



ABAQUS

6.1 Debonding

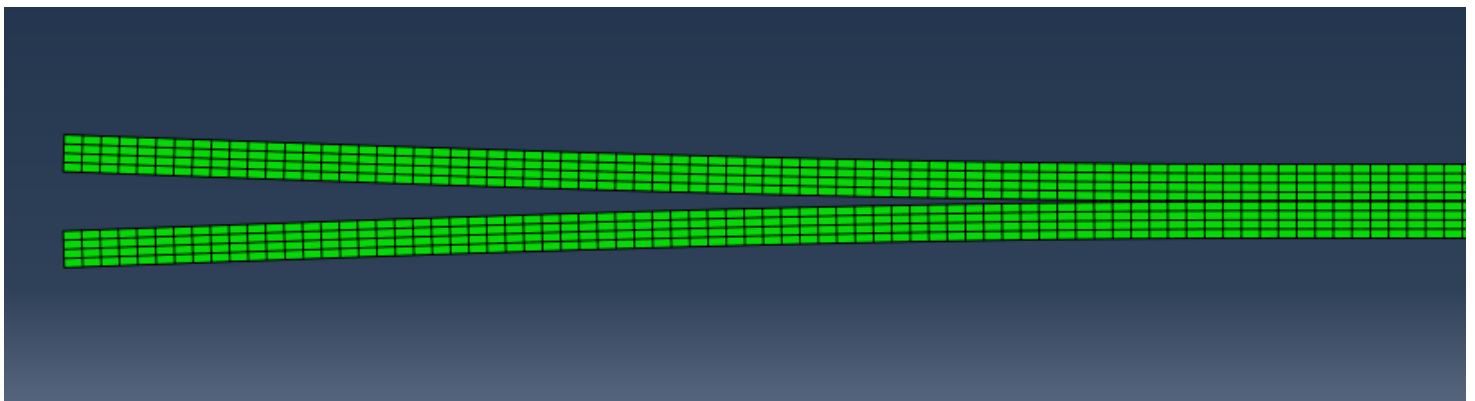
Contents

1. [Exercise 6.1: Debonding of Double Cantilever Beam](#)
2. [Credits](#)


Exercise 6.1: Debonding of Double Cantilever Beam

Problem Description

Use displacement control to conduct a double cantilever beam fracture experiment simulation and then plot the traction-separation curve.



1. First, create two parts.


1. Create 2D Planar Deformable Shell Part, named Bot. In the Sketch, draw a rectangle by using the *Create rectangle* tool . The corners should be $(0, 0)$, $(9, 0)$, $(9, 0.402)$, and $(0, 0.402)$.
2. The other part, named Top, should have corners at $(0, 0.202)$, $(9, 0.402)$.

[Data files](#) are available.

Assign Property


1. Create material named metal of type linear elastic with properties $E = 8e6$, $\nu = 0.3$.
2. Create a Solid Homogeneous section named MetalSection using the metal material.
3. Assign the section to both parts Top and Bot.

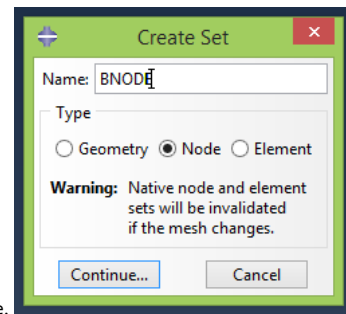
Assembly

1. Since we have already given proper coordinates for each part, we can instance both of them without further configuration. Create instance  using the two beam

Mesh and Create Sets

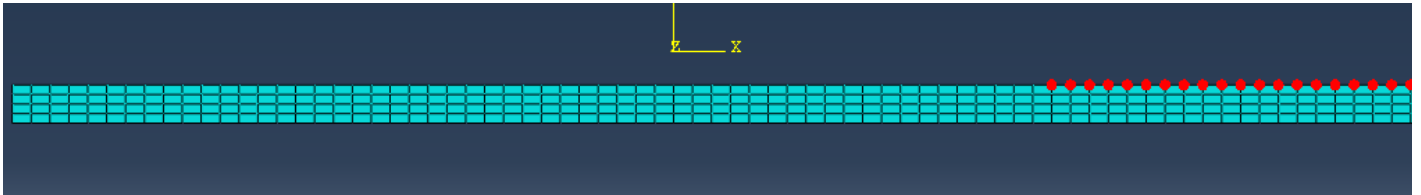
1. Assign the element type Plane Strain and toggle off Reduced integration (CPE4); the mesh control should be Quad/ Structured mesh.

