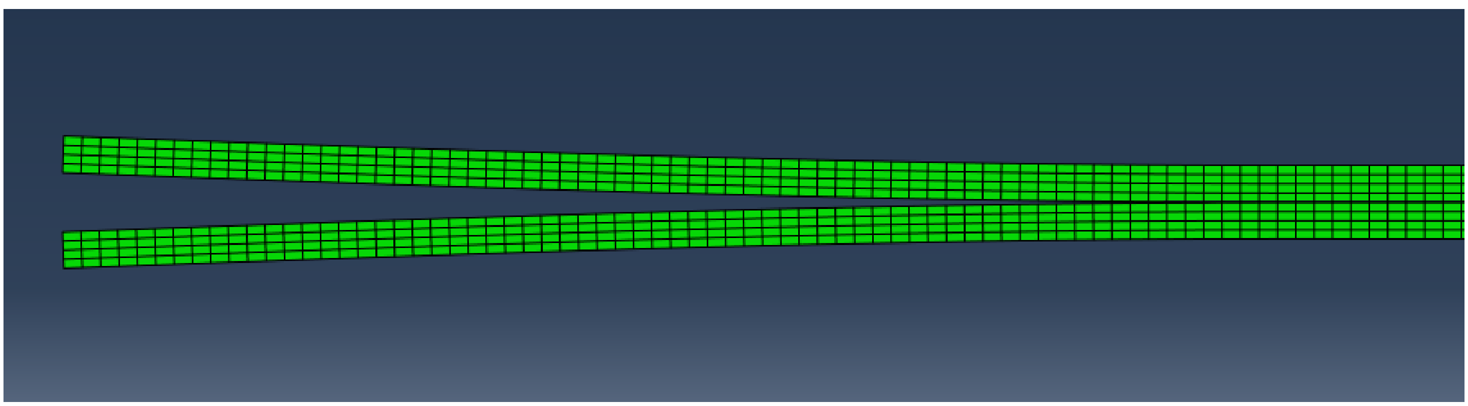
6.1 Debonding

Contents

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
    `0), (9, 0.2)`.

    2. The other part, named `Top`, should have corners at `(0, 0.202)`, `(9, 0.402)`.

    Data files are available.

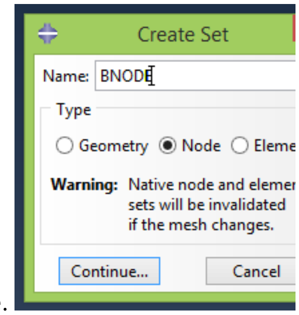
Assign Property¶

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¶

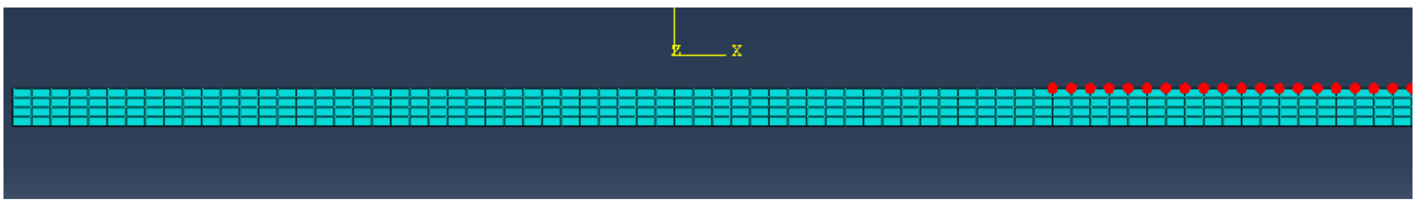
1. Since we have already given proper coordinates for each part, we can instance both of them without further configuration. `Create` beam parts with an `Independent` mesh.

