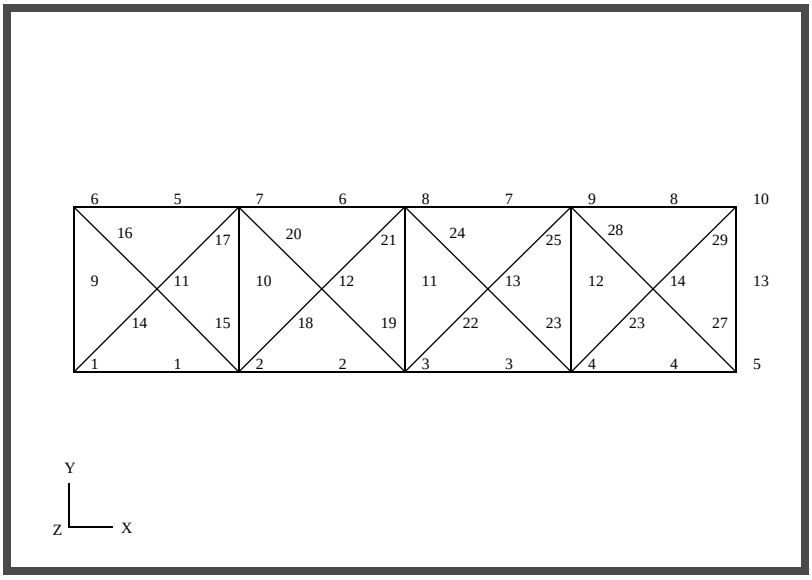
WORKSHOP 23

Linear Static Analysis of a Freebody Truss



Objectives:

- Create a finite element model by explicitly defining node locations and element connectivities.
- Define a MSC.Nastran analysis model comprised of CBAR elements.
- Prepare a MSC.Nastran input file for a linear static analysis.
- Visualize analysis results.

Model Description:

Figure 26.1 is a finite element representation of the truss structure shown on Page 23-1. The nodal coordinates provided are defined in the global cartesian coordinate system (MSC.Nastran Basic system).

The structure is fixed at nodes 1 and 6. There are permanent constraints at all of the nodes. Point forces are applied at Nodes 7, 8, 9, and 10.

Figure 23.1 - Grid Coordinates and Element Condtions

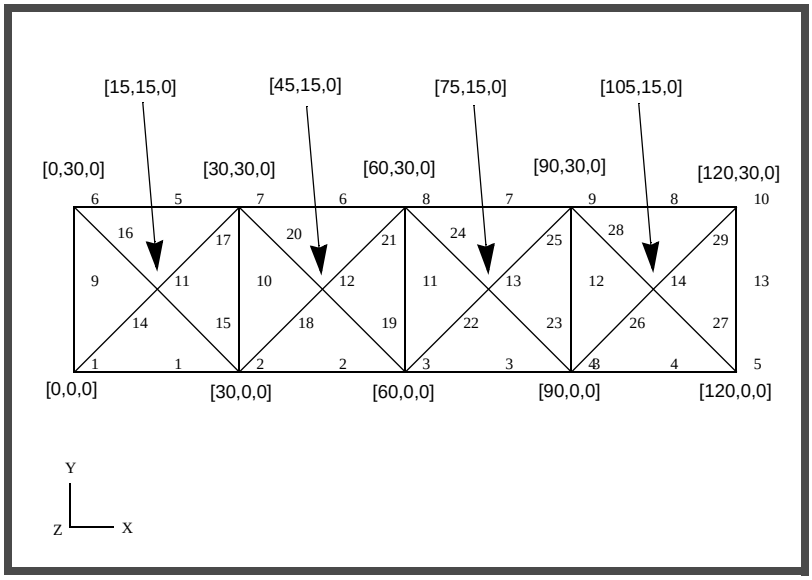


Figure 23.2 - Loads and Boundary Conditions

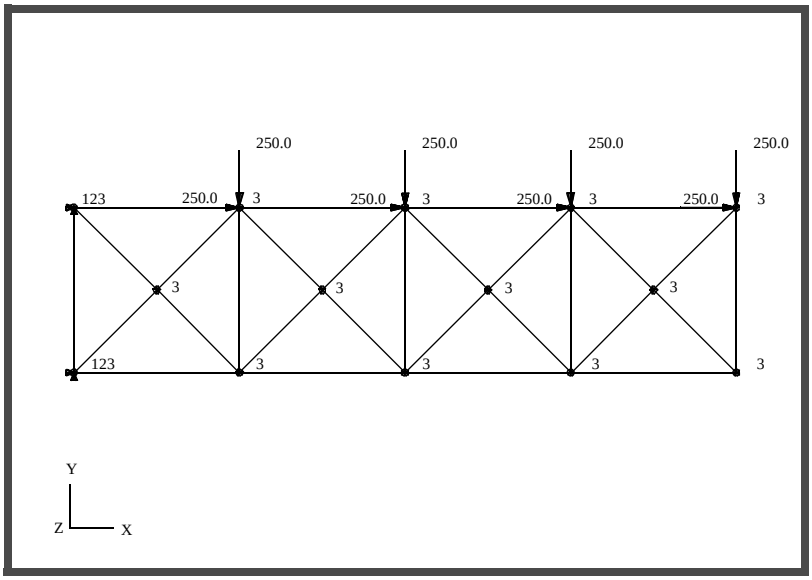
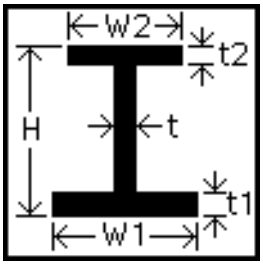


Table 23.1 - Element Properties

Elastic Modulus =	**10E6 lb/in^2**
Poisson's Ratio =	**0.3**



Beam Dimensions		
H		**2.0 in.**
W1		**1.0 in.**
W2		**1.0 in.**
t		**0.1 in.**
t1		**0.1 in.**
t2		**0.1 in.**

Suggested Exercise Steps:

- Open a new database**.**
- Explicitly generate a finite element representation of the truss structure without defining any geometry, i.e., the nodes (GRID) and element connectivities (CBAR) should be defined manually.
- Define material (MAT1) and element (PROD) properties.
- Apply simply-supported boundary constraints (SPC1) and point forces (FORCE).
- Prepare the model for a linear static analysis (SOL 101 and PARAMs).
- Generate an input file and submit it to the MSC.Nastran solver.
- Post-process results.
- Review the results.
- Quit MSC.Patran.

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 11 otherwise, proceed to Step 2.

1. Create a new database called **workshop23.db**

File/New Database

New Database Name **workshop23**

OK

In the ***New Model Preferences*** form set the following:

Tolerance: ◆ **Default**

Analysis code: **MSC/NASTRAN**

OK

 **Show Labels**

1. Create the nodes by manually defining their respective coordinates.

◆ **Finite Elements**

Action: **Create**

Object: **Node**

Method: **Edit**

❑ **Associate with Geometry**

❑ **Auto Execute**

Node Location List: **[0, 0, 0]**

Apply

Repeat the previous operation to create the remaining nodes. Refer to the Figure 23.1 on page 23-3 for the nodal coordinates. . .