2. Use *Display Group*  to see only bottom beam. Seed the top and bottom edge with 90 elements, while the two sides have 4 elements; then c multiply select.)



3. Create a set named **BNODES** from *Tools*→*Sets*→*Create* or you can create set from model tree. Type **Node**.

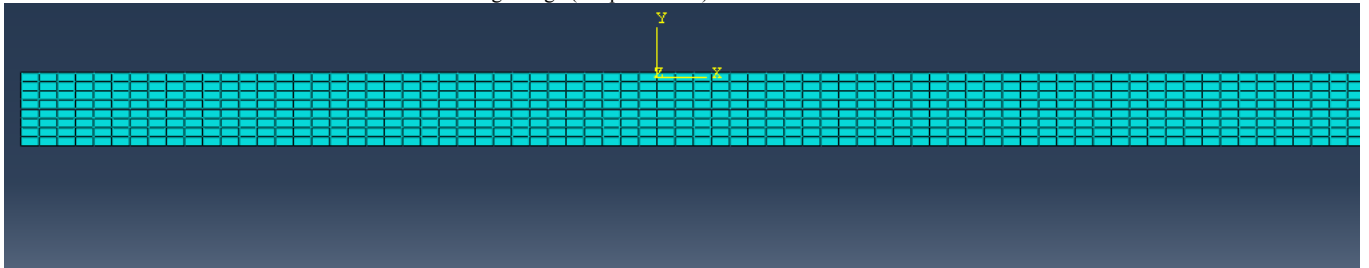
Select some nodes (half) along top edge from the right (see below). (**Note:** This node will not break in the simulation, and is the reference node sets.)




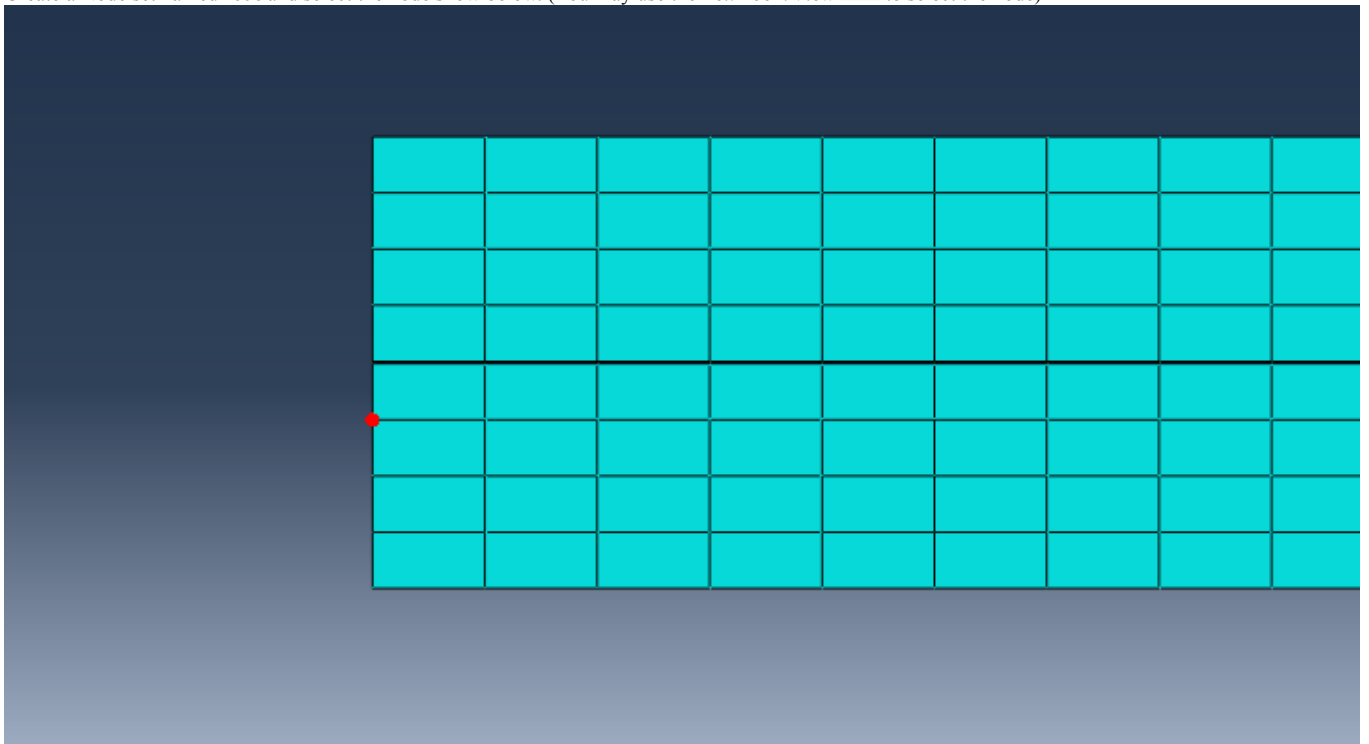
4. Now you can mesh the topbeam. Do the same as in step 2.

