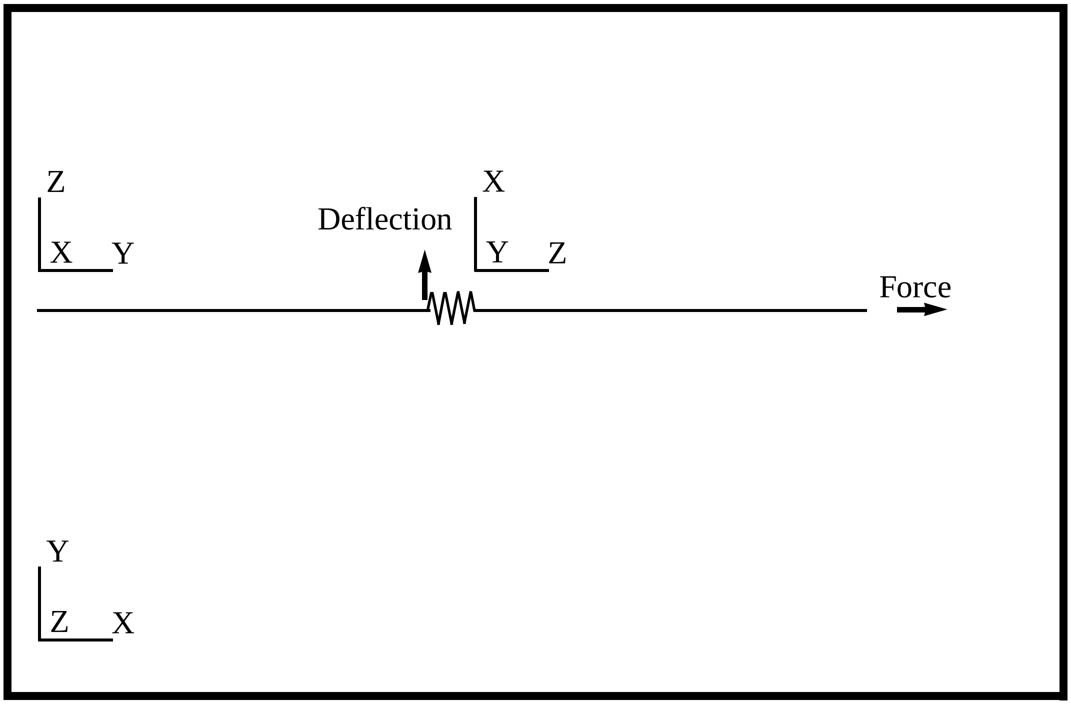
WORKSHOP 38

Debugging of Common Errors of CELAS (SPRING)



Objectives:

- Create two bars and connect with a spring
- Demonstrate the improper and proper ways to use the spring.
- Obtain results with proper use of the spring
- Obtain results with improper use of the spring and get invalid answers

Model Description:

In this exercise, we are going to investigate the use and mis-use of spring elements (CELAS elements). The CELAS element is a scalar element, and this means that it can be used for only one degree of freedom (DOF). That DOF is defined by the output displacement coordinate system of the nodes it connects.

This exercise will go through several steps to show you how you can get FATAL errors when you mis-use the spring elements, and it will also show you how to properly build the model shown in Figure 38.1. Finally, this exercise will show you how you can get MSC.Nastran to run without FATAL errors but with completely invalid answers.

The model below is two beams that are connected with a spring. The beam on the left is fixed on it’s left end, and an axial load is applied to the right end of the beam on the right. We want to build a spring element between the two beams that will carry this axial load to the reaction point on the left.