Mesh and Create Sets¶

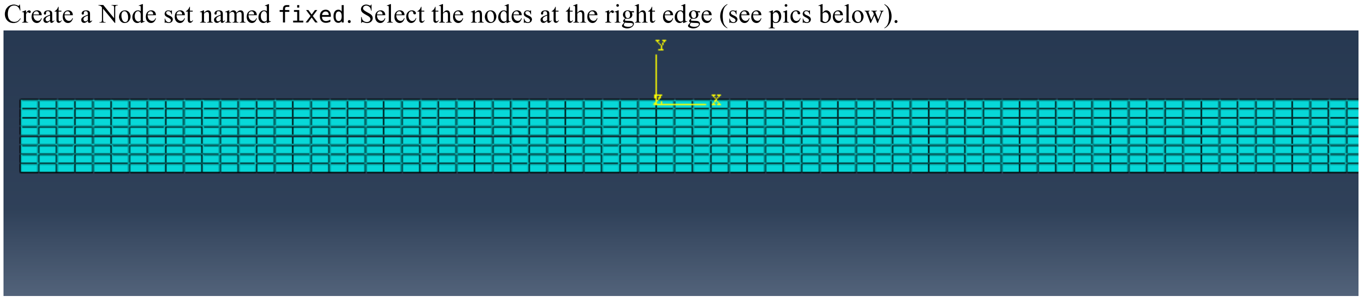
1. Assign the element type `Plane Strain` and toggle off `Reduced integration (CPE4)`; the mesh control should be `Quad/ Struc`
2. Use *Display Group*  to see only bottom beam. Seed the top and bottom edge with `90` elements, while the two s create the mesh. (Hold *Shift* to multiply select.)
3. 

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

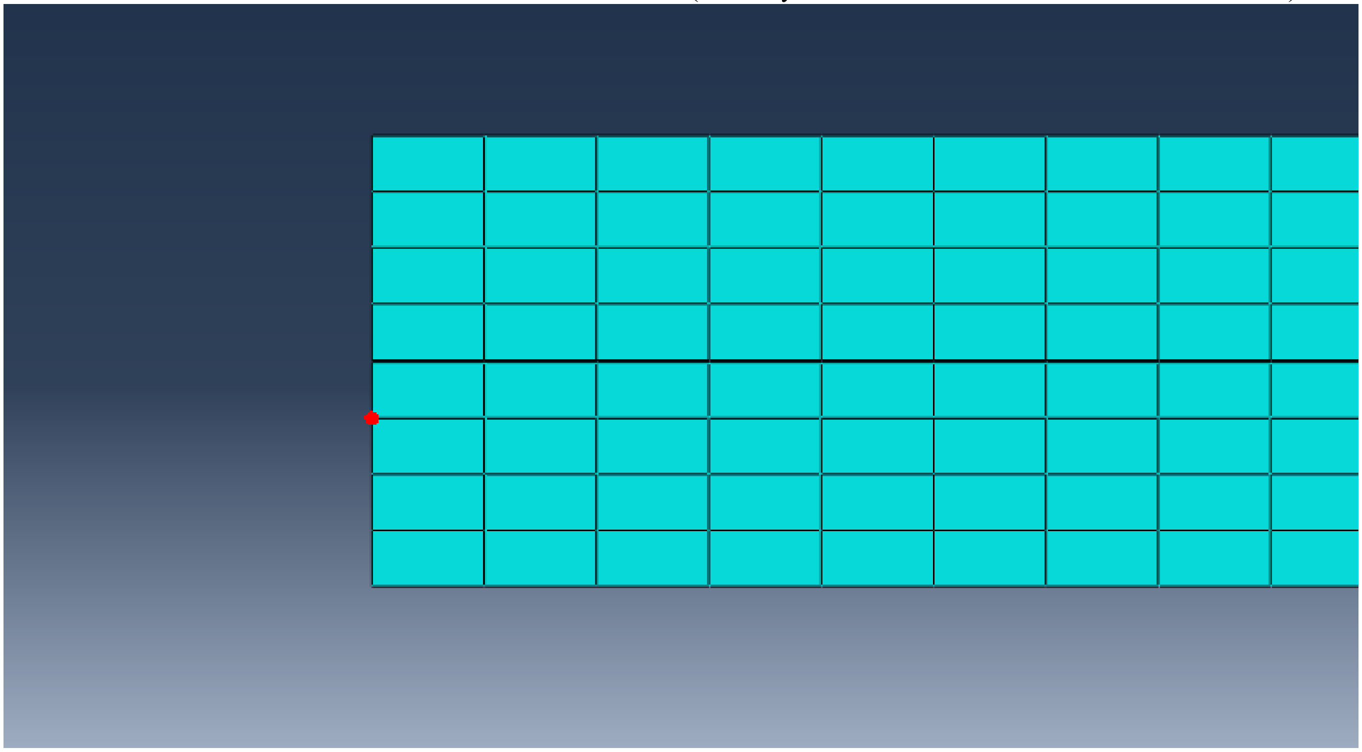
   Select some nodes (half) along top edge from the right (see below). (**Note**: This node will not break in the simulation, and is the refe

