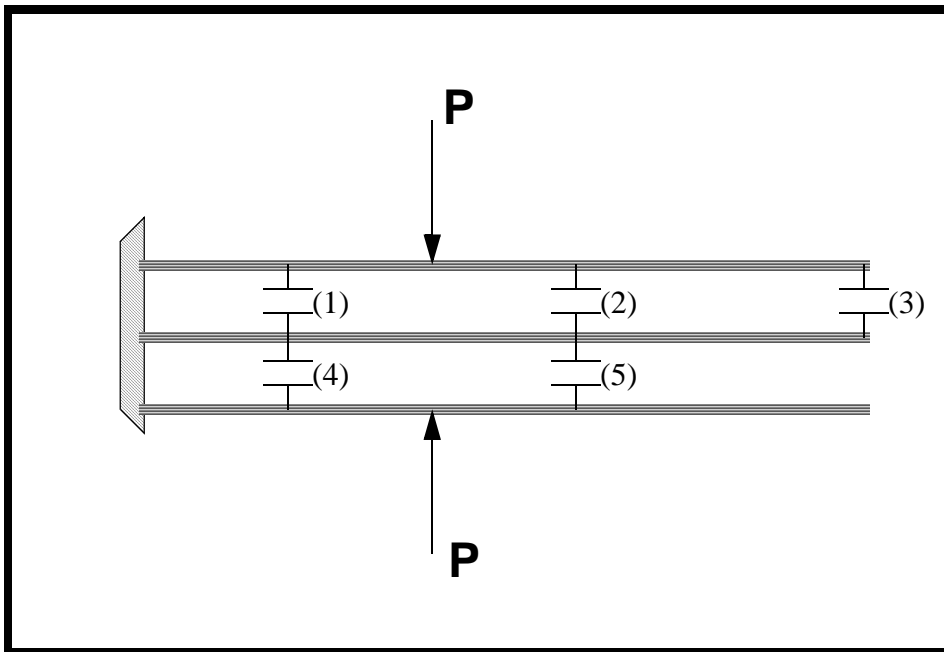

LESSON 5

Gap Elements



Objectives:

- Use of GAP contact elements
- Small deformation theory
- Varying contact condition

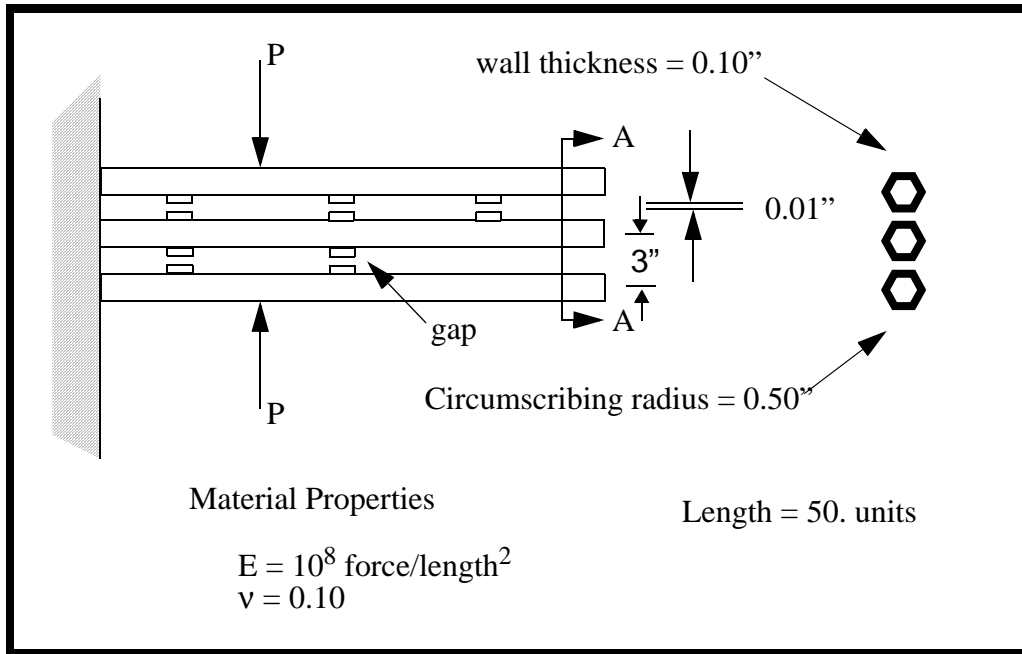


Model Description:

Three parallel cantilever beams are initially separated, but have possible contact points in five locations, as shown in the figure on the following page. A pair of pinching loads is applied, as shown. The pinching loads are increased from no load at the beginning of the step, to full load, 200 lbs, at the end of the step. Only small displacements are considered, so that each beam responds in pure bending. The problem is entirely linear, except for the switching contact conditions. The sequence of events is readily imagined:

1. The top and bottom beams bend as the pinching forces are applied, and the first contact occurs when the tip of the top beam hits the tip of the middle beam (gap 3 closes). Up to this point the problem is symmetric about the middle beam, but it now loses that symmetry.
2. Subsequent to this initial contact, the top and middle beams bend down and the bottom beam continues to bend up, until contact occurs at gap 5.
3. As the load continues to increase, gap 2 closes.
4. Next, gap 3 opens, as the support provided to the top beam by gap 2 causes the outboard part of the top beam to reverse its direction of rotation. At this point (when gap 3 opens), the solution becomes symmetric about the middle beam once again.
5. Finally, as the pinching loads increase further, gaps 1 and 4 also close. From this point on, the contact conditions do not switch, no matter how much more load is applied.

Initially all gaps are open, with an initial gap clearance of 0.01. The pinching loads are increased monotonically from 0 to 200. The beam lengths, modulus, and cross-section are shown on the following page.



Suggested Exercise Steps:

- Create a new database and name it **gapuni.db**.
- Change your analysis code preference to P3/ADVANCED FEA.
- Create the 3 lines 50 units long, and mesh each of them with 5 cubic beam elements.
- Specify material properties and element properties.
- Apply the loads and boundary conditions.
- Analyze the model and post process the results.

Exercise Procedure:

1. Create a new database named **gapuni.db**.

File/New ...

Database Name:

gapuni.db

OK

2. The viewport will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms inside MSC/PATRAN. In the *New Model Preference* form set the analysis preference to MSC/ADVANCED FEA.

Analysis Code:

MSC/ADVANCED_FEA

OK

3. Create the geometry for the model using the curve XYZ option with $\langle 50, 0, 0 \rangle$ for the vector and $[0\ 0\ 0]$, $[0\ 3\ 0]$, and $[0\ 6\ 0]$ for the starting points.

First, turn on the entity labels using the following toolbar icon:



Show Labels

For **Curve 1**.

◆ Geometry

Action:

Create

Object:

Curve

Method:

XYZ

Vector Coordinates List:

$\langle 50\ 0\ 0 \rangle$

Origin Coordinates List:

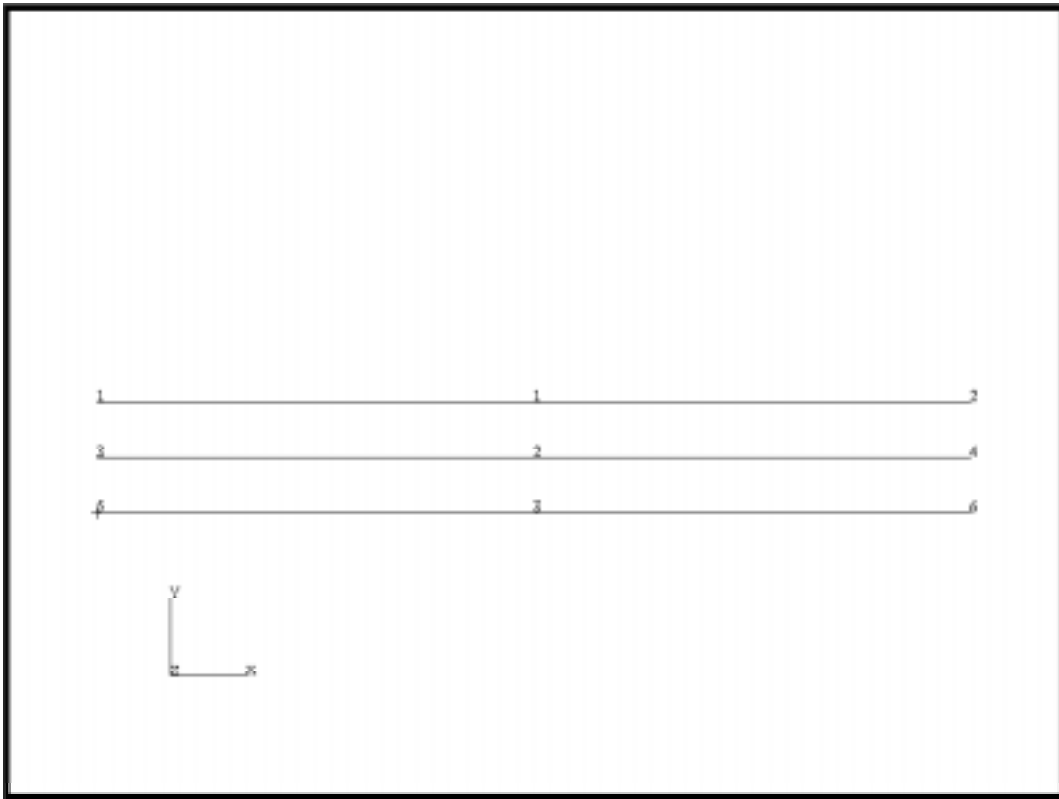
$[0\ 6\ 0]$

Apply

For **Curve 2** and **3** modify the *Origin Coordinates List* to read $[0\ 3\ 0]$ and $[0\ 0\ 0]$ respectively.

These curves will represent the three beams for this example. Your model should appear as shown in Figure 6.1:

Figure 6.1 - Three beams



4. Create a new group, **fem**, to hold all the FEM entities. Make it the current group.

Group/Create ...

New Group Name:

fem

Make Current

Apply

Cancel

5. Mesh the three beams with 10 as the global length, choose bar 2 elements.

◆ Finite Elements

Action:

Create

Object:

Mesh

Type:

Curve

Global Edge Length:

10

Element Topology:

Bar2

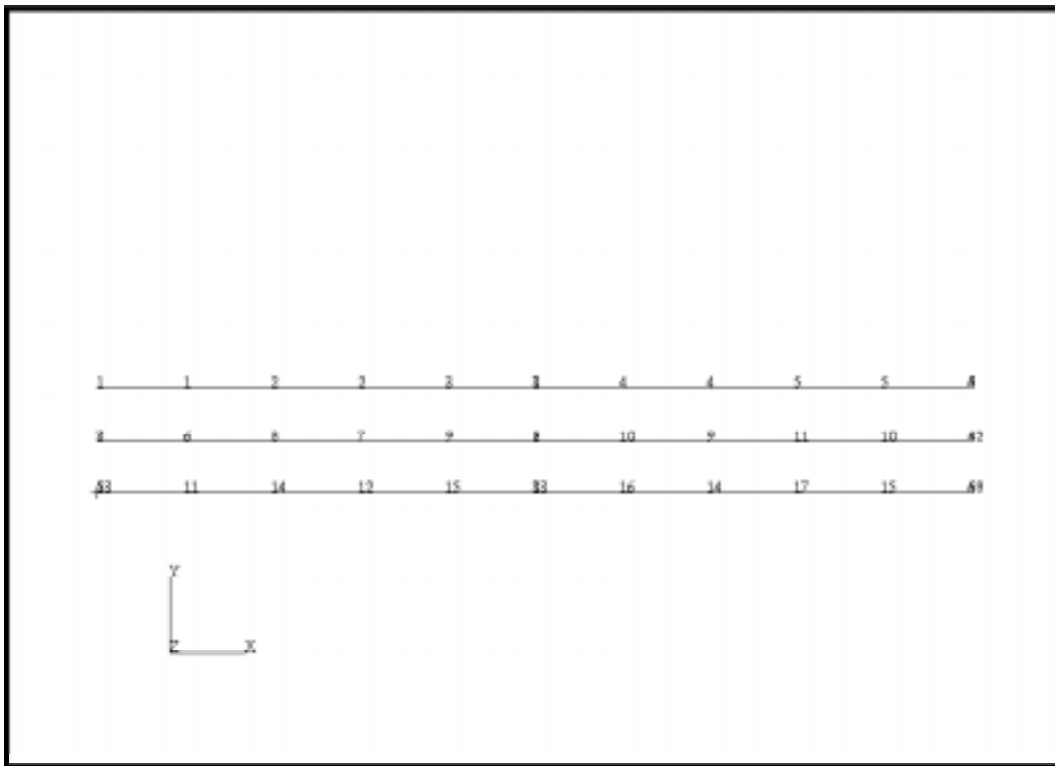
Curve List:

select all three curves

Apply

Your model should appear as shown in Figure 6.2:

Figure 6.2 - Three beams meshed with bar elements



6. Create group **gap** and make it current. It will hold the gap elements.

Group/Create ...

New Group Name:

gap

Make Current

Unpost All Other Groups

Apply

Cancel

7. Create the gap elements between the node pairs (2, 8), (4, 10), (6, 12), (8, 14), (10, 16). These are modeled with **Bar2** elements.

