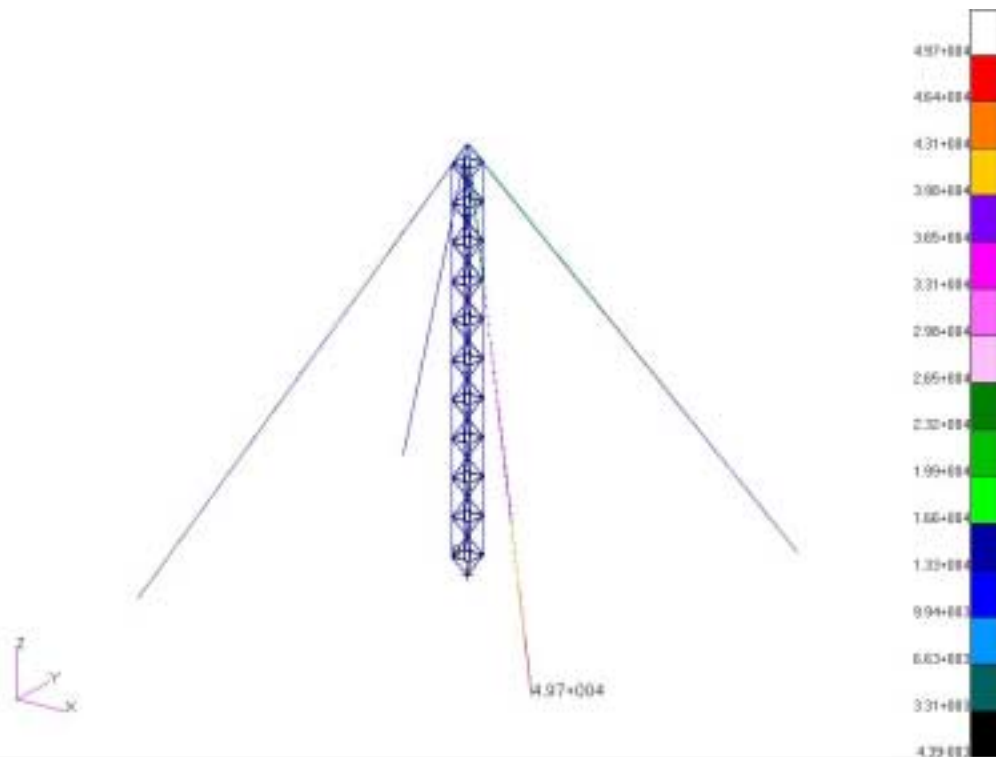


## LESSON 22

# *Frequency Response Analysis of Transmission Tower*



### Objectives:

- Creation of preload static step.
- Set-up and solution of frequency response analysis.
- Interpretation of results from frequency response analysis.



**Model Description:**

- This model represents a transmission tower such as those used to broadcast radio and television signals.
- The model will be pre-loaded to simulate the tensioning of the guy wires for the tower. After this pretensioning analysis, the model will be subjected to a frequency response analysis in order to determine its response to a load applied at the tip of the tower.

**Suggested Exercise Steps:**

- Read the Tower.ses file to create geometry.
- Mesh the model using 1 element per curve.
- Apply the model properties.
- Apply the model boundary conditions.
- Set-up and solve the complete frequency response analysis.
- Interpret results.