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).

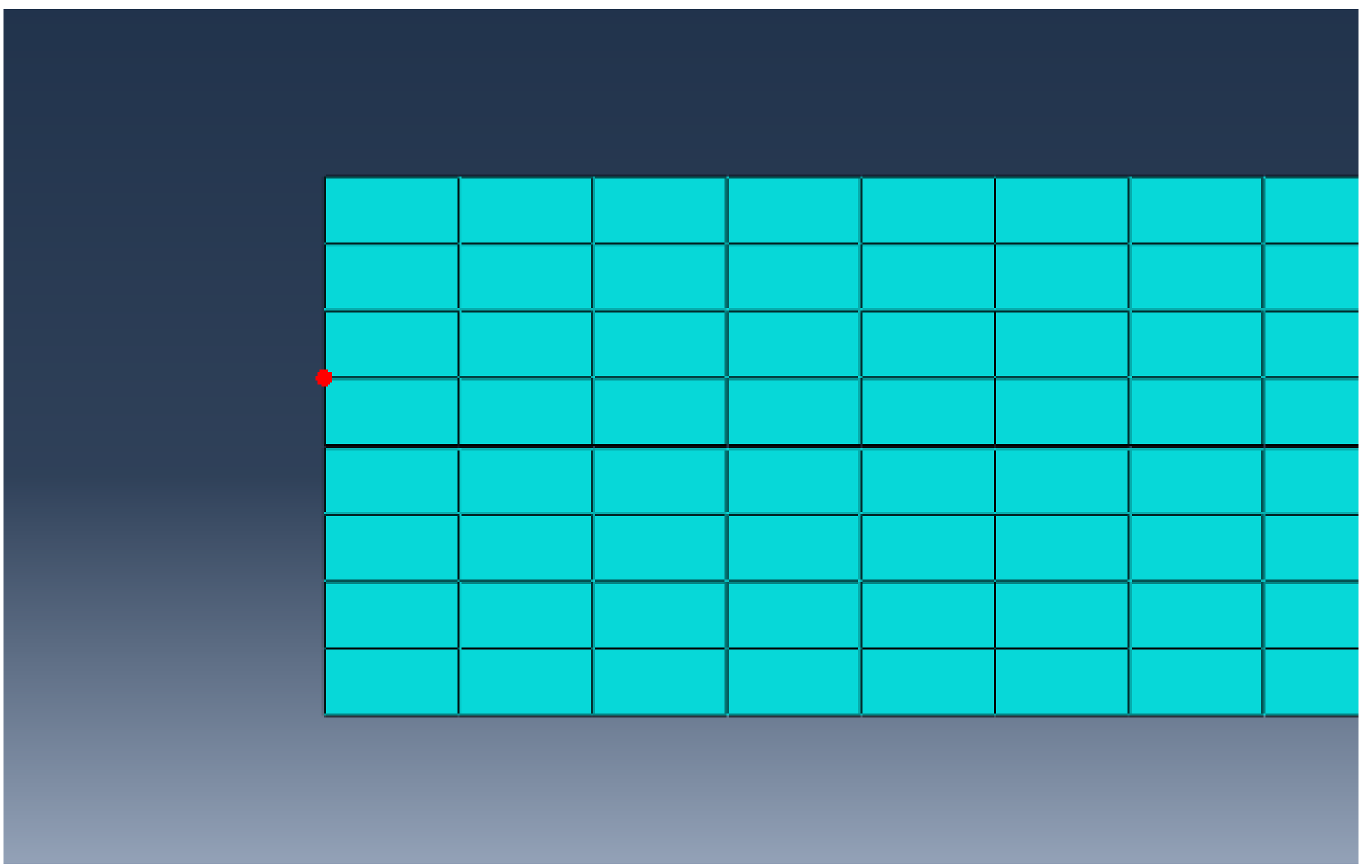
   

2. Create a Node set named `Dbot` and select the node show below. (You may use the *Box Zoom View*  to select the node)



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.)



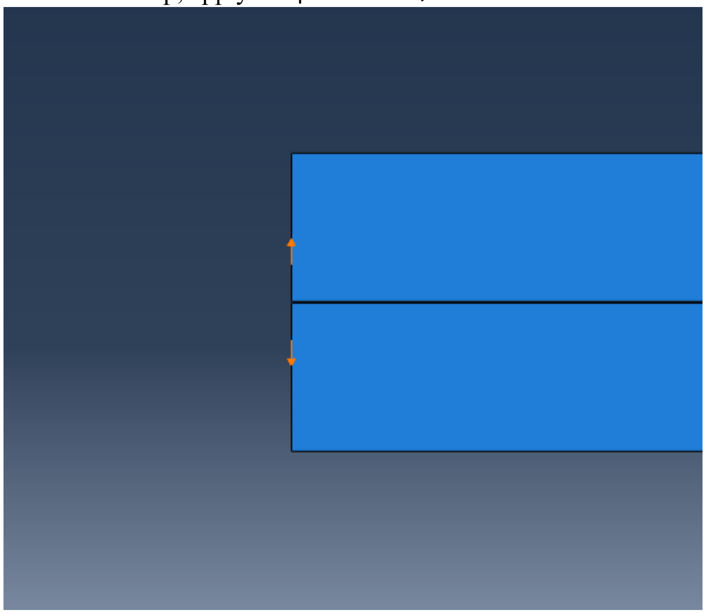
Apply load/displacement and BC¶