Figure 38.1 - Elements, Nodes, and Coordinate Systems

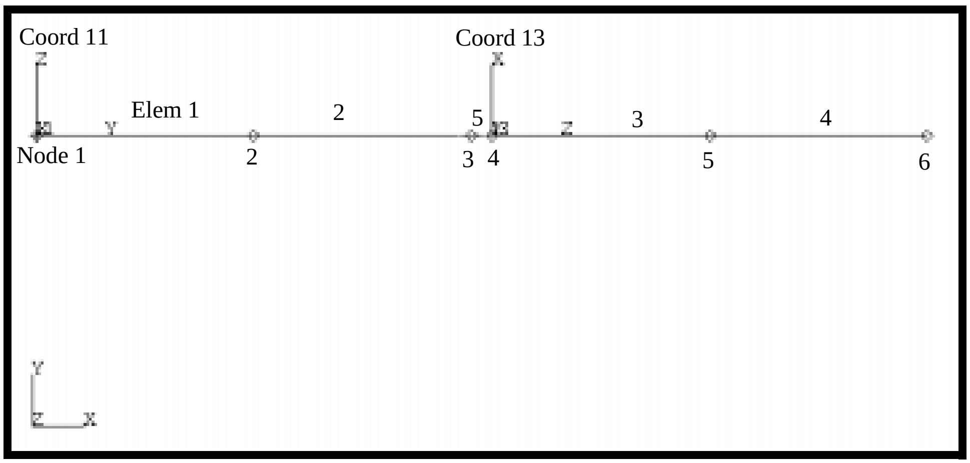


Table 38.1 - Materials

Elastic Modulus:	**10E6**
Poisson Ratio;	**0.3**

Table 38.2 - Properties

Area:	**10**
Inertia I_1:	**100**

Inertia I_2:	**100**
Torsional Constant:	**0.1**

Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0 and begin to create a new model.

   Double click on the icon labeled **MSC.Nastran for Windows V4.0**.

   On the *Open Model File* form, select **New Model.**

Open Model File:	**New Model**

2. Make the two coordinate systems for the beams. These coordinate systems are being built for demonstration purposes only. We are trying to show the effects of different output displacement coordinate systems (Analysis coordinate system) on the nodes.

   Coordinate 11

Model/Coord Sys...

Id:	**11**
Title:	**Bar 1**
Method:	● **XY Locate**

OK

X:	*Y:*	*Z:*	
0	**0**	**0**	**OK**
0	**0**	**1**	**OK**
1	**0**	**0**	**OK**

Coordinate 13

Id:	**13**
Title:	**Bar 2**
Method:	● **XY Locate**

OK

X:	*Y:*	*Z:*	
21	**0**	**0**	**OK**
21	**1**	**0**	**OK**
21	**0**	**1**	**OK**

Cancel

Click on **View/AutoScale** to see both coordinate systems.

View/AutoScale <Ctrl+A>

Turn off the workplane.

Tools/Workplane... <F2>

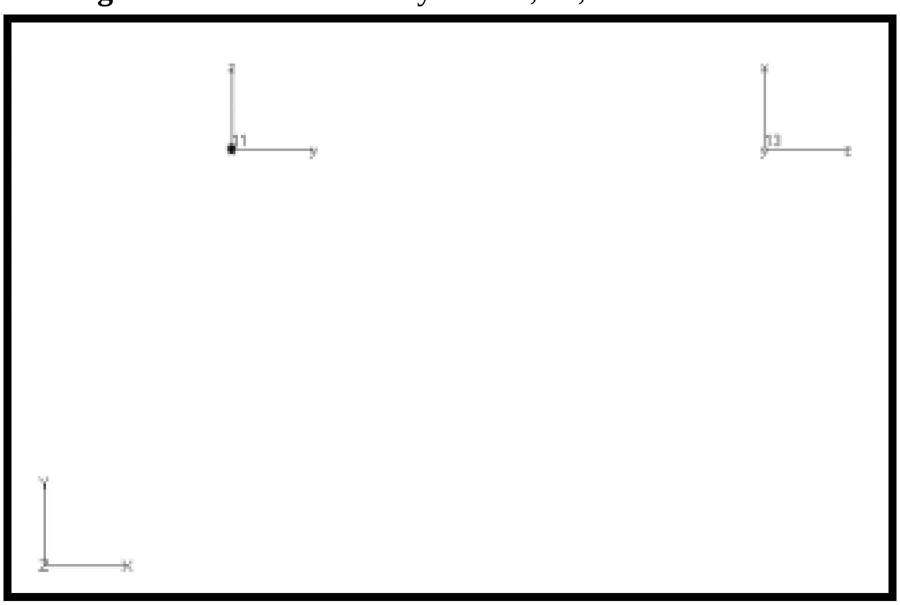
☐ **Draw Workplane**

Done

View/Regenerate... <Ctrl+G>

The Model should look similar to Figure 38.2.

Figure 38.2 - Coordinate Systems 1, 11, and 13



Notice that the y-axis of Coord 11 lines up with the z-axis of Coord 13.

3. Create material mat_1.

Model/Material...

Title:	**Mat_1**
Youngs Modulus, E:	**10E6**
Poisson's Ratio, nu:	**0.3**

OK

Cancel

4. Create the properties for the spring and bars.

First the spring:

Model/Property...