---

## Exercise Procedure:

1. Open a new database. Name it **Tower\_freq.db**.

**File/New ...**

*Database Name:*

**Tower\_freq.db**

**OK**

*Analysis Code:*

**MSC.Marc**

**OK**

2. Read the session file to create the Geometry.

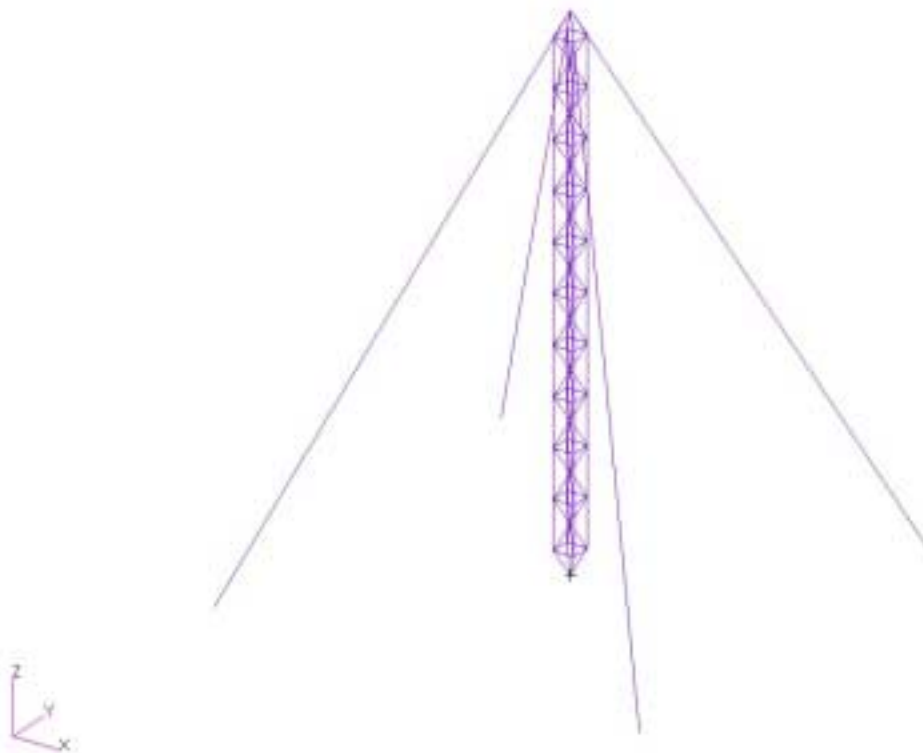
**File/Session/Play...**

*Session File List:*

**tower.ses**

**Apply**

**Figure 22.1 - Geometry created from session file.**



### 3. Create the Mesh Seeds

#### ■ Elements

Action:

Object:

Type:

#### ● Number of Elements

Number:

Curve List:

There are 3 curves which do not exist in the range. If you input the range by hand, choose *Yes for All* for the error message.

### 4. Create the Mesh for the tower.

#### ■ Elements

Action:

Object:

Type:

Topology:

Curve List:

Again, choose *Yes for All* to the error message.

### 5. Equivalence all for the tower.

#### ■ Elements

Action:

Object:

Method:

---

6. Create the Material property for the tower.

■ **Materials**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Isotropic"/>
<i>Method:</i>	<input type="text" value="Manual Input"/>
<i>Material Name:</i>	<input type="text" value="steel"/>
<input type="button" value="Input Properties..."/>	
<i>Elastic Modulus:</i>	<input type="text" value="30E6"/>
<i>Poission Ratio:</i>	<input type="text" value="0.3"/>
<i>Density:</i>	<input type="text" value="7.54E-4"/>
<input type="button" value="OK"/>	
<input type="button" value="Apply"/>	

7. Create the Element Properties for the beams.

First create the Properties for the group, **horizontal\_members**.

**Group/Post...**

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

■ **Properties**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="1D"/>
<i>Type:</i>	<input type="text" value="Elastic Beam"/>
<i>Property Set Name:</i>	<input type="text" value="horizontal_members"/>
<input type="button" value="Input Properties..."/>	
<i>Material Name:</i>	<input type="text" value="steel"/>
<i>XZ Plane Definition:</i>	<input type="text" value="&lt;0 0 1&gt;"/>
<i>Cross-Sectional Area:</i>	<input type="text" value="0.3436"/>

$I_{xx}$ :

3.35E-2

$I_{yy}$ :

3.35E-2

OK

These properties represent a 1" 0 D pipe with wall thickness of 0.125". The beam lies along the XY vector, the resultant of XYxXZ is the orientation for the beam strong axis.

