Problem:

A square vertical column with one end fixed and one end free is made of a steel alloy (E = 200 GPa). The cross sectional area of the column is 100 mm$^2$ and the column is 50 mm long. A canteliver beam of identical material and cross section is located 2 mm above the vertical column. A 20 kN load is applied to the canteliver beam 26 mm from the free end, causing it to bend and contact the lower. Draw the contour of the stresses in the both beams.
Impact & Stresses on Beams

Overview

Anticipated time to complete this tutorial: 45 minutes

Tutorial Overview
This tutorial is divided into four parts:
1) Tutorial Basics
2) Preprocessing
3) Solution
4) Post Processing

Audience
This tutorial assumes familiarity of ANSYS 8.0; therefore, it does not go into step by step detail.

Prerequisites
1) ANSYS 8.0 in house “Structural Tutorial”
2) Completion of all Basic Machine Design Tutorials
3) Completion of three or more Guided Machine Design Tutorials

Objectives
1) Use contact elements to solve for the contact stresses
2) Use advanced solution techniques to solve for contact loading

Outcomes
1) Know how to solve simple geometries involving contact
2) Increase efficiency in problem set up and solving speed
In this tutorial:

- Instructions appear on the left.

- Visual aids corresponding to the text appear on the right.

- All commands on the toolbars are labeled. However, only operations applicable to the tutorial are explained.

The instructions should be used as follows:

- **Bold >** Text in bold are buttons, options, or selections that the user needs to click on.

  Example: > **Preprocessor > Element Type > Add/Edit/DeleteFile** would mean to follow the options as shown to the right to get you to the **Element Types** window.

- **Italics** Text in italics are hints and notes.

- **MB1** Click on the left mouse button.

- **MB2** Click on the middle mouse button.

- **MB3** Click on the right mouse button.

Some basic ANSYS functions are:

To **rotate** the models use Ctrl and MB3.

To **zoom** use Ctrl and MB2 and move the mouse up and **down**.

To **translate** the models use Ctrl and MB1.
Impact & Stresses on Beams

Preprocessing

1) Change the working directory, jobname, and title of your project.

2) Set Preferences to Structural.

3) Create two rectangles using the Rectangle by 2 Corners creation tool.
   > Preprocessor > Modeling > Create
   > Area > Rectangle > By 2 Corners

   Rect 1: (0,15,100,10)
   Rect 2: (74,13,10,-50)

3) Add a Solid Quad 4node 42 element. Also referred to as (Plane 42)

   From the options, change Element Behavior K3 to "Plane strs w/thk"

4) Add a Real Constant for PLANE42. In the thickness (THK) box, enter 10.

   Note: This defines a the thickness for the beam

5) Create a Material Model.
   E: 200000
   PRXY: .3

6) Change the Element edge length for the areas to 2.
   > Preprocessor > Meshing
   > Size Cntrs > Manual Size > Areas
   > All Areas

7) Mesh the Areas

   Your model should look similar to the picture shown to the right
8) Add a **Contact pt-to-surf 48** element (Also referred to as Contact 48).

Note: If your version of ANSYS does not have a contact 48 element, use the contact wizard instead.

From the options, change Contact time/load prediction K7 to **Reasonbl T/L inc.**

9) Add a **Real Constant** for CONTACT48. In the Normal contact stiffness KN box, enter 20000 and enter 10 in the Target length tolerance (TOLS) box.

10) At this point you are ready to define the sets of nodes that are likely to come in contact.

   > Utility Menu > Select > Entities

Select **Areas** and **By Num/Pick** from the pull down menus and select **From Full** from the radio buttons as shown to the right.

   > OK

Select the canteliver beam

   > OK

Now Select **Nodes** and **By Location** from the pull down menus, **Y coordinates** and **Reselect** from the radio buttons and enter a value of 15 as shown to the right.

   > OK

Now Select **X coordinates** and **Reselect** from the radio buttons and enter a value of 50,100 as shown to the right.

   > OK

Note: 50,100 means that all the nodes with coordinates between (50,15) and (100,15) will be selected. That represents half of the bottom edge of the canteliver beam.
11) Create a component.
   > Utility Menu > Select
   > Comp/Assembly > Create
   > Component

Enter the component name “Source” as shown to the right.
   > OK

12) It is necessary to reselect all nodes before starting to select others. To do so
   > Utility Menu > Select > Entities

Select the Also Select radio button and click Selez All.
   > OK

13) Now select the nodes associated with the vertical beam. Repeat the last few steps but select the vertical beam this time. The X coordinates vary from 74 to 84 and the Y coordinate is 13. When creating the component this time, enter the name Target.

14) Be sure to reselect all nodes before moving on
   > Utility Menu > Select > Entities

Select the Also Select radio button and click Selez All.
   > OK

15) Set the contact elements attributes as shown to the right.
   > Preprocessor > Modeling > Create
   > Elements > Elem Attributes

   > Preprocessor > Modeling > Create
   > Elements > Surf/Contact
   > Node to Surf
Your model should look similar to the one shown below.
16) Change some of the Sol’n Control.
   > Solution > Analysis Type
   > Sol’n Control

Under the “basic” tab, make the following changes as shown to the right.

Under the “Nonlinear” tab, make the following changes as shown to the right.
Impact & Stresses on Beams

Solution

17) Apply **Structural Displacement** constraint “On Lines” at the base of each beam. Constrain all of their degrees of freedom.

18) Apply a **Structural Force** on nodes. Select the node with coordinate (74,25). Give it a value of -20000 applied in the Y direction.

19) 
   - > Solution > Analysis Type
   - > New Analysis

Select **static** and click ok.

20) Solve the model
   - > Solution > Solve > Current LS

21) When it is done solving, Select the General Postproc.

Adjust the Graphical Scaling.
   - > Utility Menu > PlotCtrls > Style
   - > Displacement Scaling

Make the following changes as shown to the right.
22) Plot the results.
   > General Postproc > Plot Results
   > Contour Plot > Nodal Solu > Stress
   > Von Mises

   The stress contours are displayed to the right.

23) Adjust the contour Scale.
   > Utility Menu > PlotCtrls > Style
   > Contours
   > Non-Uniform Contours