◆ **Finite Elements**

Action:

Create

Object:

Element

Method:

Edit

Shape:

Bar

Topology :

Bar2

Pattern:

Standard

Node1:

Node 2, 4, 6, 8, 10

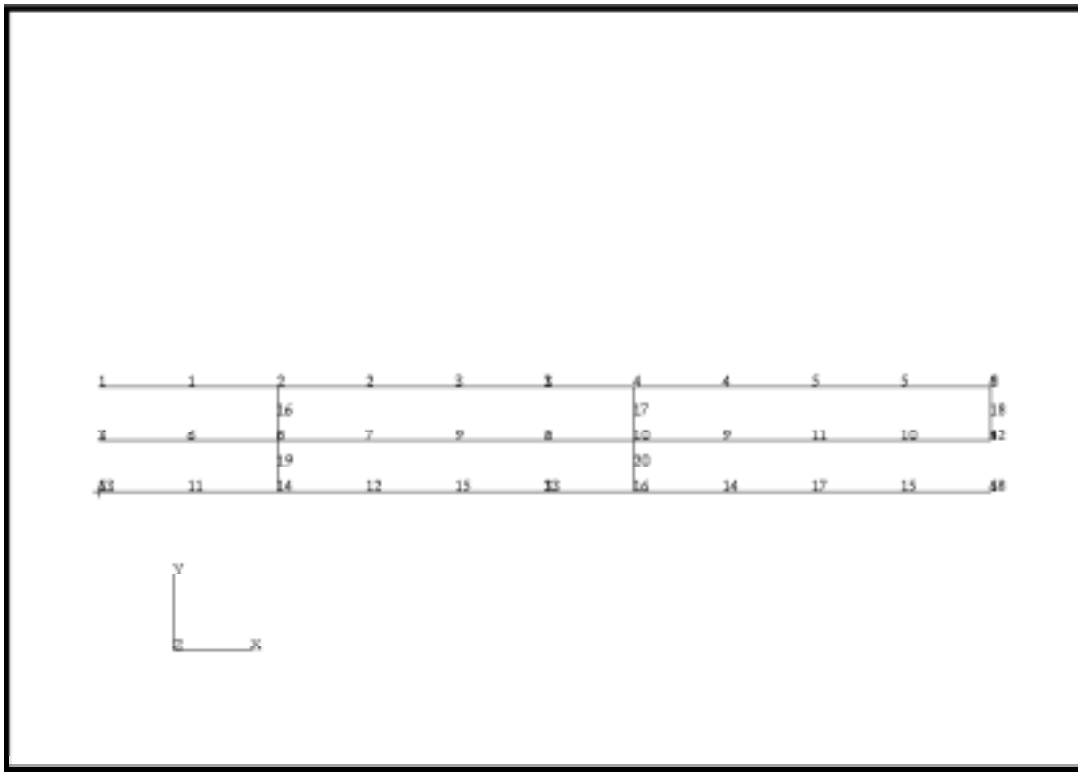
Node2:

Node 8, 10, 12, 14, 16

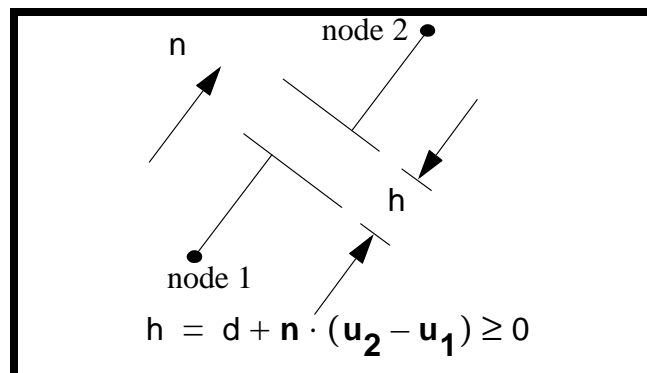
Apply

Your model should now look like the one shown in Figure 6.3:

Figure 6.3 - Beams with gap elements



Note: It should be noted that the creation direction of all the bar elements, $(\mathbf{u}_2 - \mathbf{u}_1)$, is in the negative Y direction. This direction will be used later when we define the contact closure direction under *Element Properties*. To review, contact closure is defined below:



-
8. Create the material, **beam_material**. It is elastic, with Young's Modulus of 1E8, and Poisson's Ratio of 0.10.

