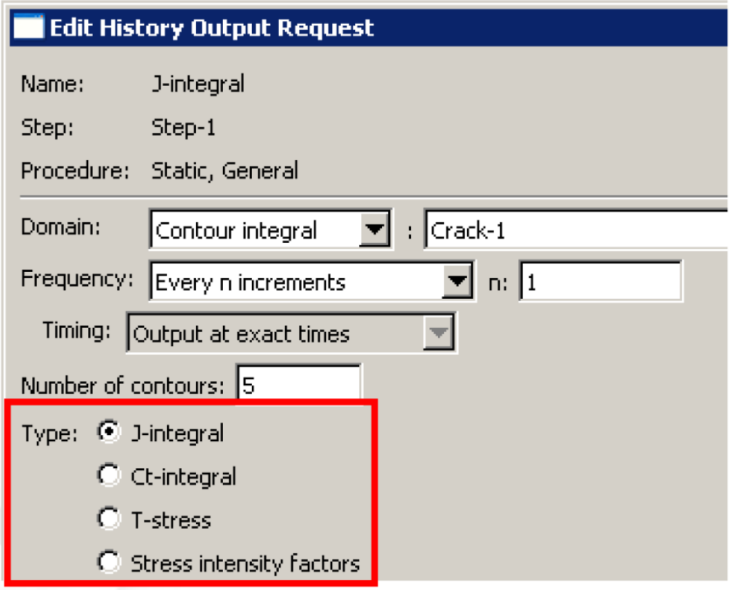
This is the output frequency in increments

• Note: In this lecture, we focus on the output-specific parameters of the *CONTOUR INTEGRAL option. The crack-specific parameters SYMM and NORMAL were discussed in the previous lecture.

Calculation of Contour Integrals

•Usage:

```
*CONTOUR INTEGRAL, CONTOURS= n,
TYPE={J, C, T STRESS, K FACTORS},
DIRECTION = {MTS, MERR, KII0}
```

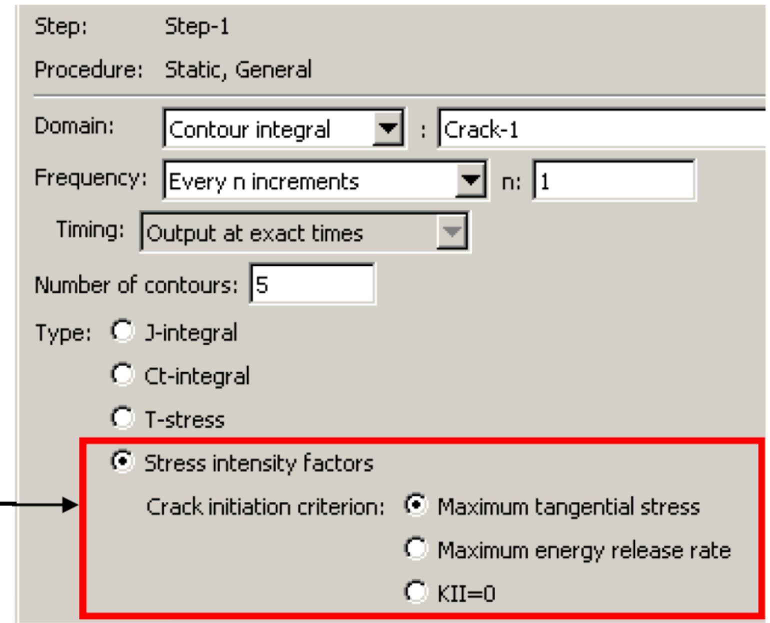


- J for J-integral output,
- C for C_t-integral output.
- T STRESS to output T-stress calculations
- K FACTORS for stress intensity factor output