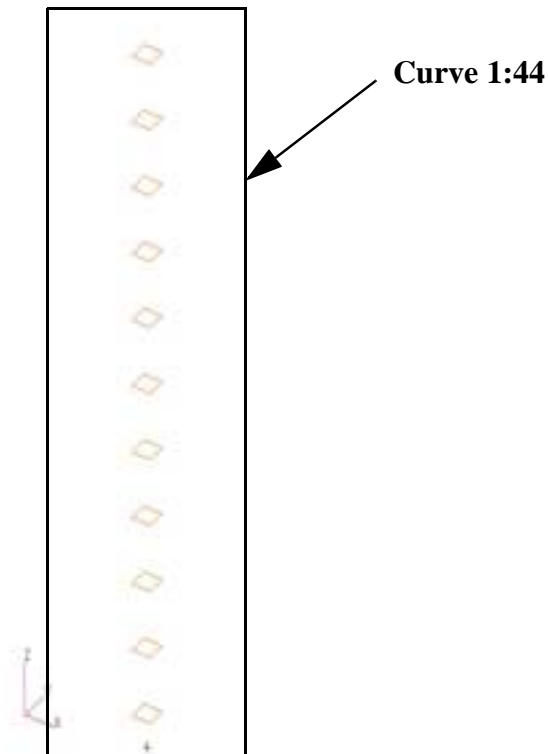
*Select Members:*

see Figure 22.2 and select all geometry in group,  
**Curve 1:44**

Add

Apply

**Figure 22.2 - All the curves to be selected for the Property set.**



---

Repeat the step for the group, **vertical\_members**, as well.

**Group/Post...**

*Groups to Post:*

**vertical\_members**

**Apply**

**Cancel**

■ **Properties**

*Action:*

**Create**

*Object:*

**1D**

*Type:*

**Elastic Beam**

*Property Set Name:*

**vertical\_members**

**Input Properties...**

*Material Name:*

**steel**

*XZ Plane Definition:*

**<1 0 0>**

*Cross-Sectional Area:*

**0.3436**

*Ixx:*

**3.35E-2**

*Iyy:*

**3.35E-2**

**OK**

*Select Members:*

select all geometry in group,  
**Curve 45:84**

**Add**

**Apply**

Now continue creating the Properties for the following groups, **cross\_orient1**, **cross\_orient2**, **cross\_orient3**, and **cross\_orient4**, changing only the Property Set Name and the XZ Plane Definition.

**Group/Post...**

*Groups to Post:*

**cross\_orient1**

**Apply**

**Cancel**

*Property Set Name:*

**cross\_orient1**

**Input Properties...**

*XZ Plane Definition:*

**<1 -1 0>**

**OK**

*Select Members:*

select all geometry in group,  
**Curve 85:105 175**

**Add**

**Apply**

**Group/Post...**

*Groups to Post:*

**cross\_orient2**

**Apply**

**Cancel**

*Property Set Name:*

**cross\_orient2**

**Input Properties...**

*XZ Plane Definition:*

**<1 1 0>**

**OK**

*Select Members:*

select all geometry in group,  
**Curve 107:128**

**Add**

**Apply**

**Group/Post...**

*Groups to Post:*

**cross\_orient3**

**Apply**

**Cancel**

*Property Set Name:*

**cross\_orient3**

---

**Input Properties...**

*XZ Plane Definition:*

**<1 1 0>**

**OK**

*Select Members:*

select all geometry in group,  
**Curve 130:151**

**Add**

**Apply**

**Group/Post...**

*Groups to Post:*

**cross\_orient4**

**Apply**

**Cancel**

*Property Set Name:*

**cross\_orient4**

**Input Properties...**

*XZ Plane Definition:*

**<1 -1 0>**

**OK**

*Select Members:*

select all geometry in group,  
**Curve 153:174**

**Add**

**Apply**

8. Create the Element Properties for the guy wires.

**Group/Post...**

*Groups to Post:*

**wire**

**Apply**

**Cancel**

*Action:*

**Create**

*Object:*

**1D**

*Type:*

**Truss**

<i>Property Set Name:</i>	<b>wire</b>
<b>Input Properties...</b>	
<i>Material Name:</i>	<b>steel</b>
<i>Cross-Sectional Area:</i>	<b>0.049</b>
<b>OK</b>	
<i>Select Members:</i>	Input or select, <b>Curve 176:179</b>
<b>Add</b>	
<b>Apply</b>	