◆ **Materials**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Type:</i>	<input type="text" value="Isotropic"/>
<i>Method:</i>	<input type="text" value="Manual Input"/>
<i>Material Name :</i>	<input type="text" value="beam_material"/>
<input type="button" value="Input Properties..."/>	
<i>Constitutive Model:</i>	<input type="text" value="Elastic"/>
<i>Elastic Modulus:</i>	<input type="text" value="1E8"/>
<i>Possion's Ratio:</i>	<input type="text" value="0.10"/>
<input type="button" value="Apply"/>	
<input type="button" value="Cancel"/>	

9. Post the beam finite element group, **fem**.

Group/Post ...

<i>Select Groups to Post:</i>	<input type="text" value="fem"/>
<input type="button" value="Apply"/>	
<input type="button" value="Cancel"/>	

10. Create the element property, **beam_prop**. The thickness is 0.1, the direction is the global y direction, the material is the beam_material. This is applied to the beam elements only.

◆ **Properties**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Dimension:</i>	<input type="text" value="1D"/>
<i>Type:</i>	<input type="text" value="Beam in XY Plane"/>
<i>Property Set Name:</i>	<input type="text" value="beam_prop"/>
<i>Options:</i>	<input type="text" value="Hexagonal Section"/>

Cubic Interpolation	
Input Properties...	
<i>Material Name:</i>	beam_material
<i>Circumscribing Radius:</i>	0.5
<i>Wall Thickness:</i>	0.1
OK	

In the *Application Region* part of the *Element Properties* form, click in the *Select Members* databox. The select menu will appear. Select the icon for the Beam elements as shown below.



Select *all* posted entities in the viewport. This should correspond to elements 1:15.

Add
Apply

- Post the gap finite element group, **gap**.

Group/Post ...

<i>Select Groups to Post:</i>	gap
Apply	
Cancel	

- Create the element property, **gap_prop**. The initial clearance is 0.01, the direction is $\langle 0 \ -1.0 \ 0 \rangle$. This is applied to the gap elements only.

<i>Action:</i>	Create
<i>Dimension:</i>	1D
<i>Type:</i>	Gap
<i>Property Set Name:</i>	gap_prop

Options:

	<input type="text" value="Uniaxial"/>
	<input type="text" value="True Distance"/>
<input type="button" value="Input Properties..."/>	
Initial Clearance:	<input type="text" value="0.01"/>
Vector:	<input type="text" value="<0 -1.0 0>"/>
<input type="button" value="OK"/>	

Note: Remember, the opening/penetration of the gap contact is determined by comparing the relative displacements of the nodes in the contact direction, \mathbf{n} , to a user specified initial distance, \mathbf{d} . The gap opening can be defined by:

$$h = d + \mathbf{n} \cdot (\mathbf{u}_2 - \mathbf{u}_1) \geq 0$$

The relative displacements of the nodes are defined by the node ordering of the bar elements, node 1 and node 2 (which corresponds to \mathbf{u}_1 and \mathbf{u}_2). The normal direction, \mathbf{n} , of the closure is defined in the form above.

In the *Application Region* part of the *Element Properties* form, click in the *Select Members* databox. The select menu will appear. Select the icon for the Beam elements as shown below.



Select *all* posted entities in the viewport. This should correspond to elements 16:20.

<input type="button" value="Add"/>
<input type="button" value="Apply"/>

13. Create a new group **all_fem**.

This group will have all the finite elements for the entire model in it.

Group/Create ...

New Group Name:

Make Current

Unpost All Other Groups

Group Contents:

Add All FEM**Apply****Cancel**

14. Create the loads and boundary conditions. The ends are fixed in all three translations and rotations.

First, fit the display to the model using the following toolbar icon:

**Fit View**◆ **Load/BCs**

Action:

Create

Object:

Displacement

Type:

Nodal

New Set Name:

fixed_end**Input Data...**Translations $\langle T1, T2, T3 \rangle$:**<0, 0, >**Rotations $\langle R1, R2, R3 \rangle$:**<, , 0>****OK****Select Application Region...**