Node Location List: **[30, 0, 0]**

Apply

Node Location List: **[60, 0, 0]**

Apply

Node Location List: **[90, 0, 0]**

Apply

Node Location List: **[120, 0, 0]**

Apply

Node Location List: **[0, 30, 0]**

Apply

Node Location List: **[30, 30, 0]**

Apply

Node Location List: **[60, 30, 0]**

Apply

Node Location List: **[90, 30, 0]**

Apply

Node Location List: **[120, 30, 0]**

Apply

Node Location List: **[15, 15, 0]**

Apply

Node Location List: **[45, 15, 0]**

Apply

Node Location List: **[75, 15, 0]**

Apply

Node Location List: **[105, 15, 0]**

Apply

Next, manually define the truss segment connectivites with BAR elements using our newly created nodes. Again, refer to page 23-3 for connectivity information.

◆ **Finite Elements**

Action: **Create**

Object: **Element**

Method: **Edit**

Shape: **Bar**

Topology: **Bar2**

❑ **Auto Execute**

Node 1 = **Node 1**

Node 2 = **Node 2**

Apply

Repeat the previous operation until all the truss segments have been created.

Node 1 = **Node 2**

Node 2 = **Node 3**

Apply

Node 1 = **Node 3**

Node 2 = **Node 4**

Apply

Node 1 = **Node 4**

Node 2 = **Node 5**

Apply

Node 1 = **Node 6**

Node 2 = **Node 7**

Apply

Node 1 = **Node 7**

Node 2 = **Node 8**

Apply

Node 1 = **Node 8**

Node 2 = **Node 9**

Apply

Node 1 = **Node 9**

Node 2 = **Node 10**

Apply

Node 1 = **Node 1**

Node 2 = **Node 6**

Apply

Node 1 = **Node 2**

Node 2 = **Node 7**

Apply

Node 1 = **Node 3**

Node 2 = **Node 8**

Apply

Node 1 = **Node 4**

Node 2 = **Node 9**

Apply

Node 1 = **Node 5**

Node 2 = **Node 10**

Apply

Node 1 = **Node 1**

Node 2 = **Node 11**

Apply

Node 1 = **Node 2**

Node 2 = **Node 11**

Apply

Node 1 = **Node 6**

Node 2 = **Node 11**

Apply

Node 1 = **Node 7**

Node 2 = **Node 11**

Apply

Node 1 = **Node 2**

Node 2 = **Node 12**

Apply

Node 1 = **Node 3**

Node 2 = **Node 12**

Apply

Node 1 = **Node 7**

Node 2 = **Node 12**

Apply

Node 1 = **Node 8**

Node 2 = **Node 12**

Apply

Node 1 = **Node 3**

Node 2 = **Node 13**

Apply

Node 1 = **Node 4**

Node 2 = **Node 13**

Apply

Node 1 = **Node 8**

Node 2 = **Node 13**

Apply

Node 1 = **Node 9**

Node 2 = **Node 13**

Apply

Node 1 = **Node4**

Node 2 = **Node 14**

Apply

Node 1 = **Node 5**

Node 2 = **Node 14**

Apply

Node 1 = **Node 9**

Node 2 = **Node 14**

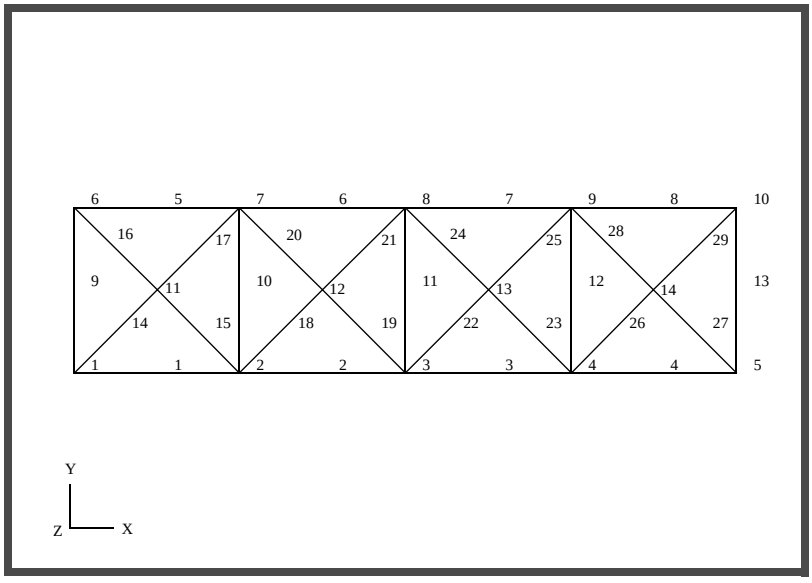
Apply

Node 1 = **Node 10**

Node 2 = **Node 14**

Apply

Figure 23.3



2. Next, define a material using the specified modulus of elasticity and Poisson's ratio.

◆ **Materials**

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

Input Properties ...

Constitutive Model:	**Linear Elastic**
Elastic Modulus =	**10e6**
Poisson Ratio =	**0.3**

OK

Apply

In the *Current Constitutive Models* data box, you will see **Linear Elastic - [„„] - [Active]** appear. Click on **Cancel** to close the form.

Cancel

3. Define element properties for the analysis model.

◆ **Properties**

Action:	**Create**
Dimension:	**1D**
Type:	**Beam**
Property Set Name:	**truss**
Option(s):	**General Section**

Input Properties ...

Material Name:	**m:alum**
Bar Orientation:	**<1, 0.5, 0>**

■ **Associate Beam Section**

Click the beam library icon:

Action:	**Create**
Type:	**Standard Shape**
New Section Name:	**truss_dim**
H	**2**
W1	**1**
W2	**1**
t	**0.1**
t1	**0.1**
t2	**0.1**

OK

OK

Select Members:	**Elm 1:29**

Add

Apply

4. Shrink the elements by 10% for clarity; this allows us to easily assess the element connectivities. Use the **Display/Finite Elements...** option.

Display/Finite Elements...

FEM Shrink:	**0.10**

Apply

Cancel

5. Create two displacement constraints and apply them to the analysis model.

5a. Define the left constraints.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**fixed_end**

Input Data ...

Translations < T1 T2 T3 >	**<0, 0, 0>**
Rotations < R1 R2 R3 >	**< >**

OK

Select Application Region...

Geometry Filter: ◆ **FEM**

Select Nodes: **Node 1,6**

Add

OK

Apply

5b. Define the permanent constraints.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**permanent_constraint**

Input Data ...

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

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

OK

Select Application Region...