Calculation of Contour Integrals

•Usage:

```
*CONTOUR INTEGRAL, CONTOURS= n,
TYPE={J, C, T STRESS, K FACTORS},
DIRECTION = {MTS, MERR, KII0}
```



Three criteria to calculate the crack propagation direction at initiation

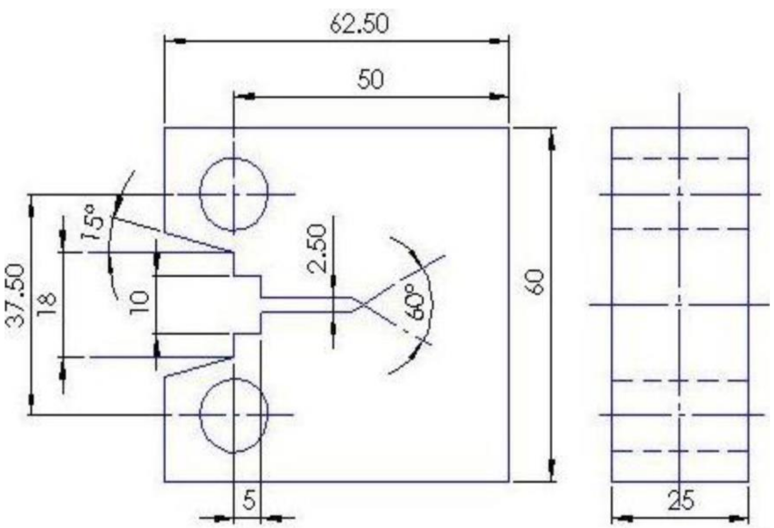
Use with TYPE=KFACTORS to specify the criterion to be used for estimating the crack propagation direction in homogenous, isotropic, linear elastic materials:

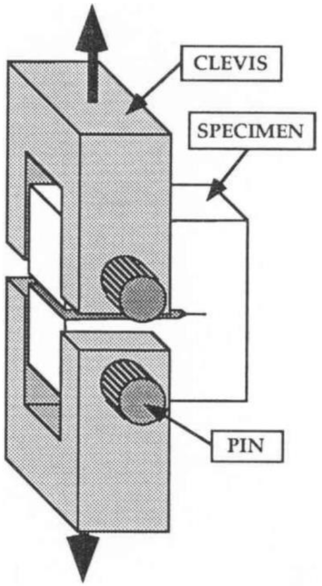
- Maximum tangential stress criterion (MTS)
- Maximum energy release rate criterion (MERR)
- K_{II}= 0 criterion (KII0)

Calculation of Contour Integrals

- Loads
  - Loads included in contour integral calculations:
    - Thermal loads.
    - Crack-face pressure and traction loads on continuum elements as well as those applied using user subroutines **DLOAD**and **UTRACLOAD**.
    - Surface traction and crack-face edge loads on shell elements as well as those applied using user subroutine **UTRACLOAD**.
    - Uniform and nonuniform body forces.
    - Centrifugal loads on continuum and shell elements.
  - Not all types of distributed loads (e.g., hydrostatic pressure and gravity loads) are included in the contour integral calculations.
    - The presence of these loads will result in a warning message.

- **Example**
  - Compact Tension Specimen
  - This is one of five standardized specimens defined by the ASTM for the characterization of fracture initiation and crack growth.
  - The ASTM standardized testing apparatus uses a clevis and a pin to hold the specimen and apply a controlled displacement.