9. Create a preload Load Case for analysis.

### ■ Load Cases

<i>Action:</i>	<b>Create</b>
<i>Load Case Name:</i>	<b>Preload</b>
<i>Load Case Type:</i>	<b>Time Dependent</b>
<b>Apply</b>	

10. Create a unit Field for non time varying loads.

### ■ Fields

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Non Spatial</b>
<i>Method:</i>	<b>Tabular Input</b>
<i>Field Name:</i>	<b>constant_on</b>
<i>Active Independent Variables:</i>	<b>■ Time (t)</b>
<b>Input Data...</b>	

	<b>Time(t)</b>	<b>Value</b>
1	<b>0</b>	<b>1</b>
2	<b>100</b>	<b>1</b>

**OK**

**Apply**

11. Create the fixed guy wire Boundary Condition.

■ **Loads/BCs**

*Action:* **Create**

*Object:* **Displacement**

*Type:* **Nodal**

*New Set Name:* **guy\_wire**

**Input Data...**

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

*Time/Freq. Dependence:*

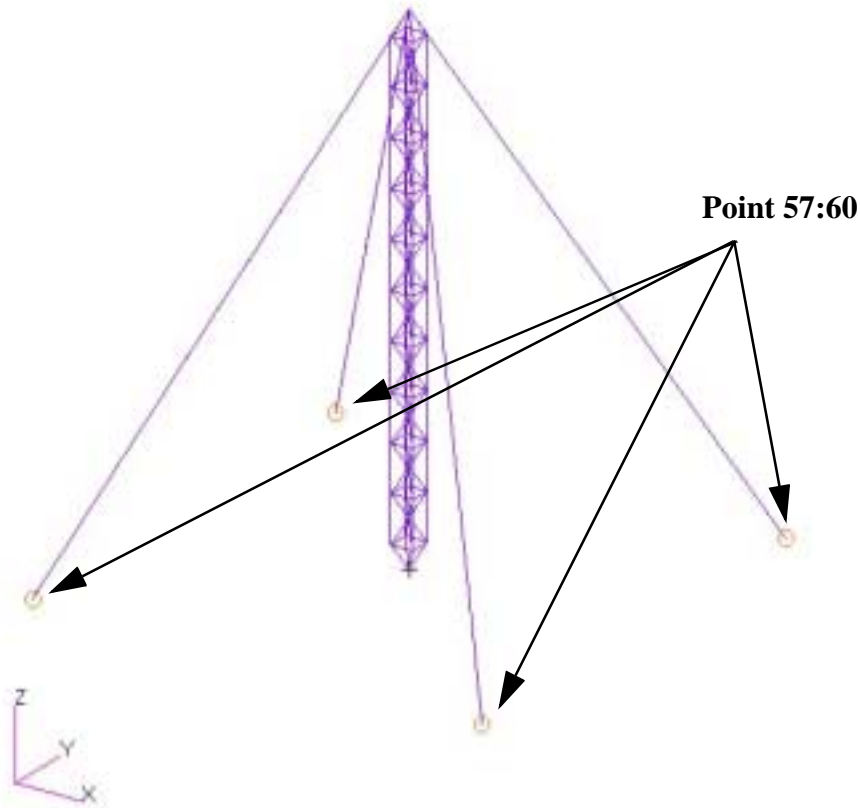
**OK**

**Select Application Region...**

*Select Geometry Entities:* **Point 57:60**

See **Figure 22.3** to pick points.