Geometry Filter: ◆ **FEM**

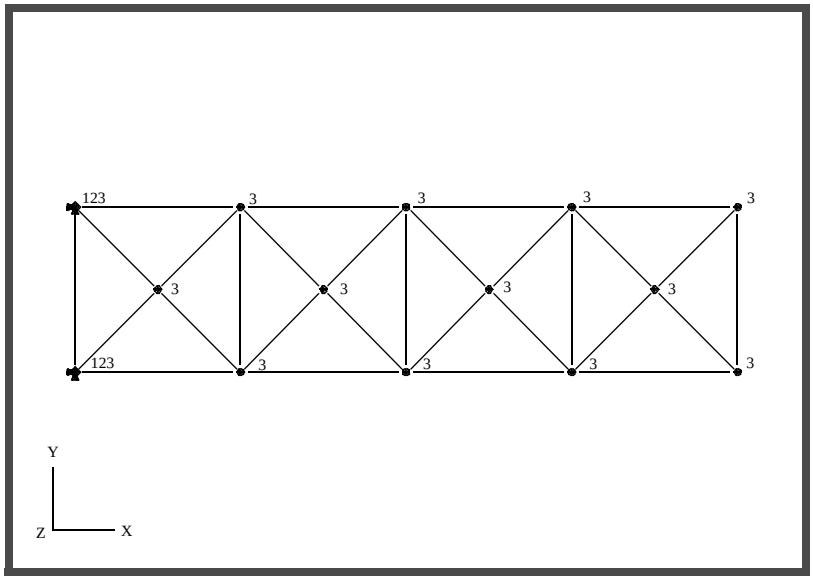
Select Nodes: **Node 1:14**

Add

OK

Apply

Figure 23.4



6. Apply forces to the upper joints of the truss as shown on page 23-3. Vertical forces and horizontal forces of 250 lbs each should be applied at the proper nodes.

6a. First, define the vertical forces.

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Method: **Nodal**

New Set Name: **vertical_load**

Input Data...

Force < F1 F2 F3 > **< , -250, >**

Moment < M1 M2 M3 > **< >**

OK

Select Application Region...

Geometry Filter: ◆ **FEM**

Select Nodes: **Node 7:10**

Add

OK

Apply

6b. Define the horizontal forces.

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Method: **Nodal**

New Set Name: **side_load**

Input Data...

Force < F1 F2 F3 > **<250, 0, 0>**

Moment < M1 M2 M3 > **< >**

OK

Select Application Region...

Geometry Filter: ◆ **FEM**

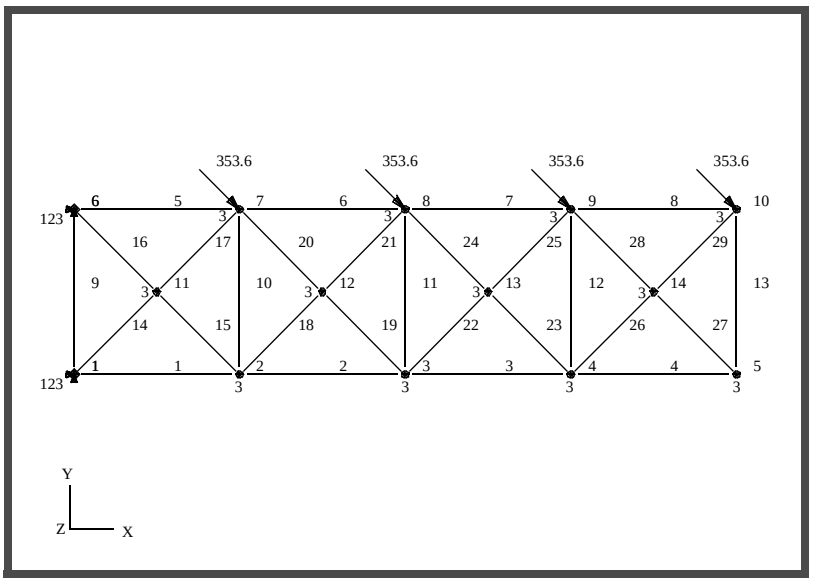
Select Nodes: **Node 7:10**

Add

OK

Apply

Figure 23.5



7. Generate an input file for analysis.

   Click on the **Analysis** radio button on the **Top Menu Bar** and complete the entries as shown here

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name:	**workshop23**

Subcase Create...

Subcase Name	**free_sub**

Output Requests

Form Type	**Basic**

In addition to the default requests, add the following result types to the **Output Requests**.

Select Result Type
*(Click each selection to add to the **Output Requests**)*

Applied Loads
Element Forces
Grid Point Force Balance

Output Requests

DISPLACEMENT(SORT1,REAL)=All FEM
STRESS(SORT1,REAL,VONMISES,BILIN)=All FEM
SPSFORCES(SORT1,REAL)=All FEM
OLOAD(SORT1,REAL)=All FEM
GPFORCE=All FEM
FORCE(SORT1,REAL,BILIN)=All FEM

OK

Apply

Cancel

Subcase Select...

Subcases For Solution Sequence

free_sub
(Click on this to select.)

Subcases Selected:

Default
(Click on this to deselect.)

OK

Apply

A MSC.Nastran input file called **workshop23.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to **step 11**.

Generating an input file for MSC.Nastran Users:

8. MSC.Nastran users can generate an input file using the data from D-3. The result should be similar to the output below (**workshop23.dat**):

```
ID SEMINAR, workshop23
SOL 101
TIME 600
CEND
SEALL = ALL
SUPER = ALL
TITLE = Linear Static Analysis of a Freebody Truss
ECHO = NONE
MAXLINES = 999999999
SUBCASE 1
   SUBTITLE=Default
   SPC = 2
   LOAD = 2
   DISPLACEMENT(SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
   OLOAD(SORT1,REAL)=ALL
   GPFORCE=ALL
   STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
   FORCE(SORT1,REAL,BILIN)=ALL
BEGIN BULK
PARAM    POST    0
PARAM    PATVER  3.
PARAM    AUTOSPC YES
PARAM    INREL   0
PARAM    ALTRED  NO
PARAM   COUPMASS -1
PARAM    K6ROT   0.
PARAM    WTMASS  1.
PARAM,NOCOMPS,-1
PARAM   PRTMAXIM YES

PBARL    1       1               I                                       +       A
+       A 2.      1.      1.      .1      .1      .1
CBAR     1       1       1       2       1.      .5      0.
CBAR     2       1       3       2       1.      .5      0.
CBAR     3       1       3       4       1.      .5      0.
CBAR     4       1       5       4       1.      .5      0.
CBAR     5       1       6       7       1.      .5      0.
CBAR     6       1       8       7       1.      .5      0.
CBAR     7       1       8       9       1.      .5      0.
CBAR     8       1       10      9       1.      .5      0.
CBAR     9       1       1       6       1.      .5      0.
CBAR     10      1       2       7       1.      .5      0.
CBAR     11      1       3       8       1.      .5      0.
CBAR     12      1       4       9       1.      .5      0.
CBAR     13      1       5       10      1.      .5      0.
CBAR     14      1       11      1       1.      .5      0.
CBAR     15      1       11      2       1.      .5      0.
CBAR     16      1       11      6       1.      .5      0.
CBAR     17      1       11      7       1.      .5      0.
```

```
CBAR     18      1       12      2       1.      .5      0.
CBAR     19      1       12      3       1.      .5      0.
CBAR     20      1       12      7       1.      .5      0.
CBAR     21      1       12      8       1.      .5      0.
CBAR     22      1       13      3       1.      .5      0.
CBAR     23      1       13      4       1.      .5      0.
CBAR     24      1       13      8       1.      .5      0.
CBAR     25      1       13      9       1.      .5      0.
CBAR     26      1       14      4       1.      .5      0.
CBAR     27      1       14      5       1.      .5      0.
CBAR     28      1       14      9       1.      .5      0.
CBAR     29      1       14      10      1.      .5      0.
MAT1     1       1.+7            .3
GRID     1               0.      0.      0.
GRID     2               30.     0.      0.
GRID     3               60.     0.      0.
GRID     4               90.     0.      0.
GRID     5               120.    0.      0.
GRID     6               0.      30.     0.
GRID     7               30.     30.     0.
GRID     8               60.     30.     0.
GRID     9               90.     30.     0.
GRID     10              120.    30.     0.
GRID     11              15.     15.     0.
GRID     12              45.     15.     0.
GRID     13              75.     15.     0.
GRID     14              105.    15.     0.
SPCADD   2       1       3
LOAD     2       1.      1.      1       1.      3
SPC1     1       123     1       6
SPC1     3       3       1       THRU    14
FORCE    1       7       0       250.    0.      -1.     0.
FORCE    1       8       0       250.    0.      -1.     0.
FORCE    1       9       0       250.    0.      -1.     0.
FORCE    1       10      0       250.    0.      -1.     0.
FORCE    3       7       0       250.    1.       0.     0.
FORCE    3       8       0       250.    1.       0.     0.
FORCE    3       9       0       250.    1.       0.     0.
FORCE    3       10      0       250.    1.       0.     0.
ENDDATA
```

