Now that you have experimentally determined the Young’s Modulus of the tissue, you can use this to simulate how the material will perform under different loading. The first step is usually to simulate the experiment you performed to ensure that the results match. Follow these steps to set up the simulation:

1. Open ABAQUS/CAE from Windows “Start menu”.
2. Select “Create Model Database”.

![ABAQUS/CAE start session]
3. From the main menu select **Part → Create**. In the **Create Part** window shown below, change the **Name** to “Tissue-beam” and **Approximate size** to 100.

![Create Part Window](image)

4. Choose **Continue**.

5. From the left tool bar click **Create lines: rectangle**.

![Create lines: rectangle](image)

6. To create the cross section of the tissue beam, you need to input the x and y coordinates of the top left corner and bottom right corner. For example, you want to create a rectangle with height of 3 mm and width of 12 mm, at first you need to input the top left corner (0, 3) and press **Enter**, and input the bottom right corner (12, 0) and press **Enter**. Click X button if you finish.

![Pick a starting corner](image)
7. Choose the **Done** button.

8. Then the **Edit Base Extrusion** dialog box appears. For example, the length of the tissue beam is 50 mm, change the **Depth** to 50. Then click **OK**.

9. If everything is correct, your model will look like this:

![Model Image]

10. Now you finished sketching the model.
11. Next you need to define the material property for the tissue. Change the **Module** from **Part** to **Property**.

12. From the main menu select **Material** → **Create**. The **Edit Material** dialog box appears. Change the name of the material to **Material-Tissue**. Choose **Mechanical** → **Elasticity** → **Elastic**.
13. In the window below input the **Young's Modules** you have determined experimentally and **Poisson's Ratio** of 0.48. Click **OK**.

14. Now you have the material property for the soft tissue, next you need to create a Section, from the main menu select **Section → Create**, the **Create Section** dialog box appears. Change the Name of the section to **Section-Tissue**. Click **Continue** and in the next window make sure the material is what you created in step 12, and then click **OK**.

15. Once you have a Section, you need to assign this section to your model. Before you assign this section to the model, you need to create a Set. From the main menu select **Tools → Set → Create**, then, the **Create Set** dialog box appears, change the name to **Set-Beam**. Choose **Continue**.
16. Next, you need to select everything in your window.

17. Then your model will become red, click Done to finish.
18. To assign the Section to your Set from main menu select Assign → Section. Click Sets to choose your Set-Beam in the Region Selection window.

19. Click Continue and then OK. Your model will become green meaning you assigned Section correctly.

20. Next you need to create an instance of your model. Change the Module from Property to Assembly (See the picture in Step 11).
21. From main menu select Instance → Create. In the Create Instance window make sure your model Tissue-beam selected and Instance Type is Dependent, and then click OK.
22. Next to run the simulation you need to create a Step. Change the Module from Assembly to Step (See the picture in Step 11).

23. From main menu select Step → Create. In the Create Step window select Continue then OK.

24. Next you need to mesh your model. Change the Module from Step to Mesh (See the picture in Step 11). In the upper toolbar select Part and your part name.
25. Before you mesh your model, you need to seed your model in order to control the density of your elements. From main menu select **Seed → Edge by number**. Select the edge in the width direction, then input 12 for the numbers of the elements along the edge.

![Seed Edge by number](image)

26. Click Constraints, then in the **Edge Seed Constraints** window select *Do not allow the number of elements to change*. Click **OK** and press **Enter**. Click **Done** to finish seeding edge.

![Edge Seed Constraints](image)

27. From main menu select **Seed → Part**, in the **Globe Seeds** window change the *Approximate globe size* to 1.5 then click **OK**.

![Globe Seeds](image)

28. From main menu select **Mesh → Element Type**, select your model and click **Done**.
29. In the **Element Type** window, select *Incompatible modes* and make sure your element type is C3D8I. Then click **OK**.

30. Now you are ready to mesh your model, from main menu select **Mesh → Part**, and click **Yes** to mesh your model.

31. If everything is correct, your model should look like this:
32. Now you are ready to set up the boundary condition and load of your model. Change the Module from Mesh to Load. (See the picture in Step 11). From main menu select Tools → Set → Manager. In the Set Manager dialog click Create.

33. In the main toolbar select Show Native Mesh to display your mesh results if it is not shown.

34. In the Create Set dialog change the Name to Set-fixed-end, select set Type as Node, and then click Continue.

35. Select one end of your model which the axes of coordinates system attached, and then click Done.
36. Back to the **Set Manager** dialog you should see the set you just created. Click **Create** again to define another set for definition of load, change the **Name** to **Set-load**, **Type** to **Node** and **Continue** (See picture in Step 33).

![Create Set dialog](image)

37. Select the node which located in the approximate position you applied the weight. For example:

![Nodes](image)

38. Then click **Done** in the status bar to finish. The **Set Manager** dialog will show all the sets you created.

![Sets](image)

39. Choose **Dismiss** to close the window. Now you are ready to define load and BC.

40. From main menu select **Load → Create**, the **Create Load** dialog appears.
41. Make sure the Concentrated force is selected, then **Continue**. Click **Sets** in the statue bar.

42. Select **Set-load** in the **Set Manager** dialog (See Figure in step 38) and **Continue**.

43. In the **Edit load** dialog change CF1 and CF3 to 0, and input the first weight in Newton to CF2 (minus here means the minus Y direction), then click **OK**.

44. From main menu select **BC → Create**, the **Create BC** dialog appears. Change the Step to **Initial** and Type for Selected Step to Displacement/Rotation, then **Continue**.
45. Select Set-fixed-end in the Set Manager dialog (See Figure in step 38) and Continue.
46. In the Edit BC dialog select U1, U2 and U3, then click OK.

![CSYS: (Global) Edit...]

47. Now you are ready to create a Job and run the simulation. Change the Module from Load to Job (See the picture in Step 11). From main menu select Job → Create. In the Create Job dialog change the job Name to e2k-beam-bending-1 and then click Continue.

![Name: e2k-beam-bending-1]

48. Click OK in the Edit Job dialog.
49. From main menu select Job → Manager, in the Job Manager dialog you will see the job you just created and click Submit to run the simulation.

![Job Manager]

50. Click Results to view the simulation results if the job finish.
51. To view the displacement in Y direction, from main menu select Results → Field Output, select U for Output Variable and U2 for Component, and then OK.
52. The displacement field of your model will look like this. Record your maximum displacement.

53. Now you finished the simulation of first weight, you can do the second weight. Change the Module back to Load. From main menu select Load → Manager. In the Load Manager dialog click Edit.
54. Change the CF2 to the second weight and **OK** (See Figure in step 43).
55. Repeat step 47 to 52 with Job name `e2k-beam-bending-2`, then resubmit the simulation.
56. Write down your experimental and simulation results in this table.

### Experimental and Simulation Displacements of Beam Bending Test.

<table>
<thead>
<tr>
<th>Displacement in Y</th>
<th>Experimental (mm)</th>
<th>Simulation (mm)</th>
<th>Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight 1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Weight 2</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>