Title:	**Spring**

Elem/Property Type...

Line Elements:	● **DOF Spring**

OK

End A:	● **TX**
End B:	● **TX**
Stiffness:	**100**

OK

Now the bars:

Title:	**Bar**

Elem/Property Type...

Line Elements:	● **Bar**

OK

Material:	**1..Mat_1**
Area, A:	**10**
I1 or Izz:	**100**
I2 or Iyy	**100**
Torsional Constant, J:	**0.1**

OK

Cancel

5. Create the nodes for the two beams.

   First, the nodes under coordinate 11.

   Model/Node...

   Change the Node Definition CSys and the Output CSys for this demonstration.

CSys:	**11..Bar 1**

Parameters...

Output Coordinate System:	**11..Bar 1**

OK

X:	*Y:*	*Z:*	
0	**0**	**0**	**OK**
0	**10**	**0**	**OK**
0	**20**	**0**	**OK**

Now, the nodes under coordinate 13.

CSys:	**13..Bar 2**

Parameters...

Output Coordinate System:	**13..Bar 2**

OK

X:	*Y:*	*Z:*	
0	0	0	**OK**
0	0	10	**OK**
0	0	20	**OK**

Cancel

View/Autoscale **<Ctrl+A>**

6. Connect the nodes with elements.

Model/Element...

Property: **2..Bar**

Type

● **Bar**

OK

Orientation: **Vector**

CSys: **0..Basic Rectangular**

When prompted, “Active Coordinate System Changed to Coordinate System 0. Vector is not defined.” choose OK.

OK

	X:	*Y:*	*Z:*
Base:	0	0	0
Tip	0	1	0

OK

Nodes: **1** **2** **OK**

Nodes: **2** **3** **OK**

Nodes: **4** **5** **OK**

Nodes: **5** **6** **OK**

Type...

Line Elements: ● **DOF Spring**

OK

Property: **1..Spring**

Nodes: **3** **4** **OK**

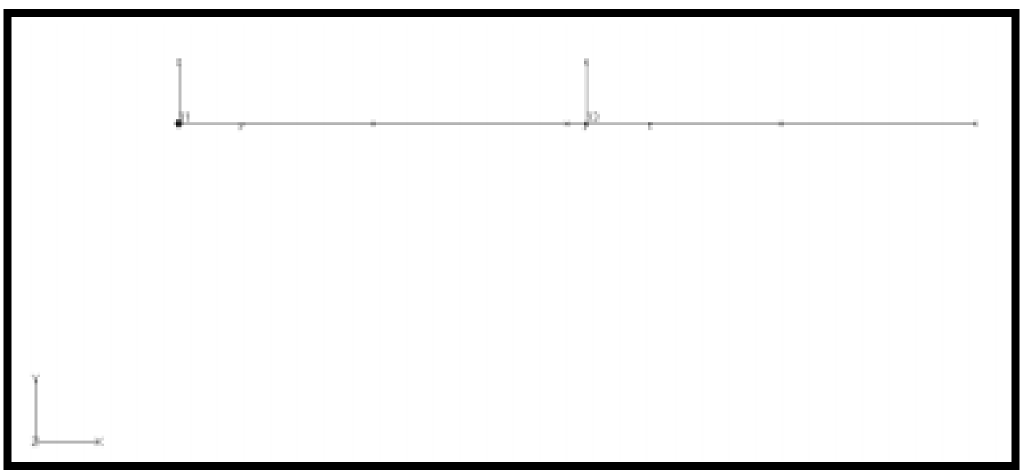
Cancel

Click on **View/AutoScale** to see the beams and spring.

View/AutoScale <Ctrl+A>

The Model should look similar to Figure 38.3.

Figure 38.3 - Beams and Spring elements



7. Create the constraint and axial load.

Model/Constraint/Set...

Title: **Constraint**

OK

Model/Constraint/Nodal...

Select Node 1 (at the left end of the model)

	+1
OK	
Fixed	
Coord Sys:	**11..Bar 1**
OK	
Cancel	

Model/Load/Set...