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 increament: `5000`; Initial increment size: `0.0`
   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*

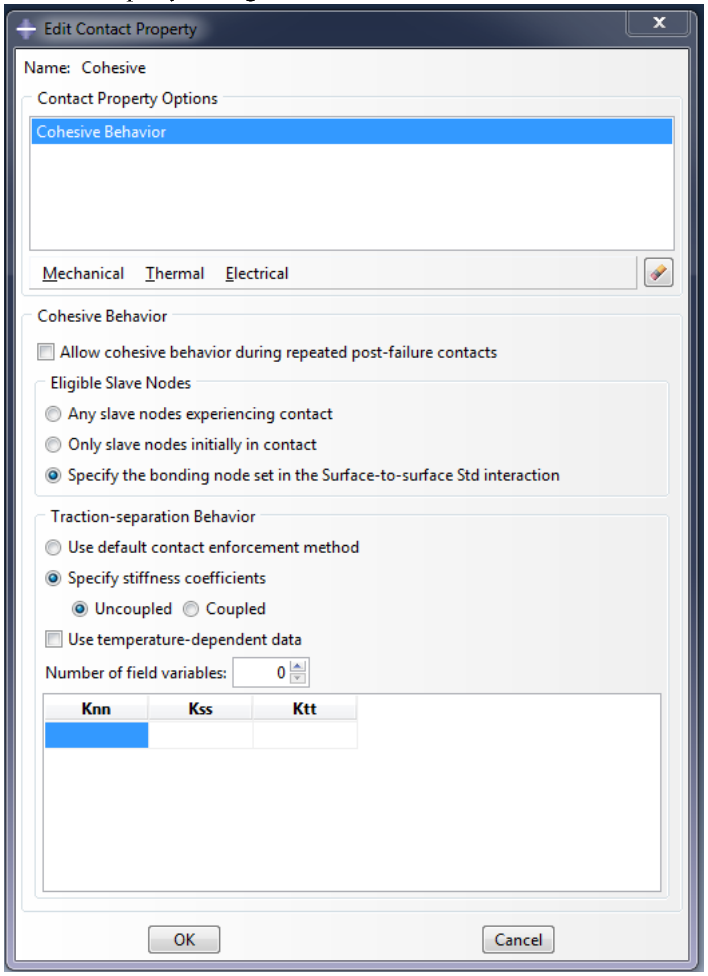
3. In the *Load* step, apply `Displacement/Rotation` BC at node set `Dbot` and `Dtop`. For the `Dbot`, set `U2` to be `-0.16` and for the `Dto`

