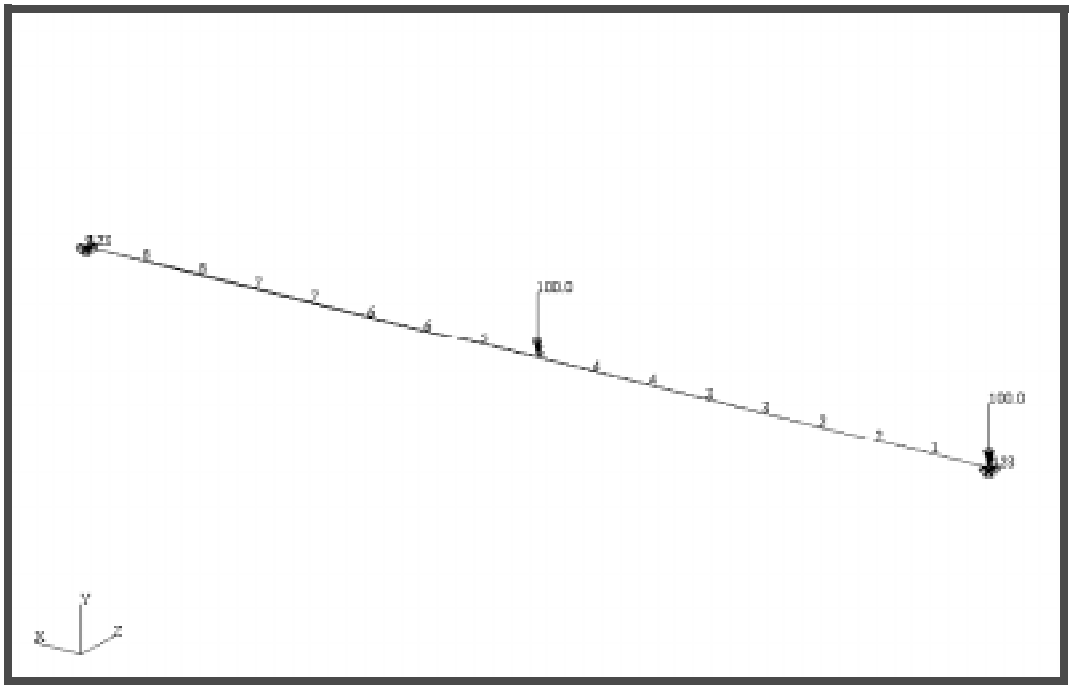
APPENDIX P

Subcases & Case Control



Objectives:

- Define multiple load cases.
- Reference subcases in the Case Control section of the MSC.Nastran input file.
- Specify Output request in the Case Control section of the MSC.Nastran input file.

Model Description:

Figure P.1 is the beam modeled on the title page. Construct a finite element model of the beam, and organize the loading conditions into separate subcases. Reference the subcases in the Case Control section of the MSC.Nastran and define output requests for each subcase within the MSC.Patran interface.

Figure P.1

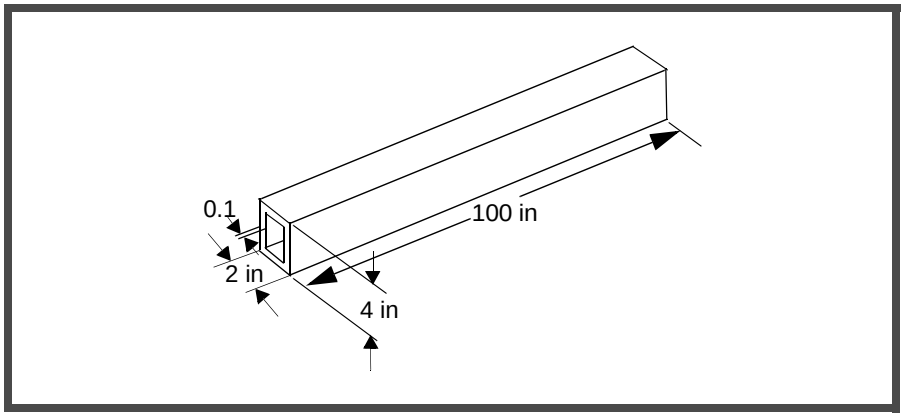


Table P.1 - Bar Properties

Material	**Aluminum**
Elastic Modulus, E =	**10E6 psi**
Poisson's Ratio, v =	**0.3**
Height	**4 in**
Width	**2 in**
Thickness, t =	**0.1 in**