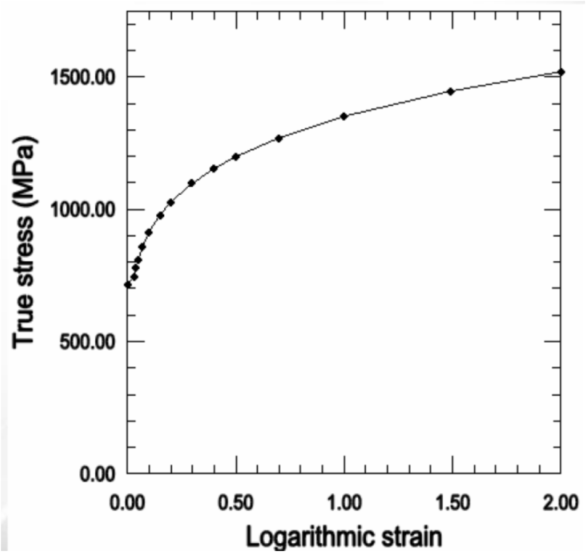


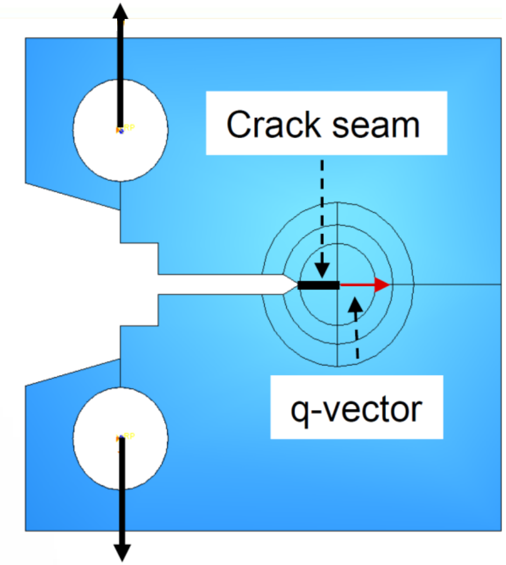
Calculation of Contour Integrals

- **Example**

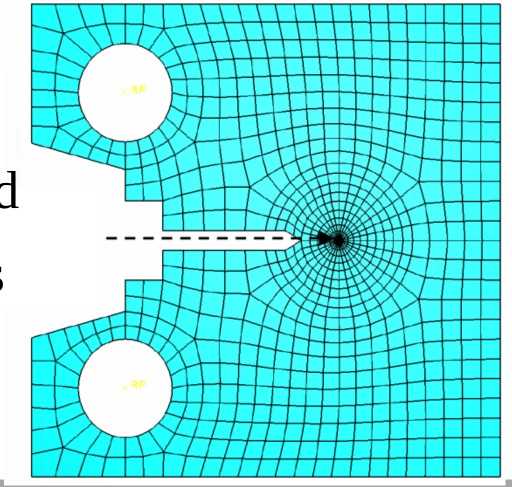
Prescribed load line displacement

- Plane strain conditions assumed.
- The initial crack length is 5 mm.
- Elastic-plastic material