5. Create a few node sets for load definition later.

1. Create a Node set named **fixed**. Select the nodes at the right edge (see pics below).

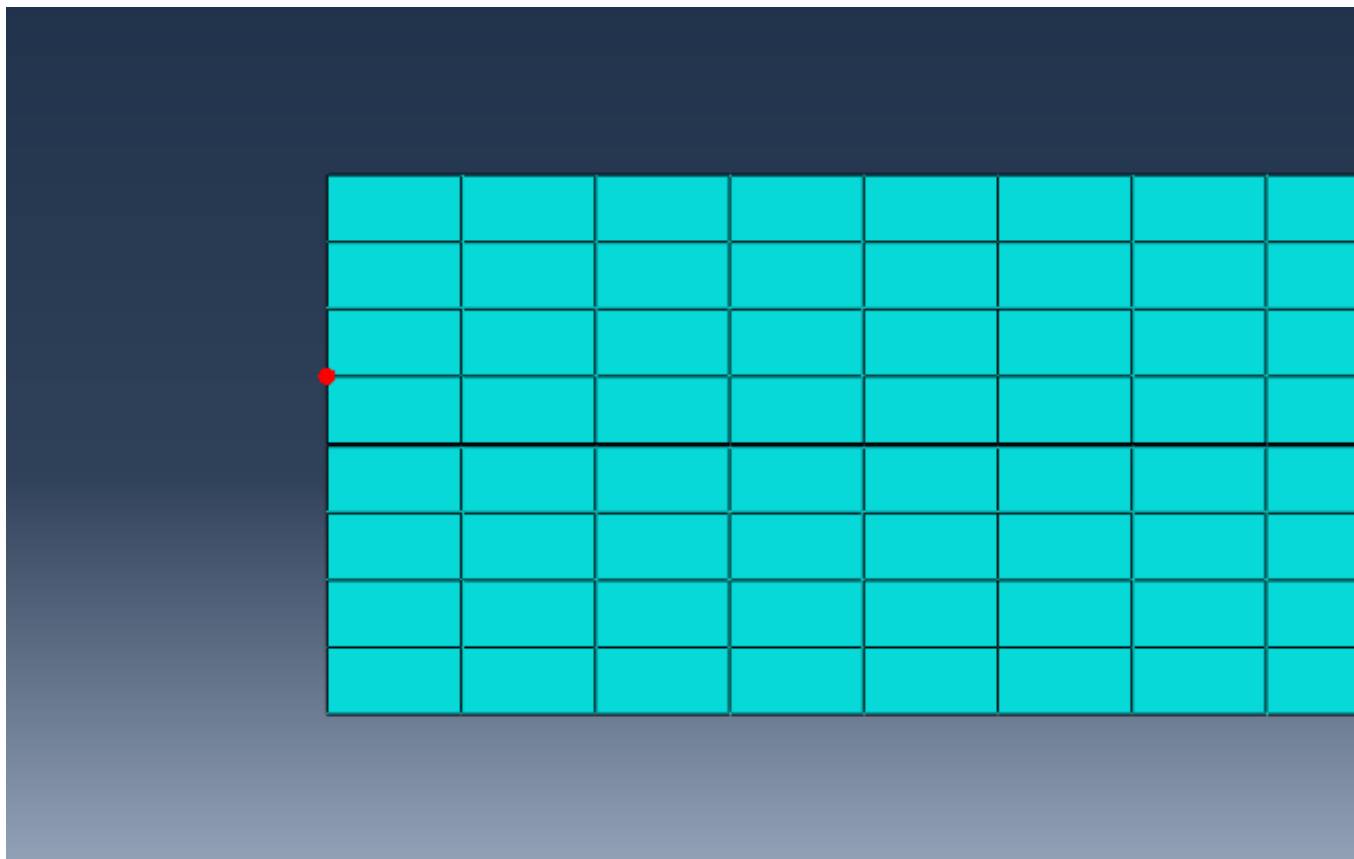


2. Create a Node set named **Dbot** and select the node show below. (You may use the *Box Zoom View*  to select the node)




3. Create a Node set named **Dtop** and select the node in the same position of Dbot at top beam (see below)

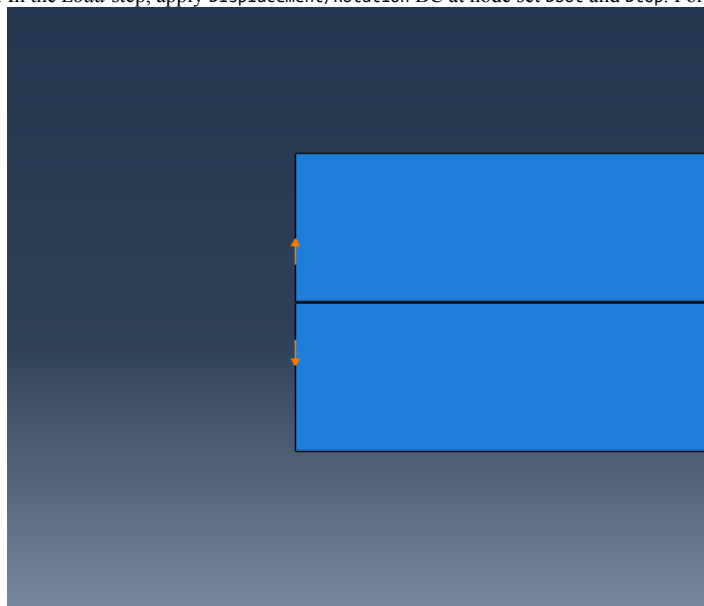
(Dbot and Dtop are the place where we will apply the displacement BC.)



Apply load/displacement and BC

Here, we apply load/BC directly on the node, since that is actually how the finite element method works.

1. Create a General, Static step named Load, Toggle on Nlgeom, Maximum number of increment: 5000; Initial increment size: 0.005, maximum increment size: 0.015.
2. In the *Initial* step, apply a Displacement/Rotation BC on node set fix, toggling on u1 and u2. (**Note:** the tool is *Select node set* .)
3. In the *Load* step, apply Displacement/Rotation BC at node set Dbot and Dtop. For the Dbot, set u2 to be -0.16 and for the Dtop set u2 to be 0.16.



Define the interaction

1. Go to the *Interaction* module and select *Create Interaction Property* . Name it cohesive and select the contact type.

2. In the *Edit Contact Property* dialog box, Choose *Mechanical Cohesive Behavior*. Toggle on *Specify the bonding nodes set in the Surface-to-surface Std interaction*