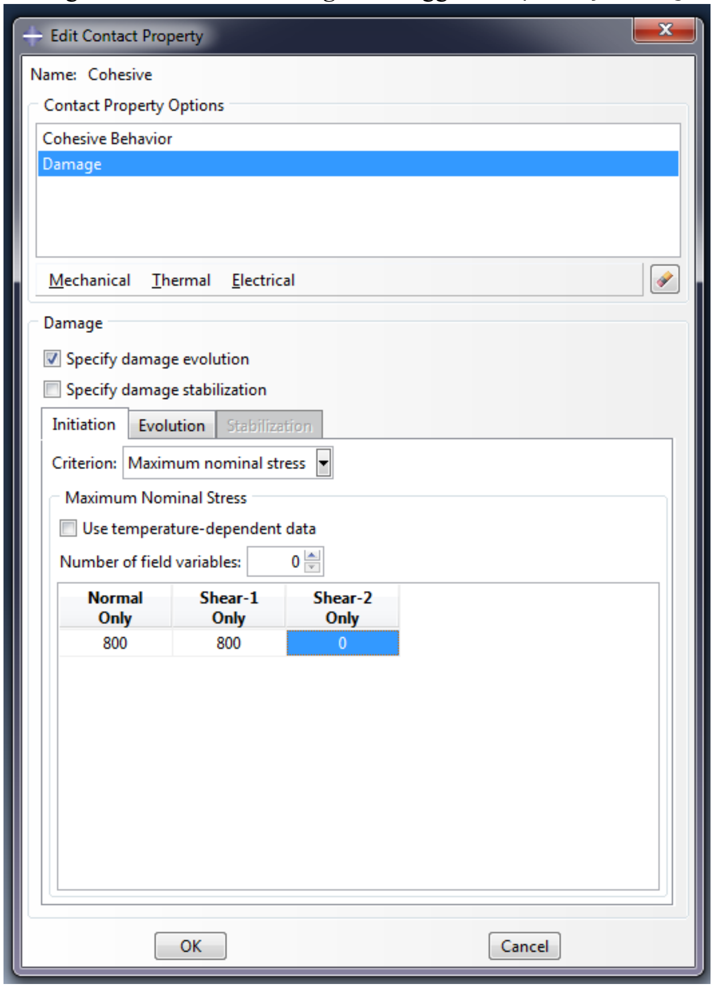
Define the interaction¶

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 Sur*

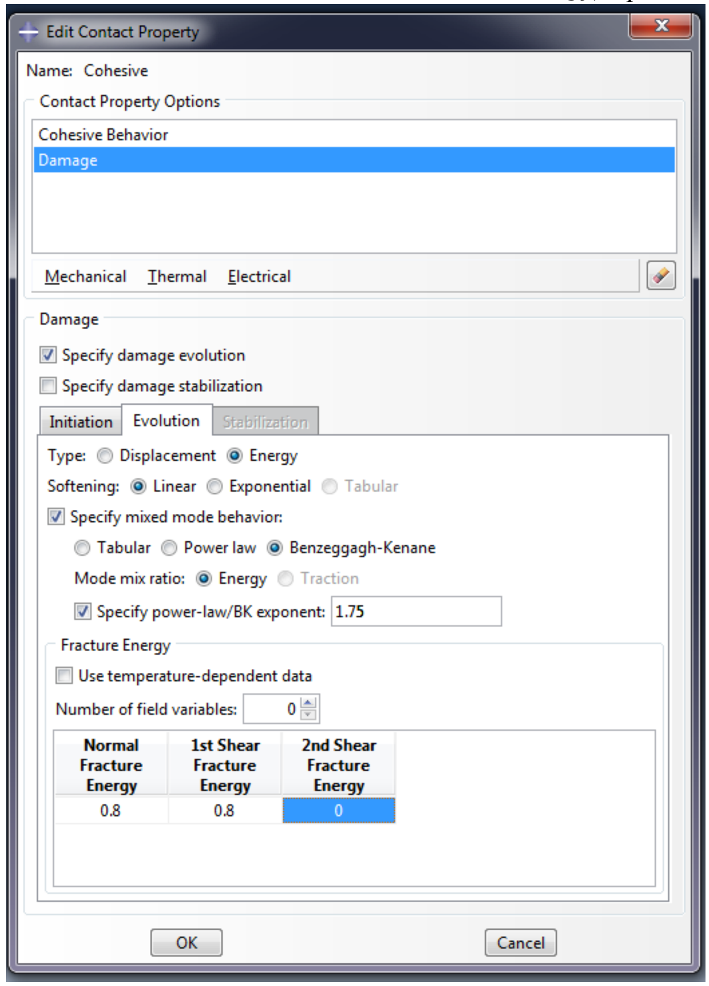
interaction 

3. Then go to *Mechanical Damage* and toggle on `Specify damage evolution`. In `Initiation`, input `800` to `Normal only` & `Shear`



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

Toggle on `Specify power-law/BK exponent` with a value of `1.75`. For the `Normal Fracture Energy`, input `0.8`; for the 1^{st} `Sh`

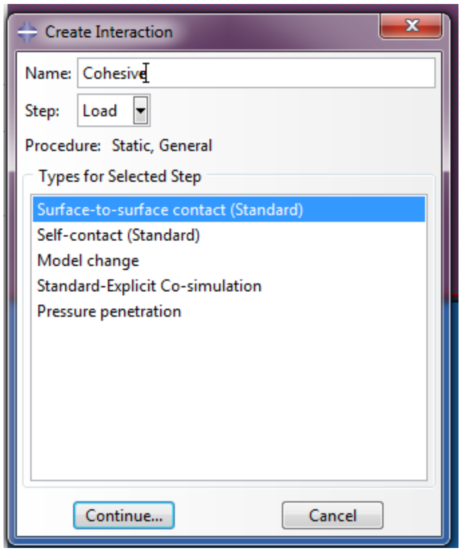


input `0.8`; 2^{nd} `Shear Fracture Energy`, input `0`.

4. Finally select `Mechanical→Geometric→Properties`. Before we create interaction, we should define the two interaction surface, w Go to `Tools→Surfaces→Create`.