SUBMITTING THE INPUT FILE FOR MSC.Nastran and MSC.Patran USERS:

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

   9a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran workshop23.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

   9b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran workshop23 scr=yes**. Monitor the run using the UNIX **ps** command.

10. When the run is completed, edit the **workshop23.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.

    10a. While still editing **workshop23.f06**, search for the word:

    D I S P L A C E (spaces are necessary).

Comparison of Results:

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

G R I D P O I N T F O R C E B A L A N C E

	POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
	1		F-OF-SPC	2.500000E+03	4.020370E+02	0.0	0.0	0.0	0.0
	1	1	BAR	-2.075490E+03	-4.462896E+00	0.0	0.0	0.0	-1.678994E+02
	1	9	BAR	-4.756760E+00	0.0	0.0	0.0	0.0	7.305706E+01
	1	14	BAR	-4.197532E+02	-3.975740E+02	0.0	0.0	0.0	9.484234E+01
	1		*TOTALS*	0.0	0.0	0.0	0.0	0.0	1.421085E-14
0	2	1	BAR	2.075490E+03	4.462896E+00	0.0	0.0	0.0	3.401250E+01
	2	2	BAR	-1.153822E+03	-3.418464E+00	0.0	0.0	0.0	-1.076527E+02
B									
	2	10	BAR	-6.077938E+00	-2.519684E+02	0.0	0.0	0.0	8.871161E+01
	2	15	BAR	-5.695641E+02	5.867043E+02	0.0	0.0	0.0	-5.727954E+01
	2	18	BAR	-3.460265E+02	-3.357804E+02	0.0	0.0	0.0	4.220812E+01
	2		*TOTALS*	3.296918E-12	2.046363E-12	0.0	0.0	0.0	-7.105427E-13
0	3	2	BAR	1.153822E+03	3.418464E+00	0.0	0.0	0.0	5.098772E+00
	3	3	BAR	-5.170431E+02	-2.145082E+00	0.0	0.0	0.0	-5.740527E+01
	3	11	BAR	-4.230173E+00	-1.834836E+02	0.0	0.0	0.0	6.389471E+01
	3	19	BAR	-3.996340E+02	4.102234E+02	0.0	0.0	0.0	-3.432991E+01
	3	22	BAR	-2.329142E+02	-2.280132E+02	0.0	0.0	0.0	2.274169E+01
	3		*TOTALS*	-4.774847E-12	-3.495870E-12	0.0	0.0	0.0	4.831691E-13
0	4	3	BAR	5.170431E+02	2.145082E+00	0.0	0.0	0.0	-6.947202E+00
	4	4	BAR	-1.215309E+02	-1.168080E+00	0.0	0.0	0.0	-2.267453E+01
	4	12	BAR	-2.525445E+00	-1.415465E+02	0.0	0.0	0.0	3.793486E+01
	4	23	BAR	-2.639968E+02	2.686620E+02	0.0	0.0	0.0	-1.332614E+01
	4	26	BAR	-1.289900E+02	-1.280925E+02	0.0	0.0	0.0	5.013011E+00
	4		*TOTALS*	-6.195933E-12	3.467449E-12	0.0	0.0	0.0	-1.477929E-12
0	5	4	BAR	1.215309E+02	1.168080E+00	0.0	0.0	0.0	-1.236788E+01
	5	13	BAR	-1.304484E+00	-1.226731E+02	0.0	0.0	0.0	1.638772E+01
	5	27	BAR	-1.202264E+02	1.215050E+02	0.0	0.0	0.0	-4.019836E+00
	5		*TOTALS*	-4.064304E-12	-4.831691E-13	0.0	0.0	0.0	2.955858E-12
0	6		F-OF-SPC	-3.500000E+03	5.979631E+02	0.0	0.0	0.0	0.0
	6	5	BAR	2.876435E+03	-6.260126E+00	0.0	0.0	0.0	-1.990210E+02
	6	9	BAR	4.756760E+00	0.0	0.0	0.0	0.0	6.964574E+01
	6	16	BAR	6.188080E+02	-5.917029E+02	0.0	0.0	0.0	1.293752E+02
	6		*TOTALS*	-1.136868E-13	0.0	0.0	0.0	0.0	2.842171E-14
0	7		APP-LOAD	2.500000E+02	-2.500000E+02	0.0	0.0	0.0	0.0
	7	5	BAR	-2.876435E+03	6.260126E+00	0.0	0.0	0.0	1.121717E+01
	7	6	BAR	1.817139E+03	-2.504817E+00	0.0	0.0	0.0	-9.104840E+01
	7	10	BAR	6.077938E+00	2.519684E+02	0.0	0.0	0.0	9.362652E+01
	7	17	BAR	3.705092E+02	4.025726E+02	0.0	0.0	0.0	-1.657279E+02
	7	20	BAR	4.327089E+02	-4.082963E+02	0.0	0.0	0.0	1.519326E+02
	7		*TOTALS*	-7.958079E-13	6.821210E-13	0.0	0.0	0.0	-3.979039E-13
0	8		APP-LOAD	2.500000E+02	-2.500000E+02	0.0	0.0	0.0	0.0
	8	6	BAR	-1.817139E+03	2.504817E+00	0.0	0.0	0.0	1.590390E+01
	8	7	BAR	9.683797E+02	-1.916859E+00	0.0	0.0	0.0	-5.437086E+01
	8	11	BAR	4.230173E+00	1.834836E+02	0.0	0.0	0.0	6.301048E+01
0									

