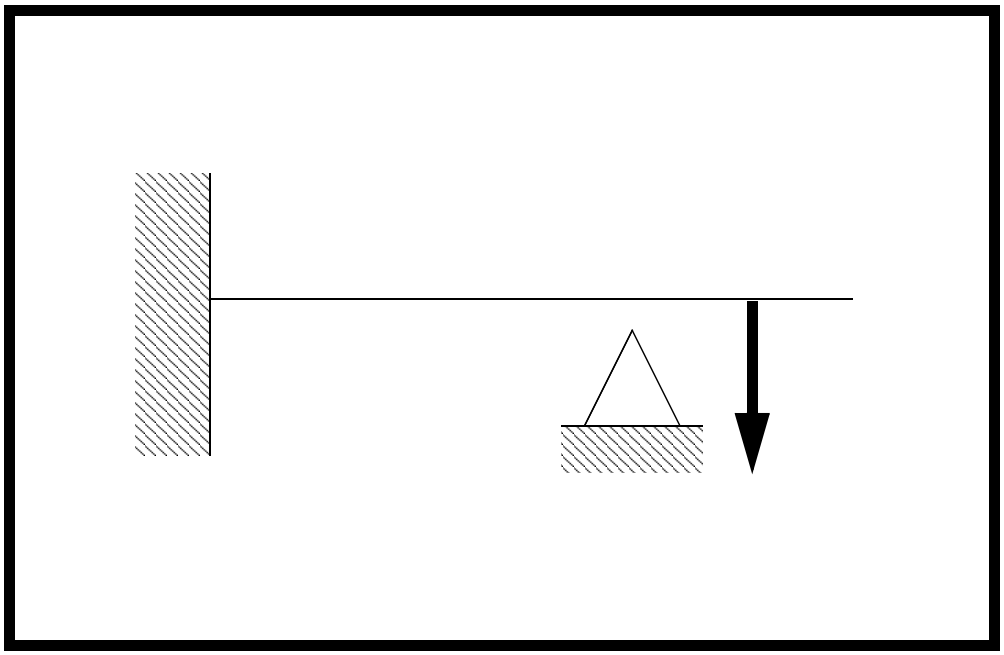
WORKSHOP 30d

Non-Linear Gap Elements with Coincident Nodes



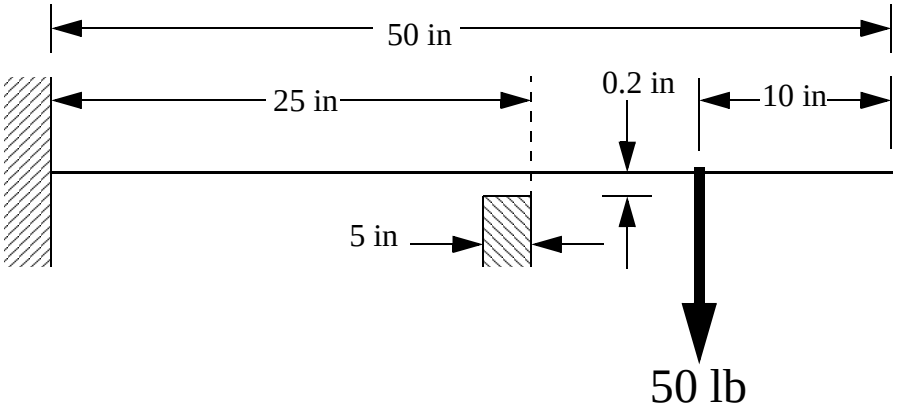
Objectives

- Model contact of beam with another object using CGAP/ PGAP.
- Find maximum deflection.
- Submit the file for analysis in NASTRAN.
- Find the displacement vectors.

Model Description:

Careful examination of CGAP/PGAP card reveals that the actual distance between GA and GB of the gap element does not matter; only the input initial opening matters. From then on, the stiffness is defined by ΔU of GA and GB ($\Delta U = U_A - U_B$) where U is the element X-direction. (If grids are coincident, U is X-direction of some coordinate frame). If ΔU < initial offset, then stiffness is open stiffness (Default = 10^{-14}). If ΔU > initial offset, then stiffness is closed stiffness (No Defaults). Knowing this, our opening can be modeled with coincident nodes.

Figure 30d.1



Assumptions:

- Maximum displacements are at the end of the beam.
- The wall thickness is not important since the deflecting beam will hit its corner.