**Figure 22.3 - Choose the guy wire end points.**






12. Create a fixed Boundary Condition at the base of the tower.

Action:

Object:

Type:

New Set Name:

---

**Input Data...**

*Translations <T1 T2 T3>:*

**<0, 0, 1>**

*Time/Freq. Dependence:*

**constant\_on**

**OK**

**Select Application Region...**

*Select Geometry Entities:*

**Point 45**

**Add**

**OK**

**Apply**

13. Create a driving Force for the tower.

■ **Loads/BCs**

*Action:*

**Create**

*Object:*

**Force**

*Type:*

**Nodal**

*New Set Name:*

**nodal\_force**

**Input Data...**

*Translations <T1 T2 T3>:*

**<0, 1000, 0>**

*Time/Freq. Dependence:*

**constant\_on**

**OK**

**Select Application Region...**

*Select Geometry Entities:*

**Point 56**

**Add**

**OK**

**Apply**

14. Setup and run prestressed frequency response Analysis.

■ **Analysis**

*Action:*

**Analyze**

## *Freq. Resp. Analysis of Transmission Tower*

---

<i>Object:</i>	<b>Entire Model</b>
<i>Method:</i>	<b>Full Run</b>
<i>Job Name:</i>	<b>prestressed_freq</b>
<b>Translation Parameters...</b>	
<b>Solver Options...</b>	
	<input checked="" type="checkbox"/> <b>Non-Positive Definite</b>
<b>OK</b>	
<b>OK</b>	
<b>Load Step Creation...</b>	
<i>Job Step Name:</i>	<b>Prestress</b>
<i>Solution Type:</i>	<b>Static</b>
<b>Solution Parameters...</b>	
<b>Load Increment Parameters...</b>	
<i>Increment Type:</i>	<b>Fixed</b>
<i>Number of Increments:</i>	<b>10</b>
<i>Total Time:</i>	<b>1</b>
<b>OK</b>	
<b>OK</b>	
<b>Select Load Case...</b>	
<i>Available Load Cases:</i>	<b>preload</b>
<b>OK</b>	
<b>Apply</b>	
<i>Job Step Name:</i>	<b>Prestress_Modal</b>
<i>Solution Type:</i>	<b>Normal Modes</b>
<b>Solution Parameters...</b>	
<i>Extraction Method:</i>	<b>Lanczos</b>

---

*Number of Modes:*

*Lowest Frequency  
(cycles/time):*

*Highest Frequency:*

*Available Load Cases:*

*Job Step Name:*

*Solution Type:*

*Lowest Excitation Freq.  
(cycles/time):*

*Excitation Freq.  
Interval:*

*Number of Excitation  
Frequencies:*

*Available Load Cases:*

*Selected Job Steps:*

15. Read in the Results from the frequency Analysis.

### ■ Analysis

<i>Action:</i>	<b>Read Results</b>
<i>Object:</i>	<b>Result Entities</b>
<i>Method:</i>	<b>Attach</b>
<b>Select Results File...</b>	
<i>Selected Results File:</i>	<b>prestressed_freq.t16</b>
<b>OK</b>	
<b>Apply</b>	

16. Post-process the Results from the pre-stress analysis.

### ■ Results

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Graph</b>
<i>Method:</i>	<b>Y vs X</b>
<i>Select Result Case(s):</i>	<b>Prestress_Freq</b>
<b>Filter</b>	
<b>Apply</b>	
<b>Close</b>	
<i>Select Y Result:</i>	<b>Displacement, Translation</b>
<i>X:</i>	<b>Global Variable</b>
<i>Variable:</i>	<b>Frequency</b>