G R I D P O I N T F O R C E B A L A N C E

	POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
	8	21	BAR	3.129516E+02	3.338533E+02	0.0	0.0	0.0	-1.122981E+02
	8	24	BAR	2.815776E+02	-2.679249E+02	0.0	0.0	0.0	8.775459E+01
	8		*TOTALS*	-5.002221E-12	-5.684342E-13	0.0	0.0	0.0	-1.563194E-12
0	9		APP-LOAD	2.500000E+02	-2.500000E+02	0.0	0.0	0.0	0.0
	9	7	BAR	-9.683797E+02	1.916859E+00	0.0	0.0	0.0	-3.134907E+00
	9	8	BAR	3.735620E+02	-4.238446E-01	0.0	0.0	0.0	-1.474371E+01
	9	12	BAR	2.525445E+00	1.415465E+02	0.0	0.0	0.0	3.782850E+01
	9	25	BAR	2.153333E+02	2.272760E+02	0.0	0.0	0.0	-6.798224E+01
	9	28	BAR	1.269589E+02	-1.203155E+02	0.0	0.0	0.0	4.803236E+01
	9		*TOTALS*	2.771117E-12	-9.904966E-12	0.0	0.0	0.0	0.0
0	10		APP-LOAD	2.500000E+02	-2.500000E+02	0.0	0.0	0.0	0.0
	10	8	BAR	-3.735620E+02	4.238446E-01	0.0	0.0	0.0	2.028374E+00
	10	13	BAR	1.304484E+00	1.226731E+02	0.0	0.0	0.0	2.274679E+01
	10	29	BAR	1.222575E+02	1.269030E+02	0.0	0.0	0.0	-2.477517E+01
	10		*TOTALS*	8.242296E-13	4.263256E-12	0.0	0.0	0.0	2.273737E-12
0	11	14	BAR	4.197532E+02	3.975740E+02	0.0	0.0	0.0	2.378447E+02
	11	15	BAR	5.695641E+02	-5.867043E+02	0.0	0.0	0.0	-1.998237E+02
	11	16	BAR	-6.188080E+02	5.917029E+02	0.0	0.0	0.0	2.772019E+02
	11	17	BAR	-3.705092E+02	-4.025726E+02	0.0	0.0	0.0	-3.152229E+02
	11		*TOTALS*	5.684342E-13	6.821210E-13	0.0	0.0	0.0	-4.547474E-13
0	12	18	BAR	3.460265E+02	3.357804E+02	0.0	0.0	0.0	1.114823E+02
	12	19	BAR	3.996340E+02	-4.102234E+02	0.0	0.0	0.0	-1.245104E+02

```
         12           20    BAR          -4.327089E+02   4.082963E+02   0.0         0.0         0.0          2.142561E+02
         12           21    BAR          -3.129516E+02  -3.338533E+02   0.0         0.0         0.0         -2.012281E+02
         12                 *TOTALS*     -3.012701E-12   5.684342E-13   0.0         0.0         0.0         -7.958079E-13
0        13           22    BAR           2.329142E+02   2.280132E+02   0.0         0.0         0.0          5.077373E+01
         13           23    BAR           2.639968E+02  -2.686620E+02   0.0         0.0         0.0         -5.665258E+01
         13           24    BAR          -2.815776E+02   2.679249E+02   0.0         0.0         0.0          1.170371E+02
         13           25    BAR          -2.153333E+02  -2.272760E+02   0.0         0.0         0.0         -1.111583E+02
         13                 *TOTALS*      4.092726E-12   5.684342E-12   0.0         0.0         0.0         -2.273737E-13
0        14           26    BAR           1.289900E+02   1.280925E+02   0.0         0.0         0.0          8.448993E+00
         14           27    BAR           1.202264E+02  -1.215050E+02   0.0         0.0         0.0         -1.515966E+01
         14           28    BAR          -1.269589E+02   1.203155E+02   0.0         0.0         0.0          5.161876E+01
B
         14           29    BAR          -1.222575E+02  -1.269030E+02   0.0         0.0         0.0         -4.490808E+01
         14                 *TOTALS*      1.094236E-12  -1.448086E-11   0.0         0.0         0.0          0.0
```

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

13. Proceed with the Reverse Translation process, that is, attaching the **workshop23.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

Action:	**Attach XDB**
Object:	**Result Entities**
Method:	**Local**

Select Results File...

Selected Results File	**workshop23.xdb**

OK

Apply

Clean up the display using the **Reset Graphics** tool.

Reset Graphics

14. When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

Deactivate the entity labels by selecting the **Hide Labels** button on the toolbar.

Hide Labels

◆ **Results**

Action:	**Create**
Object:	**Freebody**
Method:	**Loads**

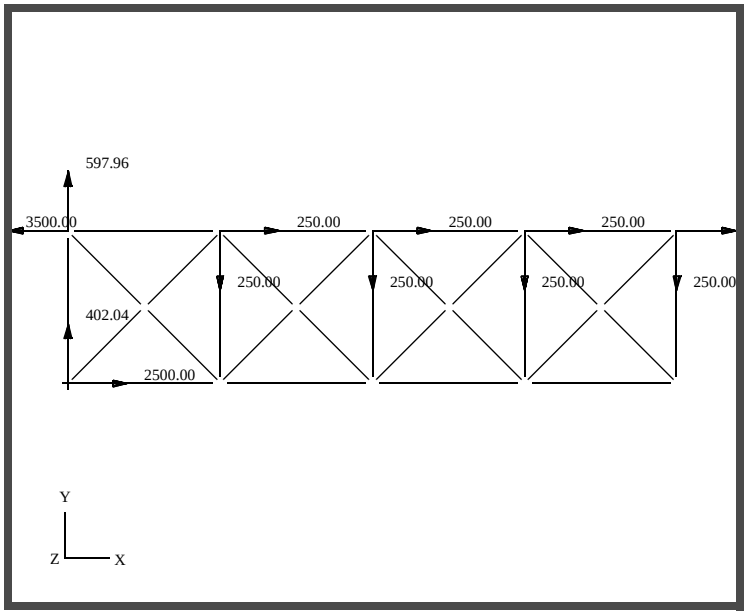
Select Results

Select Result Case: **Default**

Select Result Type: **Freebody Loads**

Apply

Figure 23.6:



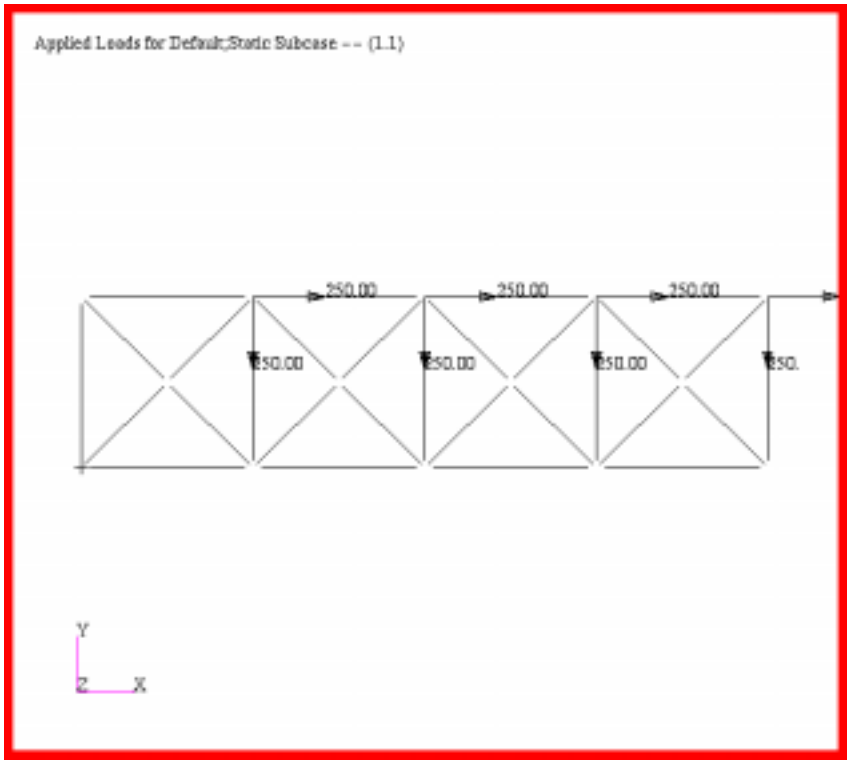
Notice that MSC.Patran plots the combination SPC forces and OLOADs for the entire model.