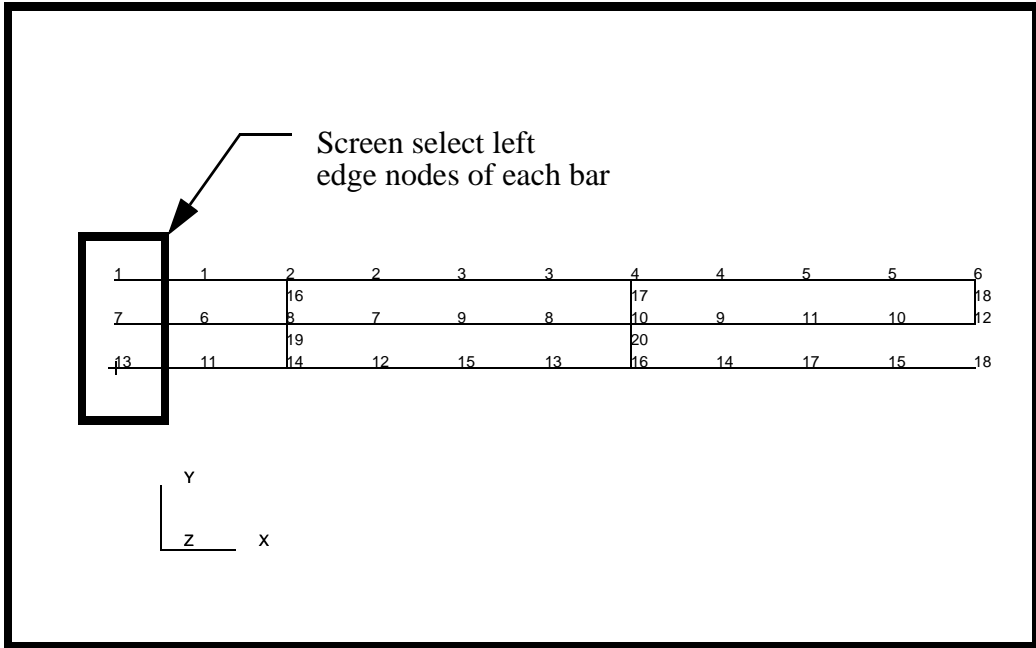
Geometry Filter:

◆ **FEM**

Select Nodes:

see Figure 6.4

Figure 6.4 - Fixed ends of beams



Add

OK

Apply

15. Create a force on node 3 pointing down and a force on node 15 pointing up. The force vectors are $\langle 0, -200, 0 \rangle$ and $\langle 0, 200, 0 \rangle$ for node 3 and 15, respectively.

Action:

Create

Object:

Force

Type:

Nodal

New Set Name:

down

Input Data...

Force $\langle F1, F2, F3 \rangle$:

$\langle , -200, \rangle$

OK

Select Application Region...

Geometry Filter:

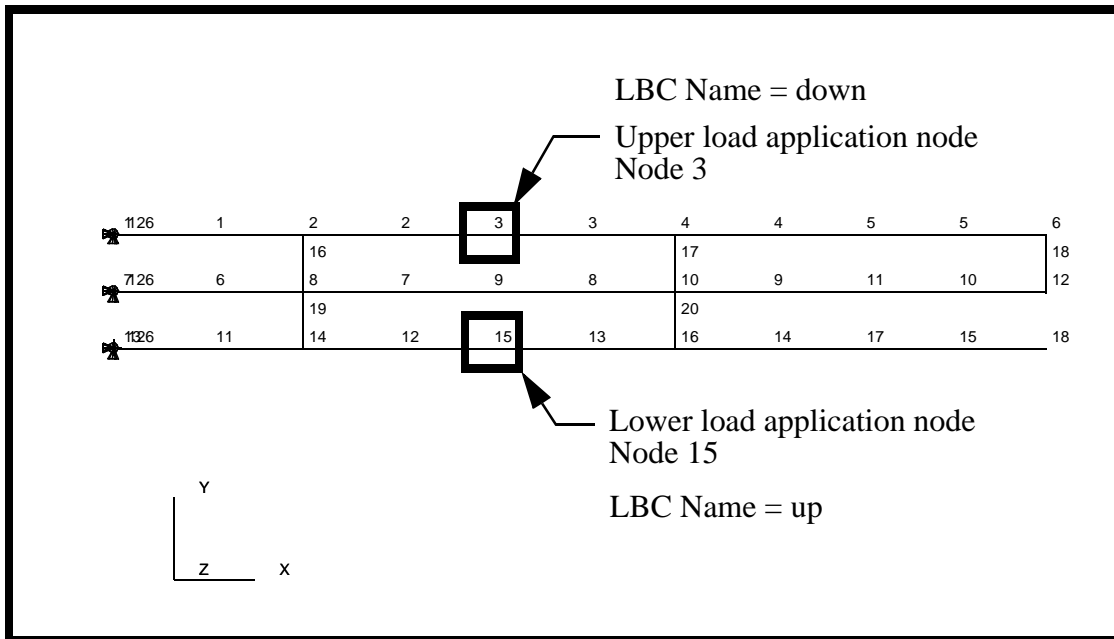
◆ **FEM**

Select Nodes:

see Figure 6.5

Click in the *Select Nodes* databox and select **Node 3** in the middle of the top beam as in Figure 6.5:

Figure 6.5 - Load application locations



-
-
-

Similarly, create a second load applied to node 15 in the opposite direction with the name of **up**. The force vector will be $\langle ,200, \rangle$, and the application region is **Node 15** in the middle of the bottom beam as shown in Figure 6.5.