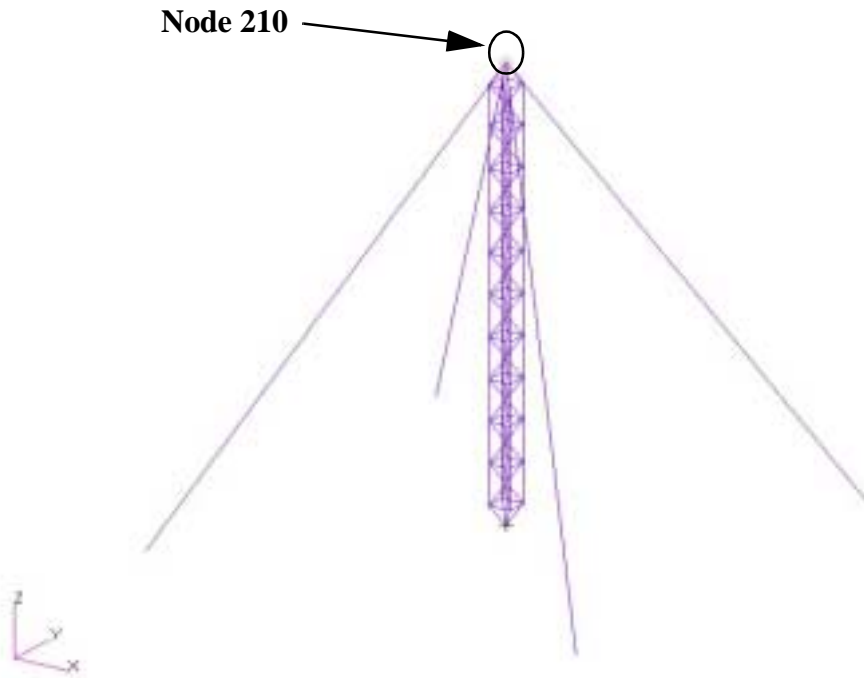
Select the **Target Entity** icon.