Questions:

- Does the beam hit the wall? *Yes, but we do not need to know this*
- How can we model the wall? *Model it with coincident nodes and CGAP.*
- What is the maximum displacement? *-0.59 inches in the last two examples.*

Figure 30d.2 - Grid Coordinates and Element Connectivities

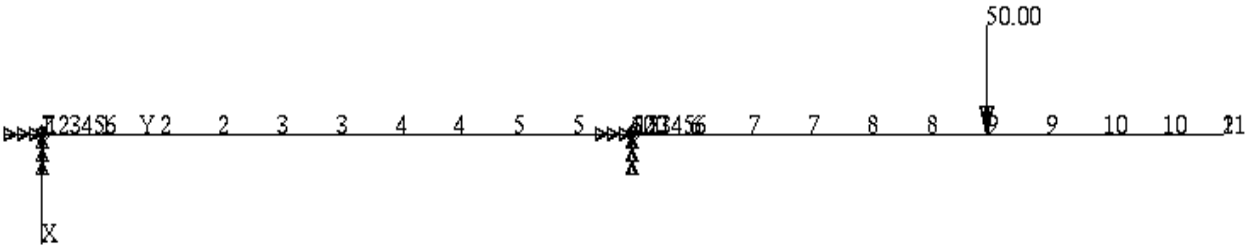


Figure 30d.3 - Beam Cross Section

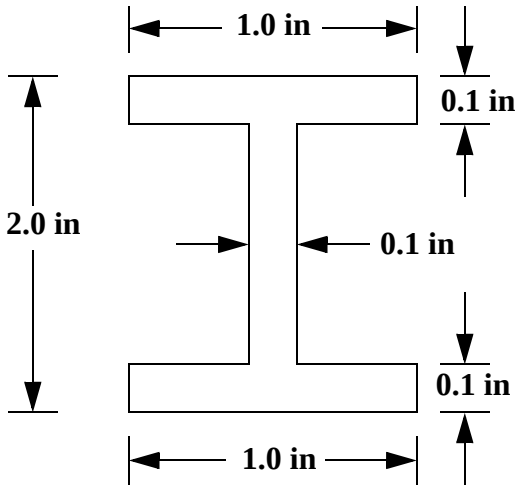


Table 30d.1 - Beam Properties

Length	**50 in**
Height	**2 in**
Width	**1 in**
Thickness	**0.100 in**
Area	**0.38 in^2**
I_1	**0.229 in^4**
I_2	**0.017 in^4**

Suggested Exercise Steps

- Modify Problem 30c as follows:
  - Move Node 12 onto Node 6.
  - Create coordinate frame for coincident GAP element (CORD2R).
  - Prepare for non-linear analysis (SOL106, NLPARM).
- Resubmit for analysis.

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 8, otherwise, proceed to Step 2.

2. Open database created in Problem 30c named **prob30c.db**.

   File/Open Database

   Existing Database Name **prob30c**

   OK

3. Activate the entity labels by selecting the **Show Labels** icon on the tool-bar.

    **Show Labels**

4. Define the finite element of the node touching the wall.

   ◆ **Finite Elements**

   Action: **Modify**

   Object: **Node**

   Method: **Move**

   Node List **Node 12**

   New Node Locations **Node 6**

   Apply

5. Create a new coordinate axis.

   ◆ **Geometry**

   Action: **Create**

   Object: **Coord**

   Method: **Axis**

   Origin **[0 0 0]**

   Point on Axis 1 **[0 -1 0]**

   Point on Axis 2 **[1 0 0]**

Apply

6. Modify the gap element to fit the new coordinate axis.

◆ **Properties**

Action:	**Modify**
Dimension:	**1D**
Type:	**Gap**
Select Prop. Set to Modify	**gap_element**
Gap Orientation/Value Type:	**CID**
Gap Orientation/Value:	**Coord 1**

OK

Apply

7. Now, you will generate the input file for analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Analysis Deck**
Job Name	**prob30d**

Solution Type...

Solution Type:	◆ **NONLINEAR STATIC**

Solution Parameters ...

*(Deselect **Automatic Constraints**.)*	❒ **Automatic Contraints**

OK

OK

Apply

An MSC.Nastran input file called **prob30d.bdf** will be generated. The process of translating your model into an input file is called Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to Step 9.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from Table 30d.1. The result should be similar to the output below.