Name: Cohesive

Contact Property Options

Cohesive Behavior

Mechanical Thermal Electrical

Cohesive Behavior

☒ Allow cohesive behavior during repeated post-failure contacts

Eligible Slave Nodes

☐ Any slave nodes experiencing contact

☐ Only slave nodes initially in contact

☒ Specify the bonding node set in the Surface-to-surface Std interaction

Traction-separation Behavior

☐ Use default contact enforcement method

☒ Specify stiffness coefficients

☒ Uncoupled ☐ Coupled

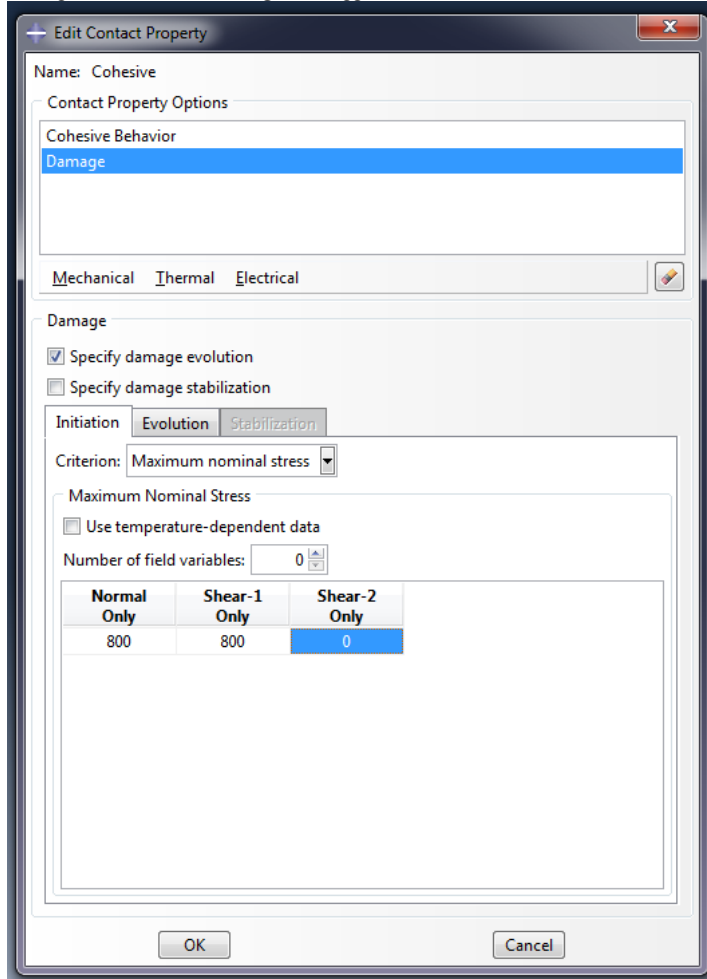
☐ Use temperature-dependent data

Number of field variables: 0

Knn	Kss	Ktt

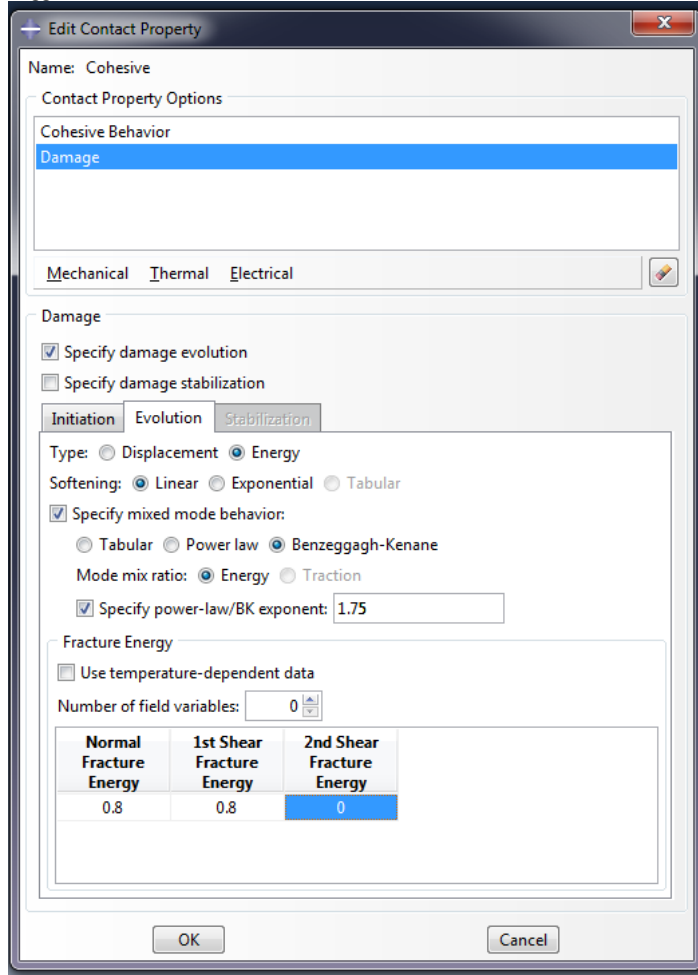
OK Cancel

3. Then go to *Mechanical Damage* and toggle on *Specify damage evolution*. In *Initiation*, input 800 to *Normal only* & *Shear-1 only*; input 0 in *Shear-2*.