16. Ready the model for analysis. Create an analysis step called **pinching_load**. For this analysis step, you will be using the default load case and default output requests.

◆ **Analysis**

Action:

Object:

Method:

Job Name:

Step Creation...

Job Step Name:

Solution Type:

Solution Parameters...

Large Deflection/Strains:

Riks Method:

Automatic Load Increment:

Delta-T:

Time duration for this step:

17. Create an analysis job and submit it. You will need to select the **pinching_load** step and unselect the default static step. Only the **pinching_load** step should appear in the selected job steps.

Step Selection...

Selected Job Steps:

18. From another UNIX shell, monitor the analysis by monitoring the **.sta** and **.msg** files. When the jobs are finished, read in the results.

Action:

Select Results File...

Available Files:

19. Change the scale of the deformation on the **Results** form

◆ **Results**

Action: **Create**

Object: **Quick Plot**

Select the **Deformation Attributes** icon from the Results form



Scale Factor:

◆ **Model Scale**

20. Create a deformed plot from the results.

Click on the **Select Results** icon



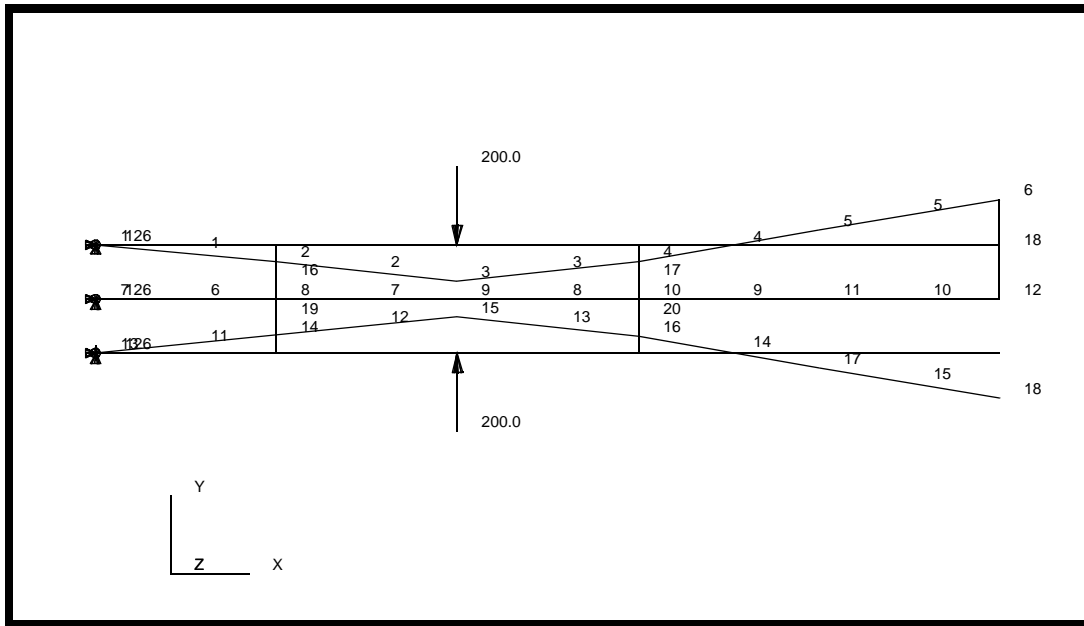
Select Results Case:

Select Fringe Result:

Select Deformation Results:

The plot should look like the one shown in Figure 6.6:

Figure 6.6 - Deformed beams with gap elements



21. Generate a report to get the forces to complete the table.

The final step in this assignment is to fill in the chart on the following page.

Table 1: Summary chart of gap forces for each analysis increment

Increment	Pinching force, P	Force in gap				
		1	2	3	4	5
1						
2						
3						
4						
5						
6	120	Open	59.82	Open	Open	59.82
7						
8						
9						
10						

To get the data for the above table, you will generate a report with all the gap results.