**Target Entity**

---

**Figure 22.4 - Choose this node for the target node on the graph.**



*Target Entity:*

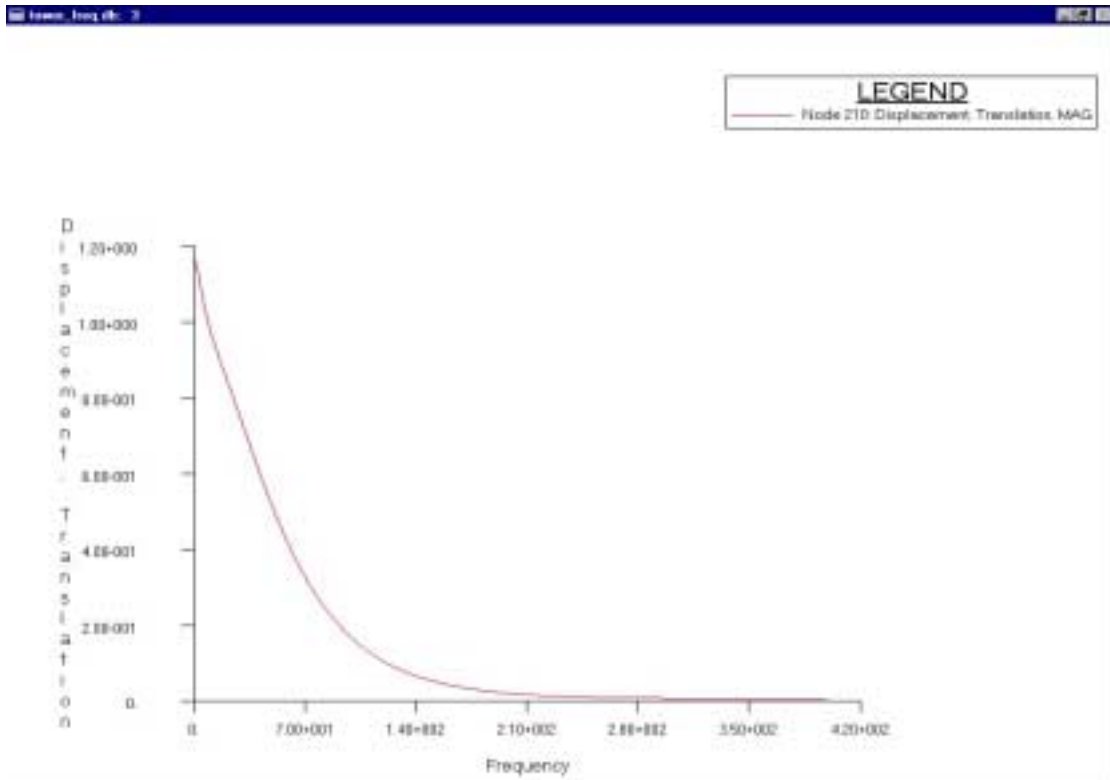
**Nodes**

*Select Nodes:*

**Node 210**

**Apply**

**Figure 22.5 - Here appears the Displacement vs. Frequency graph for Node 210.**



17. Create a Fringe plot for a selected frequency.

### ■ Results

*Action:*

Create

*Object:*

Quick Plot

*Select Result Cases:*

Prestress, A1:... Incr=5

*Select Fringe Result:*

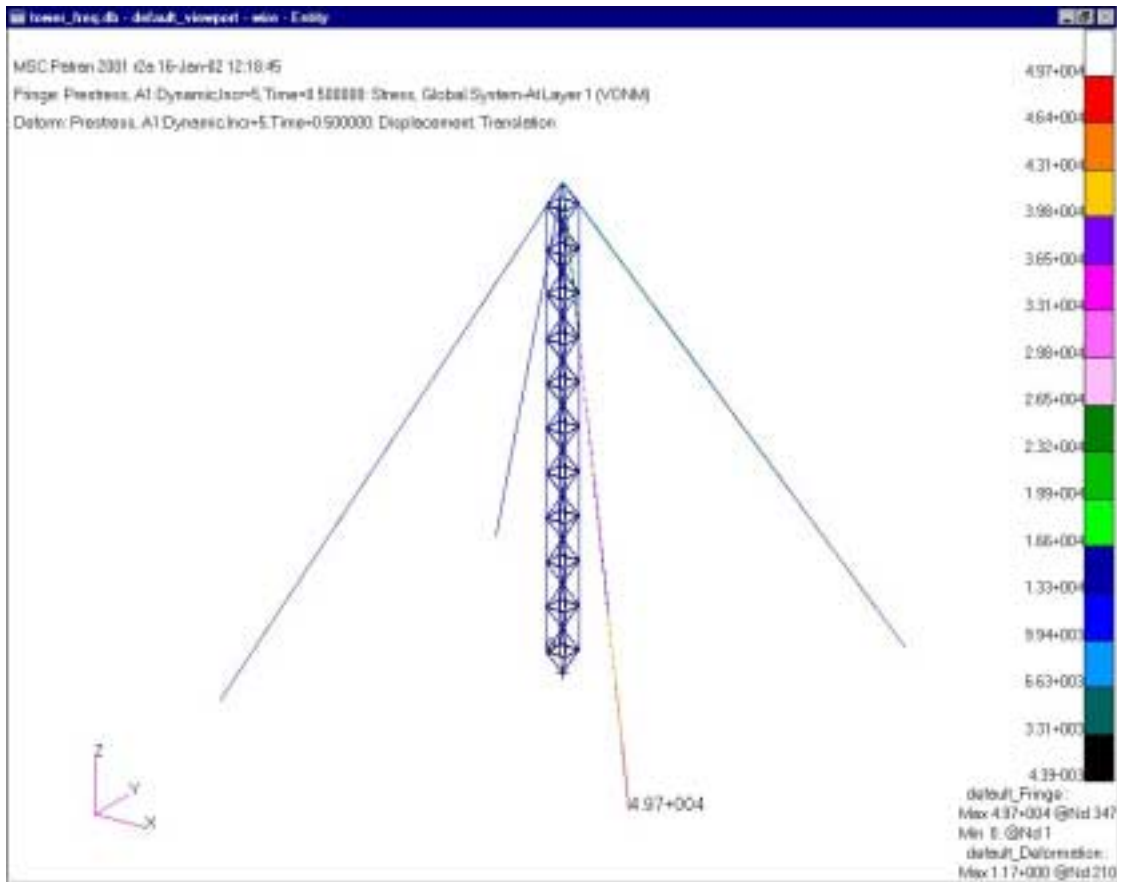
Stress, Global System

*Select Deformation Result:*

Displacement, Translation

Apply

**Figure 22.6 - This shows the fringe plot for one frequency.**



Close the database and quit PATRAN.

This concludes this exercise.