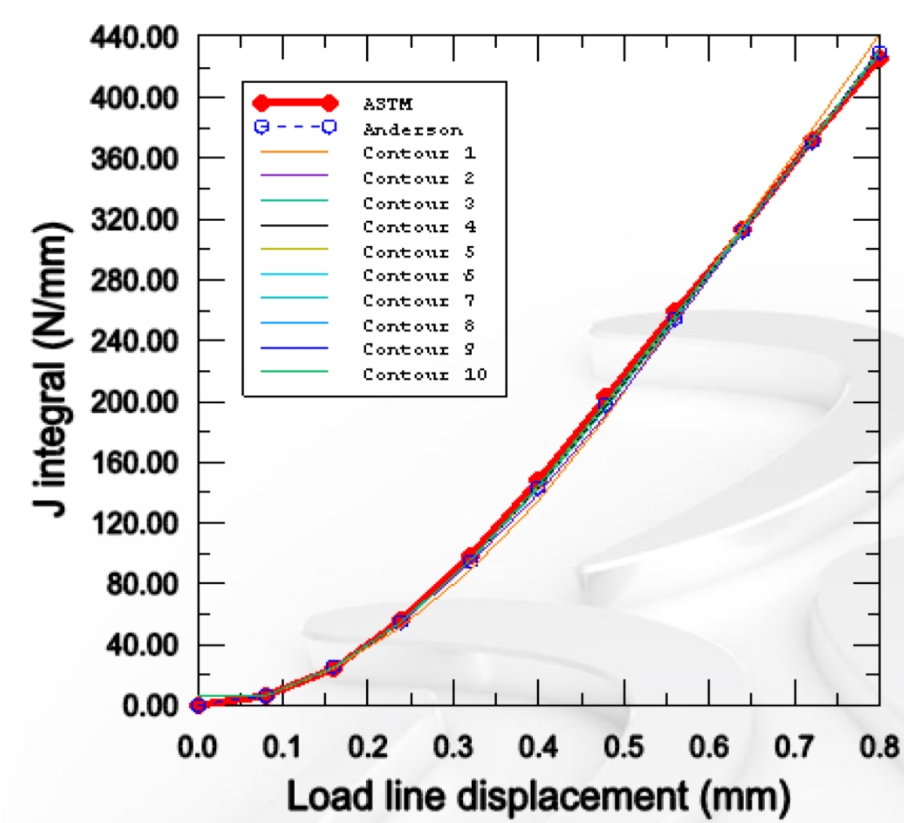


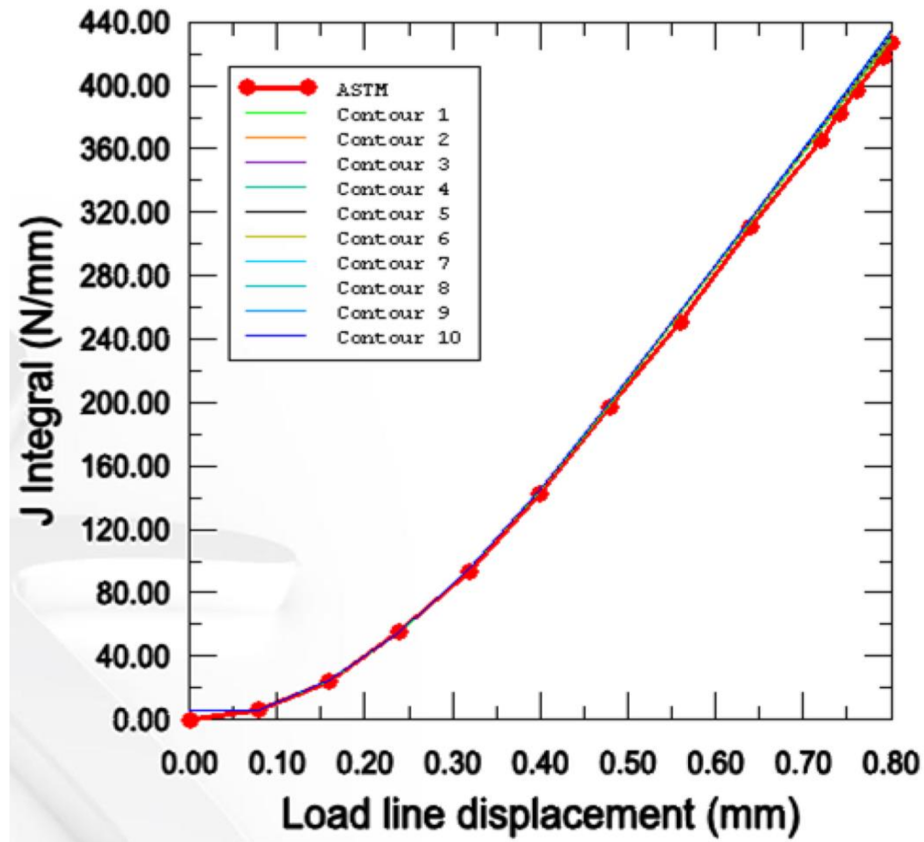
$1/\sqrt{r}$ singularity modeled in the crack-tip elements