Table P.2 - Loading Combinations

Constaint _1	**Fixed at right end.**
Constaint _2	**Pinned at both ends.**
Load_1	**Force of -100 in the y-direction at left end.**
Load _2	**Force of -100 in the y-direction at center.**
Load Case_1	**Constaint _1 and Load _1**
Load Case_2	**Constaint _2 and Load _2**
Load Case_3	**Constaint _1, Load _1, and Load _2**

Suggested Exercise Steps:

- Define Geometry for the finite element model.
- Mesh the Geometry to create finite elements.
- Define material properties.
- Define element properties.
- Define loading conditions for the model.
- Organize loading conditions into Load Cases.
- Specify different output requests for each load case.
- Select Subcases for an analysis.

Exercise Procedure:

1. Create a new database called **bar.db.**

   File/New...

   New Database Name: **bar**

   OK

   In the New Model Preference form set the following:

   Tolerance: **◆Default**

   Analysis Code: **MSC/NASTRAN**

   Analysis Type: **Structural**

   OK

   NOTE: Whenever possible, toggle off the ❑ **Auto Execute** option by left clicking the check box.

2. Create geometry for the model.

   ◆ Geometry

   Action: **Create**

   Object: **Curve**

   Method: **XYZ**

   Vector Coordinates List: **<100 0 0>**

   Origin Coordinates List: **[0 0 0]**

   Apply

3. Mesh the curve with ID elements.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**12.5**
Curve List:	**Curve 1**

Apply

4. Define Material properties.

◆**Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**alum**

Input Properties...

Elastic Modulus =	**10e6**
Poisson Ratio =	**0.3**

OK

Apply

5. Define element properties.

◆ **Properties**

Action:	**Create**
Object:	**1D**
Type:	**Beam**
Property Set Name:	**bar**

Select Members:	**Curve 1**

Add

Input properties...

Material Name:	**m:alum**
Bar Orientation:	**<0 1 0>**

Click on the **Beam Library** icon.

Beam Library (Create Section)

Choose a **Box Section** from the Beam Library.

Box Section

New Section Name:	**bar**
H:	**4**
W:	**2**
T1:	**0.1**
T2:	**0.1**

OK

OK

Apply

6. Define the fixed boundary condition.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**

Type: **Nodal**

New Set Name: **LBC_1**

Input Data...

Translations <T1 T2 T3> **<0 0 0>**

Rotations <R1 R2 R3> **<0 0 0>**

OK

Select Application Region

Geometry Filter: ◆ **FEM**

Select Nodes: **Node 9**

Add

OK

Apply

7. Define the pinned boundary conditions.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **LBC_2**

Input Data...

Translations: **<0 0 0>**

Rotation: **<0 0 0>**

OK

Select Application Region

Select Nodes: **Nodes 1 9**

Add

OK

Apply

8. Define the first loading condition.

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Type: **Nodal**

New Set Name: **LBC_3**

Input Data...

Force <F1 F2 F3> **<0 -100 0>**

OK

Select Application Region

Select Nodes: **Node 1**

Add

OK

Apply

9. Define the second loading condition.

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Type: **Nodal**

New Set Name: **LBC_4**

Input Data...

Force <F1 F2 F3> **<0 -100 0>**

OK

Select Application Region

Select Nodes: **Node 5**

Add

OK

Apply

10. Create the first load case

◆ **Load Cases**

Load case Name: **Case_1**

Assign/Prioritize Loads/Bcs

Select Individual Loads/BCs: **Displ_LBC_1**
Force_LBC_3

OK

Apply

11. Create the second load case.

◆ **Load Cases**

Action: **Create**

Load case Name: **Case_2**

Assign/Prioritize Loads/Bcs

Select Individual Loads/BCs: **Displ_LBC_2**
Force_LBC_4

OK

Apply

12. Create the third load case.

◆ **Load Cases**

Load Case Name: **Case_3**

Assign/Prioritize Loads/Bcs

Select Individual Loads/BCs: **Displ_LBC_1**, **Force_LBC_3**, **Force_LBC_4**

OK

Apply

13. Create an MSC.Nastran input deck.

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Analysis Deck**

Subcase Select...