Select Result Case: **Default**

Select Result Type: **Applied Loads**

Apply

Figure 23.7



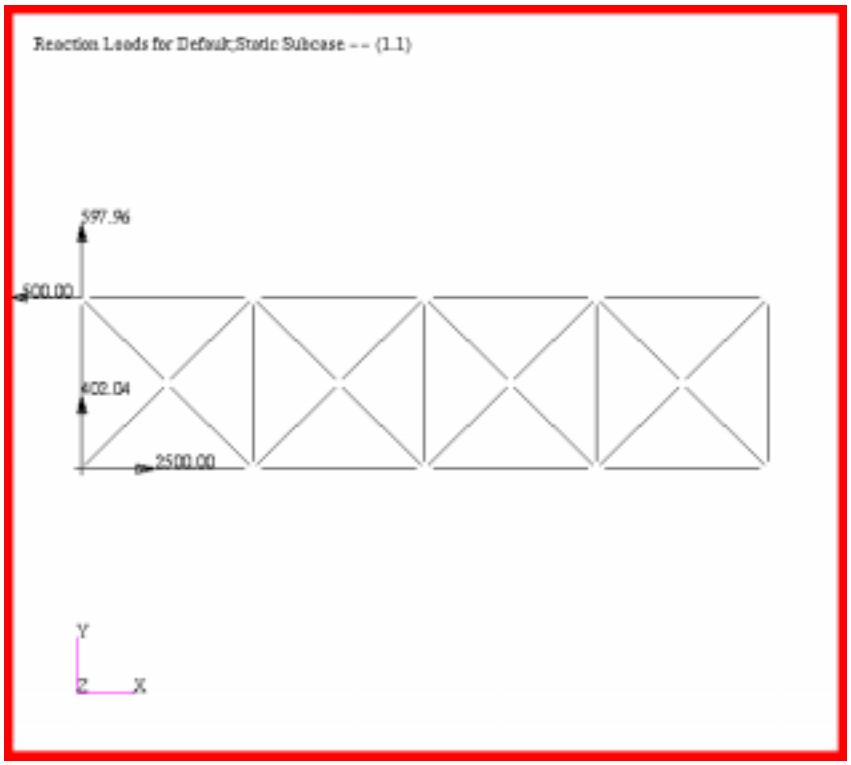
Notice that MSC.Patran plots the OLOADs for the entire model.

Select Result Case: **Default**

Select Result Type: **Reaction Loads**

Apply

Figure 23.8



Notice that MSC.Patran plots the SPC forces for the entire model.

15. In addition to plotting the SPC force and/or OLOADs for the entire model, MSC.Patran can also process grid point force data for selected elements. Bring up the **Results** form again.

◆ **Results**

Action:	**Create**
Object:	**Freebody**
Method:	**Loads**

Select Results

Select Result Case:	**Default**
Select Result Type:	**Internal Loads**

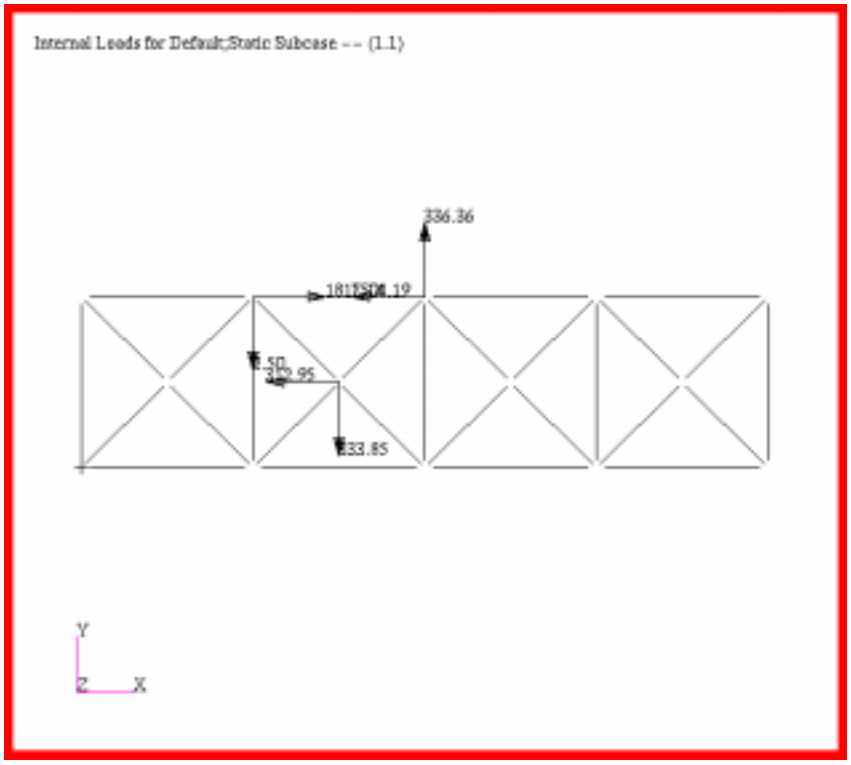
Select Entities

Select By:	**Element**
Select Elements:	**Elm 6,21**

Add

Apply

Figure 23.9



Notice that MSC.Patran plots the summation of forces for Elements 6 and 21 at node 8. Additionally, MSC.Patran plots the forces on Element 6 at Node 7 and the forces on Element 21 at Node 12. (The node locations can be found in Figure 23.1)