Calculation of Contour Integrals

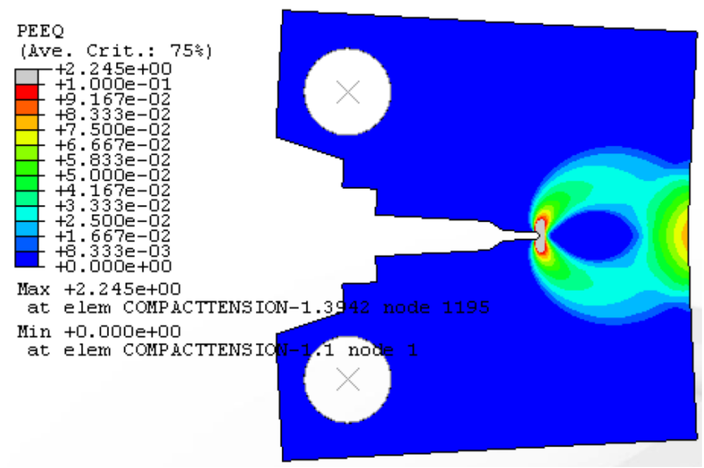
- Results

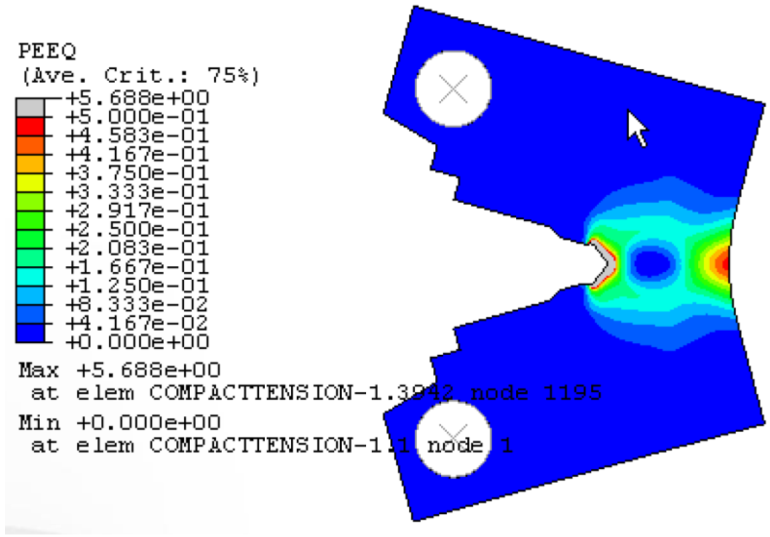




- Small strain analysis
- Finite strain analysis

- Results





- At small to moderate strain levels, the small and finite strain models yield similar results.

- Finite strain effects must be considered to represent this level of deformation and strain accurately.

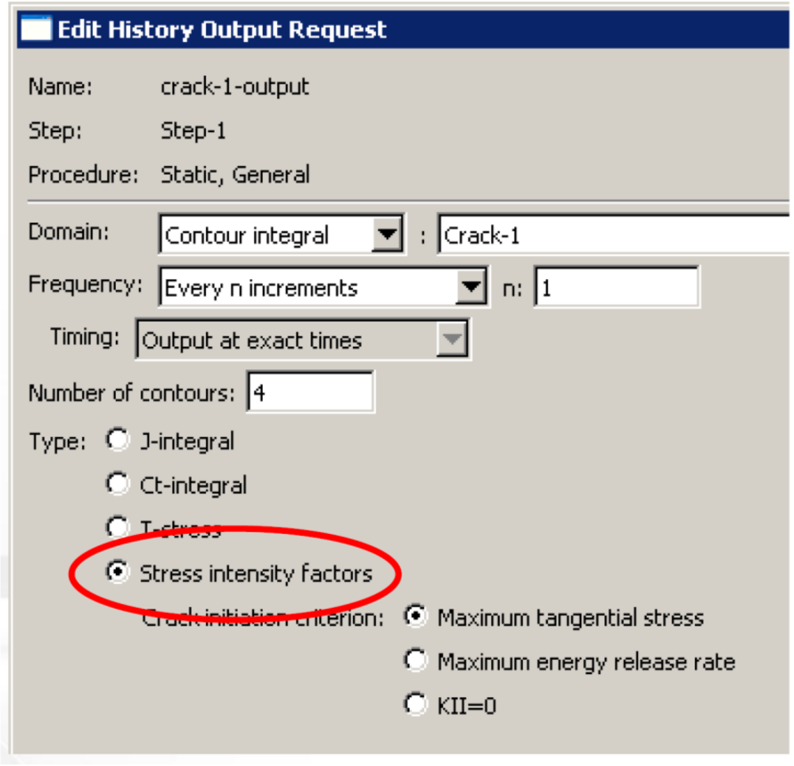
Mixed-Mode Fracture

- Abaqus uses interaction integrals to compute the stress intensity factors.