Subcases for Solution Sequence: **Case_1**, **Case_2**, **Case_3**

OK

Subcases Create...

Available Subcases: **Case_1**

Output Requests...

Output Requests: **STRESS(SORT1,REAL,...**

Delete

OK

Apply

Available Subcases: **Case_2**

Output Requests...

Output Requests: **SPCFORCES(SORT1,REAL...**

Delete

OK

Apply

Available Subcases: **Case_3**

Output Requests...

Output Requests: **STRESS(SORT1,REAL,...**

Delete

Output Requests: **SPCFORCES(SORT1,REAL...**

Delete

Select Result Type: **Grid Point Force Balance**

OK

Apply

Cancel

Apply

A new input deck has been created with a .bdf extension

The .bdf file should appear as follows:

Figure P.2 - MSC.Nastran input file: “bar.bdf”

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC.Patran Version 7.5 ) on February  10, 1998 at
$ 13:38:00.
ASSIGN OUTPUT2 = 'proj4.op2', UNIT = 12
$ Direct Text Input for File Management Section
$ Linear Static Analysis, Database
SOL 101
TIME 600
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 10-Feb-98 at 12:38:08
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : case_1
   SUBTITLE=case_1
   SPC = 2
   LOAD = 2
   DISPLACEMENT(SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
SUBCASE 2
$ Subcase name : case_2
   SUBTITLE=case_2
   SPC = 4
   LOAD = 4
   DISPLACEMENT(SORT1,REAL)=ALL
   STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 3
$ Subcase name : case_3
   SUBTITLE=case_3
   SPC = 6
   LOAD = 6
   DISPLACEMENT(SORT1,REAL)=ALL
   GPFORCE=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
PARAM    POST    -1
PARAM    PATVER  3.
PARAM    AUTOSPC YES
PARAM    INREL   0
PARAM    ALTRED  NO
PARAM   COUPMASS -1
PARAM    K6ROT   0.
```

The additional sections of the .bdf file appear on the next page.

Figure P.3 - MSC.Nastran input file (cont.): "bar.bdf"

```
PARAM    WTMASS  1.
PARAM,NOCOMPS,-1
PARAM   PRTMAXIM YES
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : bar
PBARL    1      1             BOX
+      A
+      A 2.      4.      .1      .1
CBAR     1       1       1       2       0.      1.      0.
CBAR     2       1       2       3       0.      1.      0.
CBAR     3       1       3       4       0.      1.      0.
CBAR     4       1       4       5       0.      1.      0.
CBAR     5       1       5       6       0.      1.      0.
CBAR     6       1       6       7       0.      1.      0.
CBAR     7       1       7       8       0.      1.      0.
CBAR     8       1       8       9       0.      1.      0.
$ Referenced Material Records
$ Material Record : alum
$ Description of Material : Date: 10-Feb-98           Time:
12:41:14
MAT1     1       1.+7            .3
$ Nodes of the Entire Model
GRID     1               0.      0.      0.
GRID     2               12.5    0.      0.
GRID     3               25.     0.      0.
GRID     4               37.5    0.      0.
GRID     5               50.     0.      0.
GRID     6               62.5    0.      0.
GRID     7               75.     0.      0.
GRID     8               87.5    0.      0.
GRID     9               100.    0.      0.
$ Loads for Load Case : case_1
SPCADD   2       5
LOAD     2       1.      1.      5
$ Loads for Load Case : case_2
SPCADD   4       3
LOAD     4       1.      1.      7
$ Loads for Load Case : case_3
SPCADD   6       5
LOAD     6       1.      1.      5       1.      7
$ Displacement Constraints of Load Set : LBC_2
SPC1     3       123     5
$ Displacement Constraints of Load Set : LBC_1
SPC1     5       23      9
$ Nodal Forces of Load Set : LBC_3
FORCE    5       1       0       100.    0.     -1.      0.
$ Nodal Forces of Load Set : LBC_4
FORCE    7       5       0       100.    0.     -1.      0.
$ Referenced Coordinate Frames
```

Quit MSC.Patran after finishing this exercise.