16. Create a similar a similar plot for the moments on Elements 6 and 21.

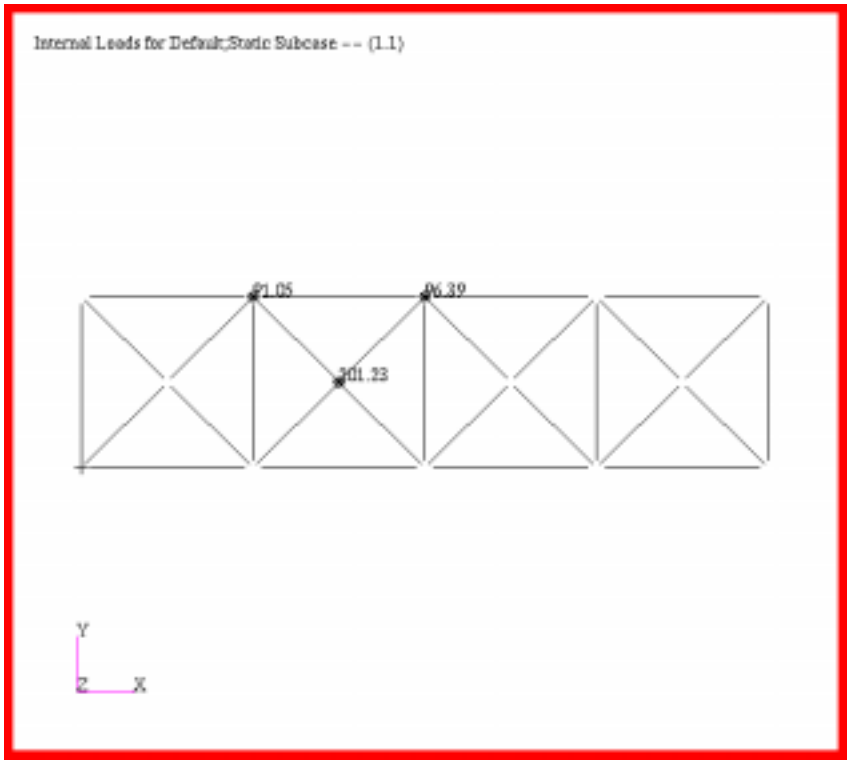
Display Attributes

Show: **Moment**

Apply

The display should resemble Figure 23.10.

Figure 23.10



17. Create similar plots for external loads on Elements 6 and 21.

◆ **Results**

Action: **Create**

Object: **Freebody**

Method: **Loads**

Select Results

Select Result Case: **Default**

Select Result Type: **External Loads**

Select Entities

Select By: **Element**

Select Elements: **Elm 6,21**

Add

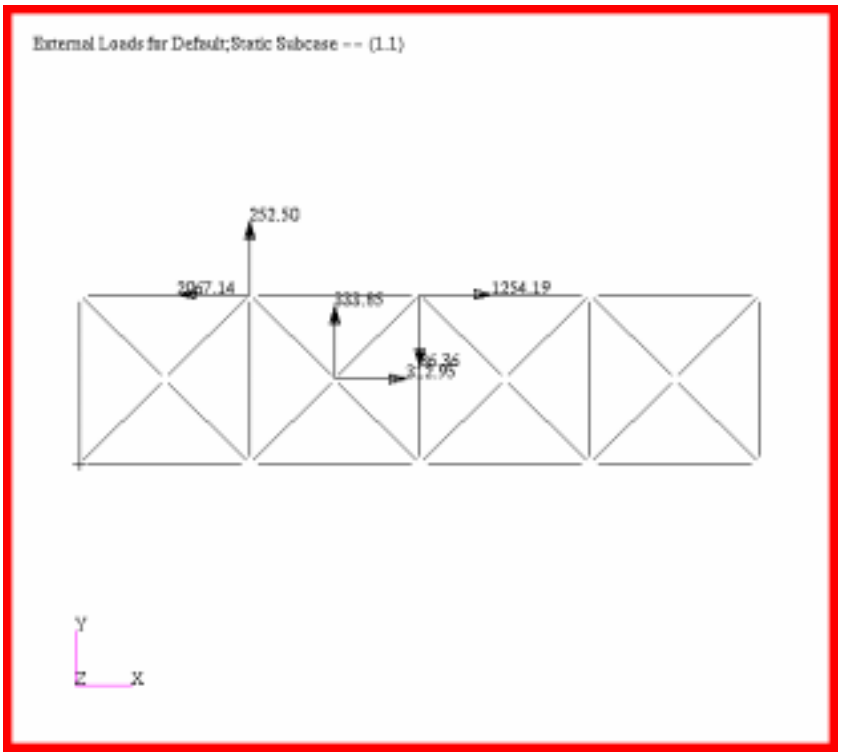
Display Attributes

Show: **Force**

Apply

The display should resemble Figure 23.11.

Figure 23.11



Notice once again that MSC.Patran plots the summation of forces acting on Elements 6 and 21 at Node 8 and the forces on Element 6 were plotted at Node 7 and the forces on Element 21 were plotted at Node 12. Also notice that NO applied loads and SPC forces were added to the summation. (The node locations can be found in Figure 23.1)

18. MSC.Patran can also calculate interface loads at any user defined location.

◆ **Results**

Action:	**Create**
Object:	**Freebody**
Method:	**Interface**

Select Results

Select Result Case:	**Default**

Select Result Type: **Internal Loads**

Select Entities

Select By: **Element**

Select Elements: **Elm 6,21**

Add

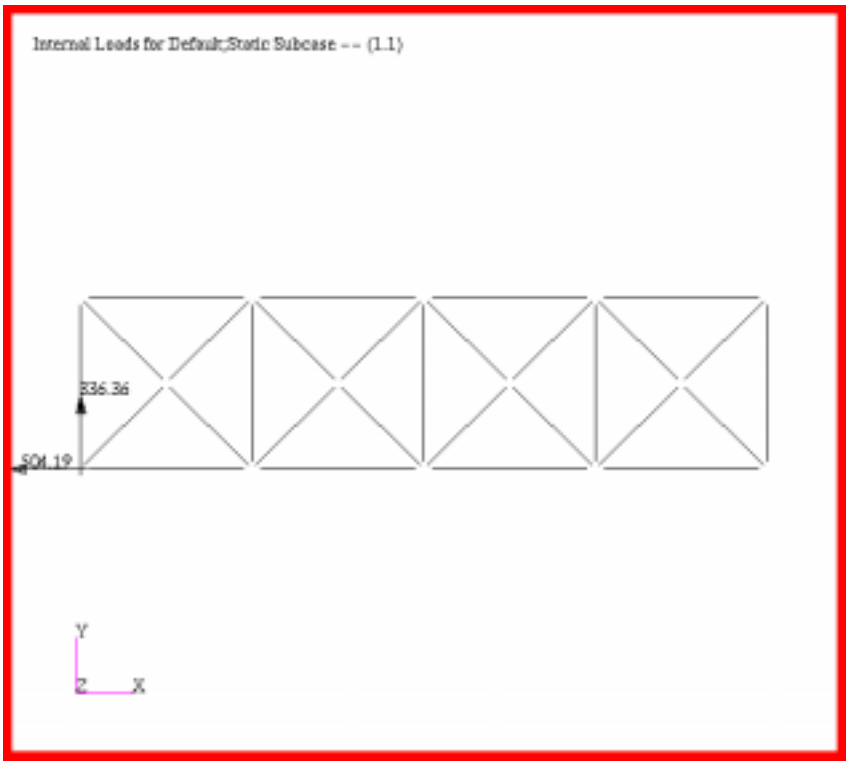
Select By: **Node**

Select Nodes: **Node 8**

Add

Apply

Figure 23.12



Notice that MSC.Patran plots the summation of forces for Elements 6 and 21 at the default setting [0 0 0}. For a similar plot of the interface moments:

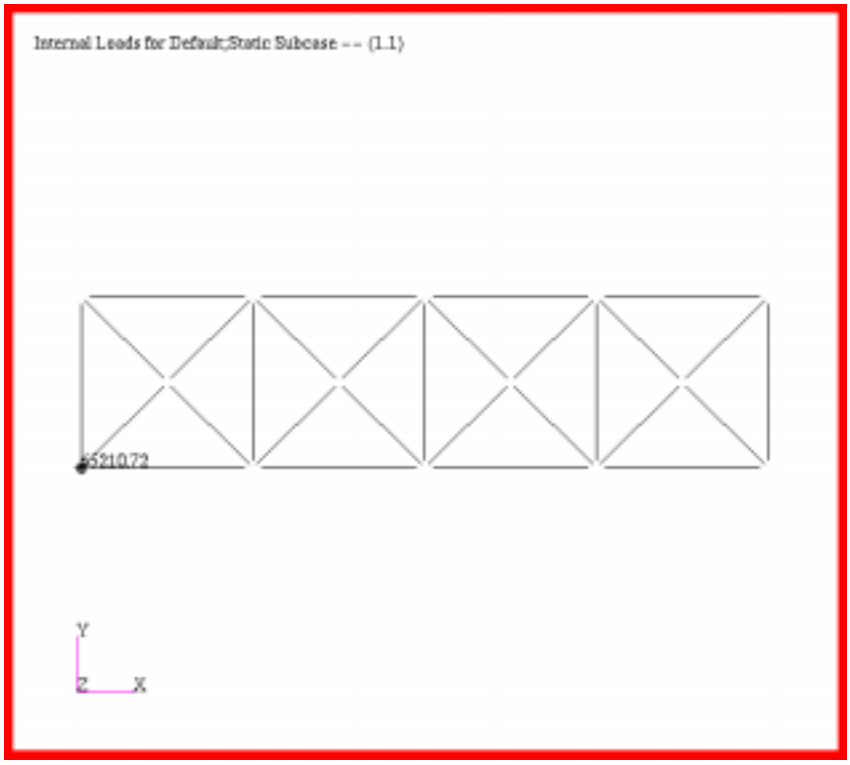
Display Attributes

Show: **Moment**

Apply

The display should resemble Figure 23.13.

Figure 23.13:



Plot the interface forces at a different location.

◆ **Results**

Action: **Create**

Object: **Freebody**

Method: **Interface**

Select Results

Select Result Case: **Default**

Select Result Type: **Internal Loads**

Summation Point: **[10 0 0]**

Select Entities

Select By: **Element**

Select Elements: **Elm 6,21**

Add

Select By: **Node**

Select Nodes: **Node 8**

Add

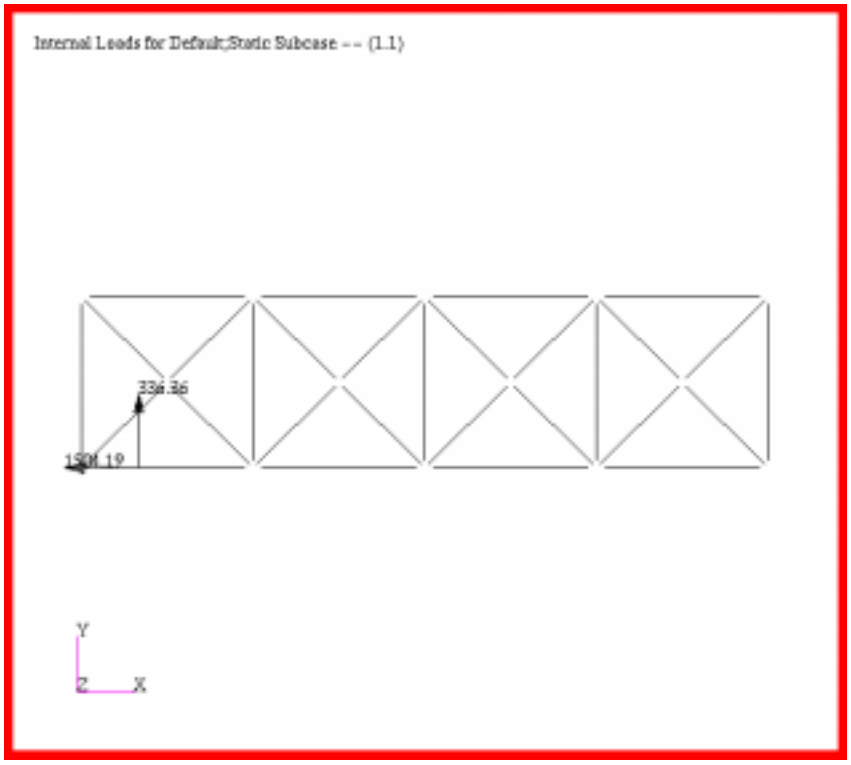
Display Attributes

Show: **Force**

Apply

The display should resemble Figure 23.14.

Figure 23.14



Notice that MSC.Patran plots the summation of forces for Elements 6 and 21 at the user defined setting [10 0 0]. To see a similar plot for the interface moments:

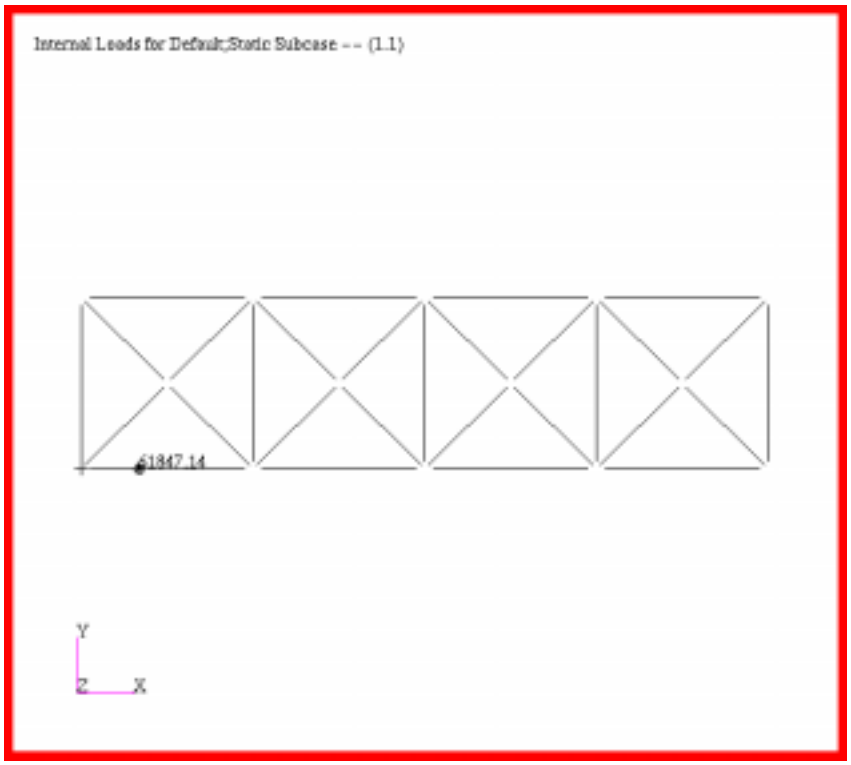
Display Attributes

Show: **Moment**

Apply

The display should resemble Figure 23.15.

Figure 23.15



Notice the difference in calculated combined moment.

19. Reset graphics using the icon on the main toolbar.

Reset Graphics

Quit MSC.Patran after finishing this exercise.