- This approach accounts for mixed-mode loading effects.
- Note that the J-or C_t-integrals do not distinguish between modes of loading.
- Usage:

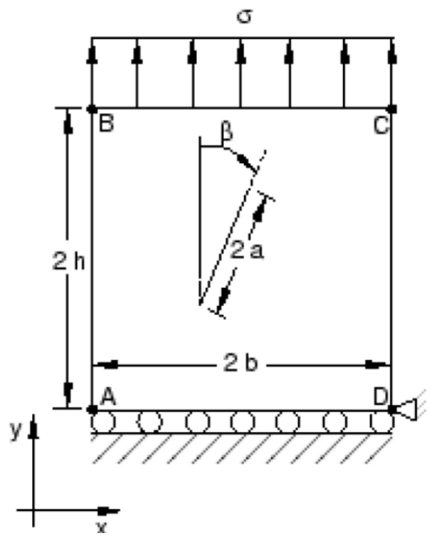
  *CONTOUR INTEGRAL,

  TYPE=K FACTORS
- Stress intensity factors can only be calculated for linear elastic materials.



Mixed-Mode Fracture

- Example: Center slant cracked plate under tension



$\beta = 22.5°, 67.5°$

$a/b = 0.5$

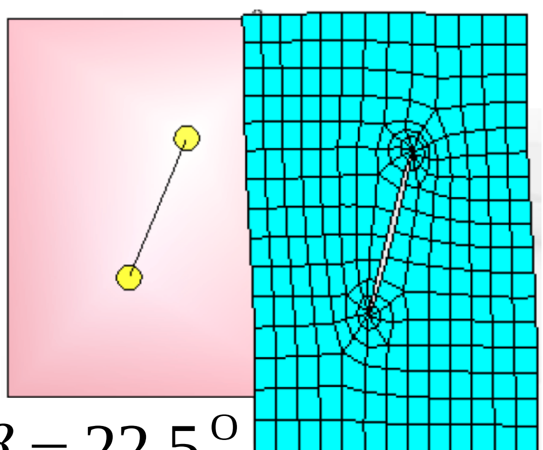
$h/b = 1.25$

$b = 50.0$ mm

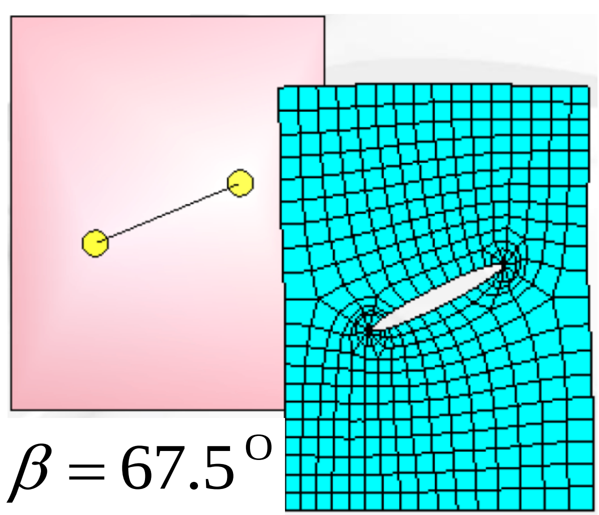
β	Element type	$\frac{K_I}{K_0}$	$\frac{K_{II}}{K_0}$
22.5°	CPE8	0.185 (–2.9%)*	0.403 (–0.2%)
22.5°	CPE8R	0.185 (–2.9%)	0.403 (–0.2%)
67.5°	CPE8	1.052 (+3.6%)	0.373 (+1.0%)
67.5°	CPE8R	1.053 (+3.8%)	0.374 (+1.3%)

$$K_0 = \sigma\sqrt{\pi a}$$

*Values enclosed in parentheses are percentage differences with respect to the reference solution. See Abaqus Benchmark Problem 4.7.4 for more information.



$\beta = 22.5^{O}$



$\beta = 67.5^{O}$