In *Evolution*, toggle on type *Energy* with *Linear softening*; also select *Specify mixed mode behavior* of type *Benzeggagh-Kenane*.


Toggle on Specify power-law/BK exponent with a value of 1.75. For the Normal Fracture Energy, input 0.8; for the 1st Shear Fracture Energy, input 0.8; 2nd Shear

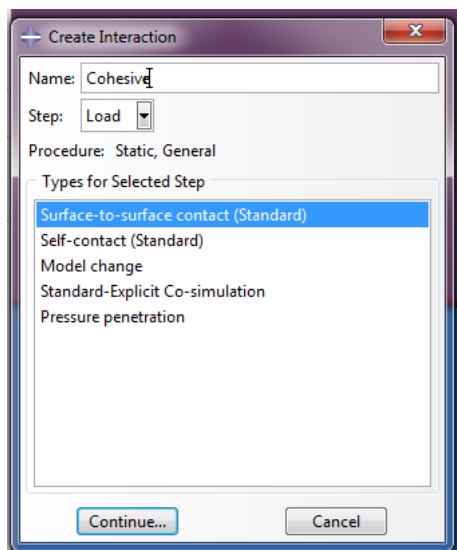


4. Finally select Mechanical→Geometric→Properties. Before we create interaction, we should define the two interaction surface, which is also the crack surface. Go to τ



Name the top surface TopSurf and the bottom surface BotSurf. (You may need to zoom in to select both surfaces.)

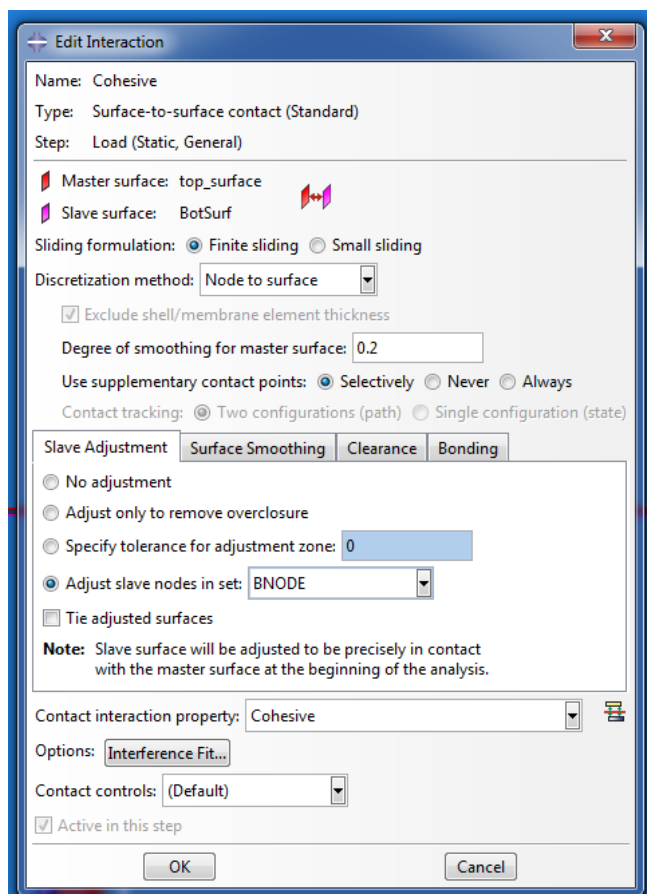
5. Use the *Create Interaction* tool  to create a Cohesive interaction of type Surface-to-surface contact (Standard) with TopSurf selected as the master surface (since the bottom beam).