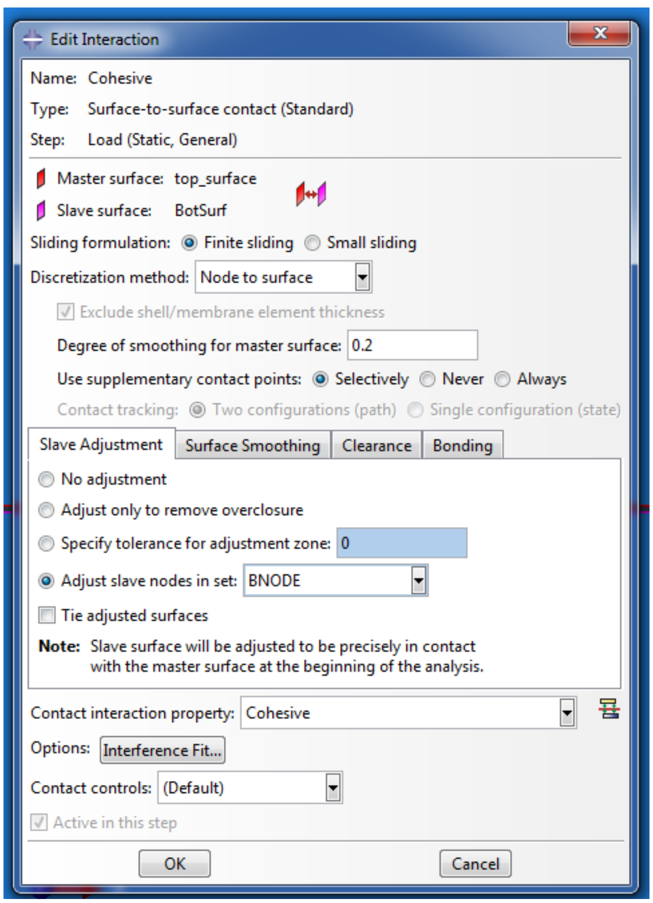
Name the top surface `TopSurf` and the bottom surface `BotSurf`. (You may need to zoom in to select both surfaces.)

5. Use the *Create Interaction* tool  to create a `Cohesive` interaction of type `Surface-to-surface contact` (`Standard`) with `Top` surface (since the adjust nodes we define are at 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.

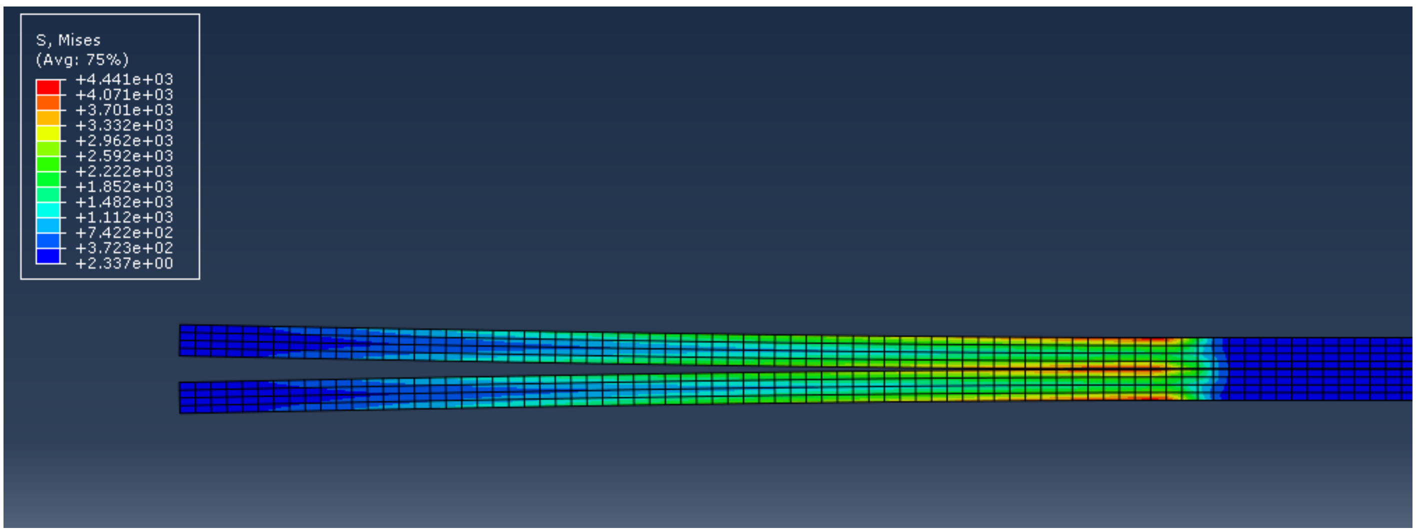
Specify the History output¶

Usually in the double cantilever beam experiment, we would like to get the traction-separation figure at the crack front. Go to *Hist mode tree and select `Domain` Set: `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 hi

---

Credits

Neal Davis, Ruizhi Li, and Binyue Hou developed these materials for Computational Science and Engineering at the University of Illinoi