Title:	**Axial_Load**
OK	

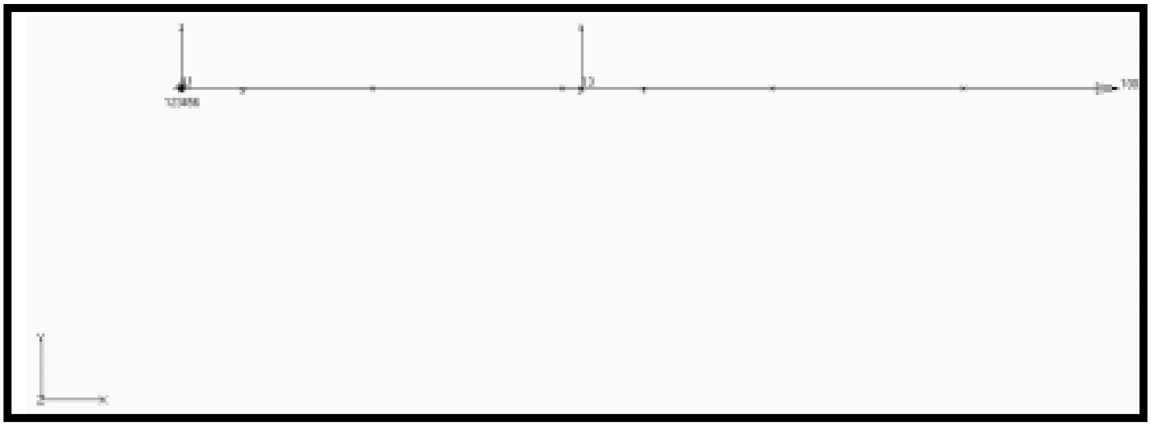
Model/Load/Nodal...

Select Node 6
(at the right end of the model)

	+6
OK	
Coord Sys:	**13..Bar 2**
FZ:	**100**
OK	
Cancel	

The Model should look similar to Figure 38.4.

Figure 38.4 - Loads and Boundary Conditions



8. Now, the model is ready for analysis.

File/Export/Analysis Model...

Type:	**1..Static**
OK	
File name:	**Run1**
Write	

Under Additional Info section:

☒ **Run Analysis**

OK

When prompt, “OK to Save Model Now” choose **Yes**

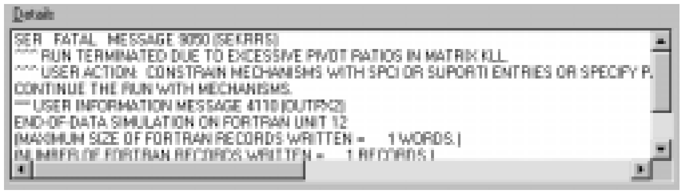
Yes	
File name:	**Prob38**
Save	

When the MSC.Nastran manager is through running, MSC.Nastran will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since there is a fatal message, this means the analysis failed. So choose **Show Details** to see the fatal message.

Show Details

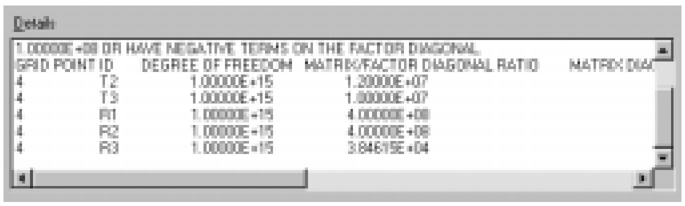
Under the Message dialog box, highlight “^^^ USER FATAL MESSAGE 9050(SEKRRS)” and this message should appear.

Message 41-1



Now highlight “*** USER WARNING MESSAGE 4698 (DCMPD)’ and this message should appear.

Message 41-2



Explanation of Error:

This model fails with excessive pivot ratios. Excessive pivot ratios is a numerical method term that means the model has poor constraints or disconnected load paths.

For this model, we connected the x DOF at both of the nodes on the spring element. We did this in Step 4 when we created the element property for the DOF spring. It is a common mistake to connect the Basic Coordinate System (Coord 0) degrees of freedom when using spring, but remember, the DOF for a spring is defined in the nodes’ output coordinate system. The x DOF of the left node is out of the plane, and the x DOF of the node on the right is in the vertical direction.