To get the results for the gap elements, those elements need to be in the current group.

Group/Set Current ...

Set Current Group:

gap

Cancel

◆ **Results**

Be sure you are on the Select Results form, click on View Subcase then Subcase Select



Select Results



View Subcase



Subcase Select

Filter method:

All

Filter

Apply

Close

Select Results:

Force and Shear Forces

Apply

The results will be printed to the UNIX window that you started P3 up in. You will need to find that shell and get your results from there. You will get one page of results for each increment. Each page of results contains results for each gap element for that increment. Below is an example of the results produced for increment 6

=====
2.1-Force and Shear Forces, Components

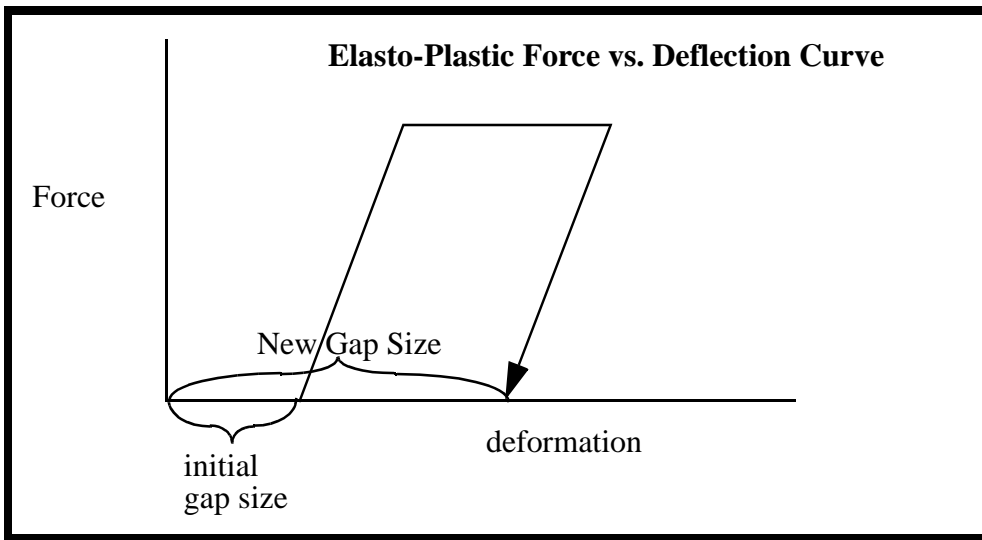
Load case 6 = Default, Step1, TotalTime=0.6 - Lid 3 = (NON-LAYERED)

LAYR ZZ-COMP	ELEM XY-COMP	EPOS	SYS YZ-COMP	XX-COMP XZ-COMP	YY-COMP
3 0.000000	16 0.000000	2	ELEM 0.000000	0.000000 0.000000	0.000000
0.000000	17 0.000000	2	ELEM 0.000000	59.819271 0.000000	0.000000
0.000000	18 0.000000	2	ELEM 0.000000	0.000000 0.000000	0.000000
0.000000	19 0.000000	2	ELEM 0.000000	0.000000 0.000000	0.000000
0.000000	20 0.000000	2	ELEM 0.000000	59.819271 0.000000	0.000000

The XX Component indicates the normal force that is transmitted across the gap. If the value is zero, the gap is not transmitting any force, and therefore would be an indication that the gap is open.

To find out how much the gaps are open, you could plot the result "4.1-Rel. Displacements and Shear Slips, Components". Again, the XX column indicates the amount the gap is open in the normal direction.

Question 1: How would you model an Elasto-plastic gap? Note that as the gap deformations are in the plastic range, the initial gap size increases.



Question 2: How would you define the gap closure direction for a set of coincident nodes.

You may now quit PATRAN.

This concludes this exercise.

Table 2: Answers for Table 1

Increment	Pinching force, P	Force in gap				
		1	2	3	4	5
1	20	Open	6.50	0.73	Open	7.97
2	40	Open	18.34	Open	Open	18.34
3	60	Open	28.71	Open	Open	28.71
4	80	Open	39.08	Open	Open	39.08
5	100	Open	49.45	Open	Open	49.45
6	120	Open	59.82	Open	Open	59.82
7	140	10.71	68.60	Open	10.71	68.60
8	160	31.61	75.88	Open	31.61	75.88
9	180	52.52	83.15	Open	52.52	83.15
10	200	73.43	90.42	Open	73.43	90.42