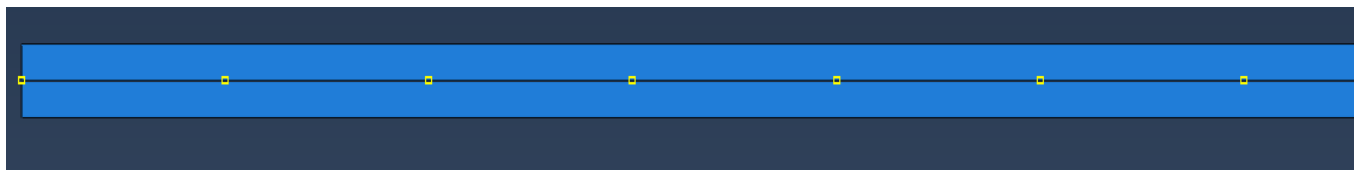
1. Now select the Slave type Surface for BotSurf.



2. In the *Edit Interaction* dialog box, select Node to Surface in the *Discretization method*



3. Toggle on Adjust slave nodes in set, select BNODE set.
4. Choose *Cohesive* in the Contact interaction property.
5. After the Interaction definition, you can see a yellow square along the interface.



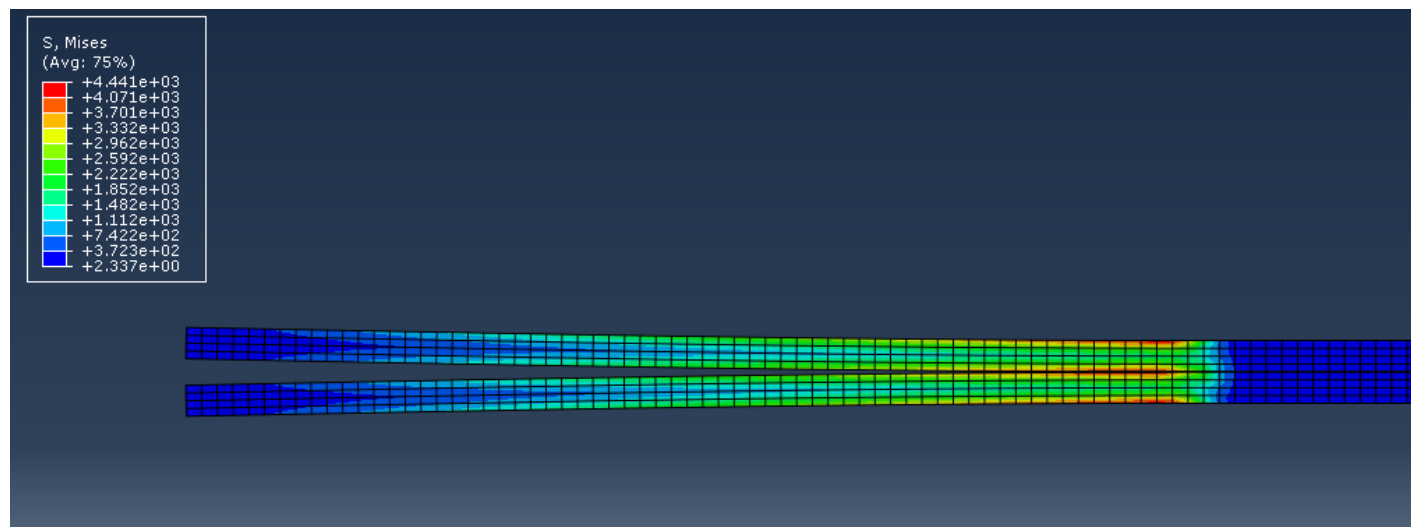
Specify the History output

Usually in the double cantilever beam experiment, we would like to get the traction-separation figure at the crack front. Go to **History Output** from the mode tree & `Dtop`. Then we have the displacement vs reaction force relation at point `Dtop`.

1. Toggle on Displacement/U/U2 and Forces/RF/RF2.

Create and submit jobs.

1. Create the job named DCB, save it, and submit it.



2. Observe the stress distribution around the double cantilever beam. As well, you can draw the traction-separation plot by using the history output created.

Credits

Neal Davis, Ruizhi Li, and Binyue Hou developed these materials for [Computational Science and Engineering](#) at the University of Illinois at Urbana–Champaign.



This content is available under a [Creative Commons Attribution-NonCommercial 4.0 Unported License](#).

Computational S
and Engine

CS