9. Modify the spring property.

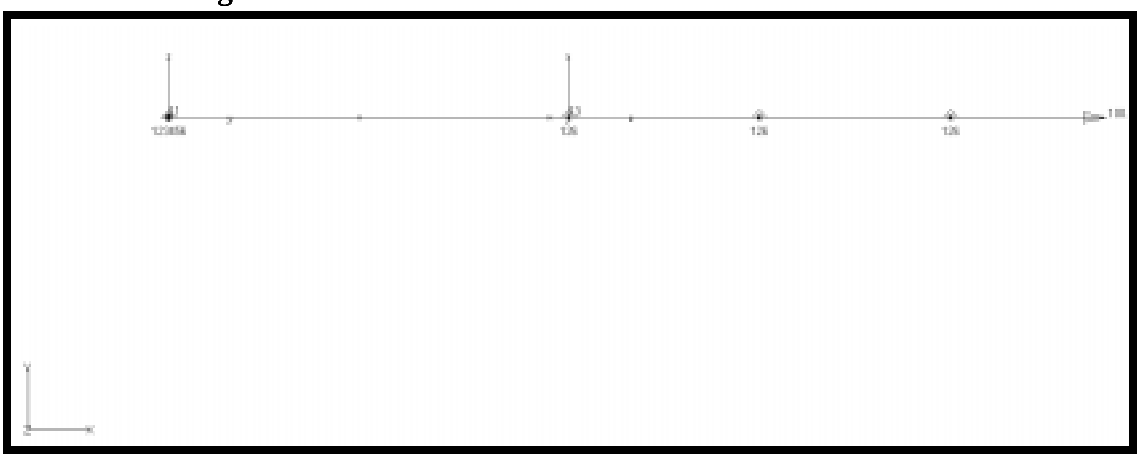
Now we will connect the proper DOF for both nodes. We want to connect the DOF that is aligned with the basic X direction (the direction of the axial load). For the left node, we want the local Y direction and for the node on the right, we want the local Z direction.

Modify/Edit/Property...

ID:	**1**
More	
OK	
End A:	● **TY**
End B:	● **TZ**
OK	

The model should look similar to Figure 38.5.

Figure 38.5 - Modified Constraints



10. Now we are ready to submit the file for analysis.

File/Export/Analysis Model...

Type:	**1..Static**
OK	
File name:	**Run2**
Write	

Under Additional Info section:

☒ **Run Analysis**

OK

When prompt, “OK to Save Model Now” choose **Yes**

Yes

When the MSC.Nastran manager is through running, MSC.Nastran will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since there is a fatal message, this means the analysis failed. So choose **Show Details** to see the fatal message.

Show Details

Under the Message dialog box, highlight the fatal and warning message and see similar messages to Message 1 and 2.

Explanation of error:

This model fails again because of excessive pivot ratio. The load path is correct, but the model is not fully constrained. The beam on the right exhibits mechanisms or rigid body motion. The two translational degrees of freedom orthogonal to the axial direction are not constrained, and the rotational degrees of freedom are also not constrained. Even though there is no load in these directions, you must constrain rigid body motion. This is necessary only from a numerical point of view and serves no practical purpose (except to make the model run).

11. Constraint the rigid body motion of the second beam.

Remember, the constraints are also defined in the output coordinate system of the nodes.

Model/Constraint/Nodal...

Select Node 4, 5, and 6

+4
+5
+6

OK

Coord Sys: **13..Bar 2**

DOF:

☒ **TX** ☒ **TY** ☐ **TZ**

☐ **RX** ☐ **RY** ☒ **RZ**

OK

Cancel

12. Now, the model is ready for analysis.

File/Export/Analysis Model...

Type: **1..Static**

OK

File name: **Run3**

Write

Under Additional Info section:

☒ **Run Analysis**

OK

When prompt, “OK to Save Model Now” choose **Yes**

Yes

When the MSC.Nastran manager is through running, MSC.Nastran will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis went through without a fatal error, choose **Continue**. .

Continue

Successful run: Third part of this exercise shows the proper model definition for this simple model. As you can see, there are many traps you can fall into when dealing with spring elements and output coordinate systems.

Hand Calculations:

$$\Delta max = \frac{P}{K_{spring}} + \frac{PL_1}{AE} + \frac{PL_2}{AE} = \frac{100}{100} + \frac{100(20 + 20)}{10(10 \times 10^6)} = 1.0004$$

$$\Delta max = 1.0004$$

L_1 and L_2 are the lengths of the two beams.

The calculation is for springs in a series since the beams act like axial springs. $A = \sum \frac{1}{K_i}$

SPC Forces at Grid 1:
T2 direction, -100

Bar Forces at Elem 1, 2, 3, 4
No moment, axial =100

Spring force at Elm 5
-100

Check these results in your model to verify all the changes.