8. MSC.Nastran input file: **prob30d.dat**

```
SOL 106
TIME 600
CEND
TITLE = Non-Linear Gap Elements w/ Coincident Nodes
ECHO = NONE
MAXLINES = 999999999
SUBCASE 1
   NLPARM = 1
   SPC = 1
   LOAD = 1
   DISPLACEMENT=ALL
BEGIN BULK
PARAM    AUTOSPC NO
PARAM    WTMASS  1.
PARAM    LGDISP  1
PARAM,NOCOMPS,-1
NLPARM   1       10              AUTO    5       25      PW      NO      +       A
+       A        .001    1.-7
PBEAML   1       1               I                                       +       B
+       B 2.     1.      1.      .1      .1      .1              YES     +       C
+       C 1.     2.      1.      1.      .1      .1      .1
CBEAM    1       1       1       2       0.      1.      0.
CBEAM    2       1       2       3       0.      1.      0.
CBEAM    3       1       3       4       0.      1.      0.
CBEAM    4       1       4       5       0.      1.      0.
CBEAM    5       1       5       6       0.      1.      0.
CBEAM    6       1       6       7       0.      1.      0.
CBEAM    7       1       7       8       0.      1.      0.
CBEAM    8       1       8       9       0.      1.      0.
CBEAM    9       1       9       10      0.      1.      0.
CBEAM    10      1       10      11      0.      1.      0.
PGAP     2       .2              1.+7
CGAP     100     2       6       12                              1
MAT1     1       1.+7            .3      .101
GRID     1               0.      0.      0.
GRID     2               5.      0.      0.
GRID     3               10.     0.      0.
GRID     4               15.     0.      0.
GRID     5               20.     0.      0.
GRID     6               25.     0.      0.
GRID     7               30.     0.      0.
GRID     8               35.     0.      0.
GRID     9               40.     0.      0.
GRID     10              45.     0.      0.
GRID     11              50.     0.      0.
GRID     12              25.     0.      0.
SPC1     1       123456  1       12
FORCE    1       9       0       50.     0.      -1.     0.
CORD2R   1               0.      0.      0.      0.      0.      1.      +       D
+       D 0.     -1.     0.
ENDDATA
```

Submit the input file for analysis:

9. Submit the input file to MSC.Nastran for analysis.

   9a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob30d.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

   9b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob30d scr=yes**. Monitor the run using the UNIX **ps** command.

10. When the run is completed, edit the **prob30d.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

11. While still editing **prob30d.f06**, search for the word:

D I S P (spaces are necessary)

What is the displacement at Node 6?

Displacement at Node 6 = _____inches

What is the maximum displacement?

Maximum Displacement = _____inches

Comparison of Results:

12. Compare the results obtained in the **.f06** file with the results on the following page:

D I S P L A C E M E N T V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	0.0	0.0	0.0	0.0	0.0
2	G	-9.465781E-06	-9.785105E-03	0.0	0.0	0.0	-3.714636E-03
3	G	-8.316410E-05	-3.698820E-02	0.0	0.0	0.0	-6.967250E-03
4	G	-2.617157E-04	-7.929895E-02	0.0	0.0	0.0	-9.757844E-03
5	G	-5.647928E-04	-1.344058E-01	0.0	0.0	0.0	-1.208638E-02
6	G	-9.943490E-04	-2.000008E-01	0.0	0.0	0.0	-1.395296E-02
7	G	-1.537440E-03	-2.737601E-01	0.0	0.0	0.0	-1.531596E-02
8	G	-2.163844E-03	-3.529693E-01	0.0	0.0	0.0	-1.613376E-02
9	G	-2.834141E-03	-4.349043E-01	0.0	0.0	0.0	-1.640639E-02
10	G	-3.507048E-03	-5.169327E-01	0.0	0.0	0.0	-1.640642E-02
11	G	-4.179956E-03	-5.989611E-01	0.0	0.0	0.0	-1.640643E-02
12	G	0.0	0.0	0.0	0.0	0.0	0.0

13. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**

14. Proceed with the Reverse Translation process, that is importing the **prob30d.op2** results file into MSC.Patran. To do this, return to the ***Analysis*** form and proceed as follows:

◆ **Analysis**

Action:	**Read Output2**
Object:	**Result Entities**
Method	**Translate**

Select Results File...

Select Results File	**prob30d.op2**

OK

Apply

15. When the translation is complete bring up the ***Results*** form.

◆ **Results**

Action:	**Create**
Object:	**Marker**
Method:	**Vector**
Select Result Case(s)	**Default, PW Linear: 100.% of Load**
Select Vector Result	**Displacements, Translational**

Apply

16. To reset the graphics, click on this icon:

Reset Graphics

You can go back and select any *Results Case, Fringe Results or Deformation Results* you are interested in.

Quit MSC.Patran when you are finished with this exercise.

Disp. @ Node 6	**-0.200 in.**
Max. Disp.	**-0.590 in.**