13. Use **Quick Option** to set the model to display Nodes and Elements only.

Quick Options... **<Ctrl+Q>**

All Entities Off

☒ **Node**

☒ **Element**

Done

14. Click on the Off box at the lower right hand corner and choose Node. By doing so, highlighting a node with the cursor will display results in detail.

Off

Node...

15. To see the total translation of the model.

View/Select... **<F5>**

Deformed Style: ● **Deform**

Deformed and Contour Data...

Output Set: **1..MSC/NASTRAN Case 1**

Deformation: **1..Total Translation**

OK

OK

Place the cursor near node 6 and compare the maximum deformation to the one in the Hand Calculation.

View/Select... **<F5>**

Deformed Style: ● **Deform**

Deformed and Contour Data...

Deformation: **51..Total Constraint Force**

OK

OK

Place the cursor on node 1 and compare the maximum force to Hand Calculation results.

View/Select... **<F5>**

Deformed Style: ● **None--Model Only**

Contour Style: ● **Criteria**

Deformed and Contour Data...

Contour: **3008..Bar EndA Axial Force**

OK

OK

Click on the Node box on the bottom right hand corner and choose Element.

Node

Element...

Move mouse over an element, and compare the axial force to the hand calculations.

View/Select... **<F5>**

Deformed and Contour Data...

Contour: **3028..spring Axial Force**

OK

OK

Place the cursor on Element 5 and compare the Spring Force with the Hand Calculation Results.

16. Modify the spring property.

Now we will change the model in what may appear to be a simple mistake, but what you will see in the results (MSC.Nastran will generate results) are completely invalid answers.

Modify/Edit/Property...

ID:	**1**
More	
OK	
End A:	● **TZ**
End B:	● **TZ**
OK	

17. Now, the model is ready for analysis.

File/Export/Analysis Model...

Type:	**1..Static**
OK	
File name:	**Run4**
Write	

Under Additional Info section:

☒ **Run Analysis**

OK

When prompt, “OK to Save Model Now” choose **Yes**

Yes

When the MSC.Nastran manager is through running, MSC.Nastran will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis went through without a fatal error, choose **Continue**. .

Continue

Explanation of results:

In this model, we connected the Z DOF for both nodes. The problem is that for the node on the left, this degree is in the vertical direction (not axial). Because MSC.Nastran is only treating the spring element as a mathematical connection of a specific DOF the axial DOF on the right node is loading the vertical DOF of the left node. This vertical loading will cause bending in the beam on the left, and this is an impossible mechanical situation.

Hand Calculations:

Displacement:

@ Grid 3

$$\Delta(\textbf{due to shear}) = \frac{\mathbf{PL^3}}{\mathbf{3EI}} = \frac{(\mathbf{100})(\mathbf{20})^{\mathbf{3}}}{\mathbf{3}(\mathbf{10}\times 10^{\mathbf{6}})(\mathbf{100})} = \mathbf{2.67}\times 10^{\mathbf{-4}}$$

$$\Delta\mathbf{G}3 = \mathbf{2.67}\times 10^{\mathbf{-4}}$$

@ Grid 4

$$\Delta\mathbf{G}4 = \Delta\mathbf{G}3 + \frac{\mathbf{P}}{\mathbf{K_{Spring}}} = 2.67\times 10^{-4} + \frac{100}{100} = 1.000267$$

$$\Delta\mathbf{G}4 = 1.000276$$

@ Grid 6

$$\Delta\mathbf{G6} = \Delta\mathbf{G4} + \frac{\mathbf{PL}}{\mathbf{AE}} = \mathbf{1.000267} + \frac{(\mathbf{100})(\mathbf{20})}{(\mathbf{10})(\mathbf{10}\times 10^{\mathbf{6}})} = \mathbf{1.000287}$$

$$\Delta\mathbf{G6} = \mathbf{1.000287}$$

18. Click on the Element box at the lower right hand corner and choose Node. By doing so, highlighting a node with the cursor will display results in detail.

    Element

    Node...

19. To see the total translation of the model.

View/Select...	**<F5>**
Deformed Style:	● **Deform**
Contour Style:	● **None-Model Only**

    Deformed and Contour Data...

Output Set:	**2..MSC/NASTRAN Case 1**
Deformation:	**3..T2 Translation**

    OK

    OK

    Place the cursor on Node 3, 4, and 6 and compare with the displacements in the Hand Calculation.

    The model should be similar to Figure 38.6.

    This concludes this exercise.

    File/Save

    File/Exit

Figure 38.6 - Horizontal Spring Load with Vertical Displacement.