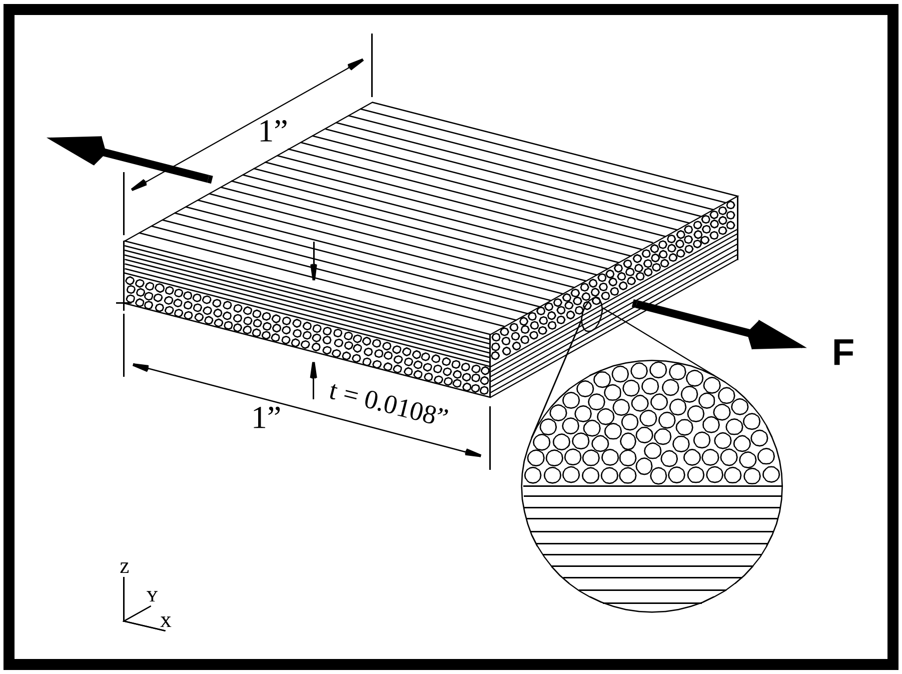
WORKSHOP PROBLEM 1a

Uniaxial Loading of a Laminar Composite Plate (Part I)



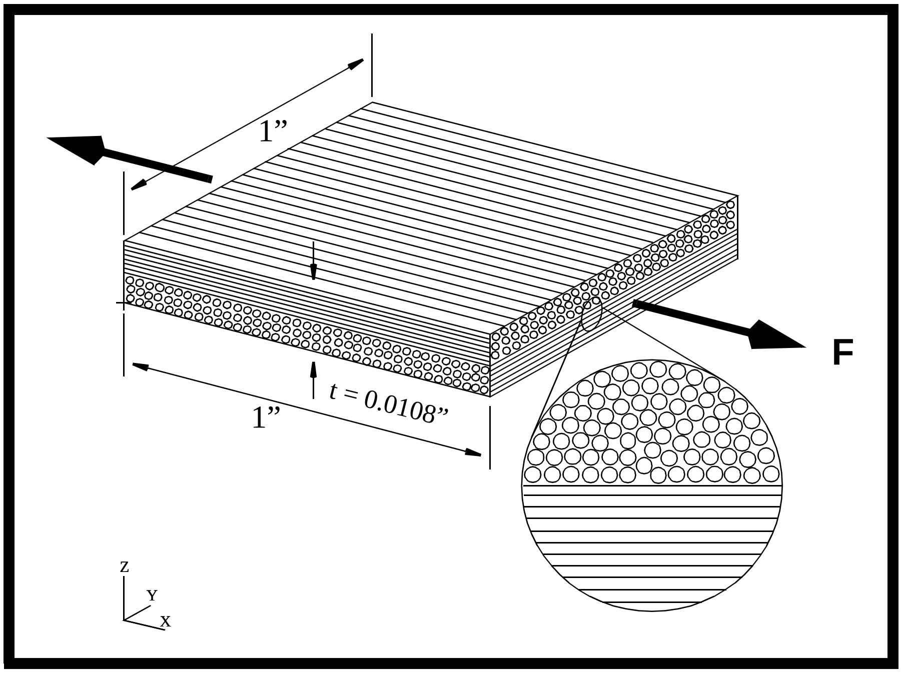
Objectives:

- Create composite material definition.
- Create model.
- Specify loads.
- Create a MSC/NASTRAN input file directly or by using MSC/PATRAN.
- Run the analysis using MSC/NASTRAN.
- Review deformed shape.

Model Description:

The figure below shows a 2-ply composite plate with uniaxial loading.

Figure 1a-1



The plies are a typical graphite/epoxy tape with the following properties:

Table 1a-1: Material Properties

Elastic Modulus, 1-1	20×10^6 psi
Elastic Modulus, 1-2	2×10^6 psi
Poisson Ratio	0.35
Shear Modulus	1×10^6 psi
Layer Thickness (in)	.0054 in

Suggested Exercise Steps:

- Create the elements.
- Define the ply material (MAT8) and composite layer element properties (PCOMP) and apply them to the model.
- Apply uniform loads and constraints.
- Prepare the model for a linear static analysis (SOL 101).
- Submit it for a linear static analysis.
- Review results.

Exercise Procedure:

1. Users who are not utilitizing MSC/PATRAN for generating an input file should go to Step 12, otherwise, proceed to step 2.

2. Create a new database called **prob1a.db**.

File/New...

New Database Name:	**prob1a**

OK

In the New Model Preferences form, set the following:

Tolerance:	◆ **Default**
Analysis Code:	**MSC/NASTRAN**
Analysis Type:	**Structural**

OK

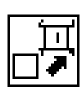
3. Create geometry for the finite element model.

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Vector Coordinate List:	**< 1, 1, 0 >**
Origin Coordinate List:	**[0, 0, 0]**

Apply

For the next step, turn on entity labels using the icon:

 **Show Labels**

4. Create distributed load to the surface at Surface (Edge) 1.3.

◆ **Loads/BCs**

Action:	**Create**

Object: **Distributed Load**

Type: **Element Uniform**

New Set Name: uniform_load

Target Element Type: **2D**

Input Data...

Edge Distr Load <f1 f2 f3>: **<0 -15 0>**

OK

Select Application Region...

Geometry Filter: ● **Geometry**

Select Surface Edges: **Surface 1.3**

Add

OK

Apply

5. Create boundary conditions for Surface (Edge) 1.1 and Point 1.

   Constraining Surface 1.1 translations in x and z directions and rotation in y-direction:

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: simple_constraint

Input Data...

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

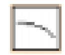
Rotations <R1 R2 R3>: **< ,0, >**

OK

Select Application Region...

Select Geometric Entities: **Surface 1.1**

(Remember...“**Curve or Edge**” icon to select the edge.)



Curve or Edge

Add
OK

Apply

Constraining Point 1 in y-direction:

◆ **Loads/BCs**

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

Input Data...

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

OK

Select Application Region...

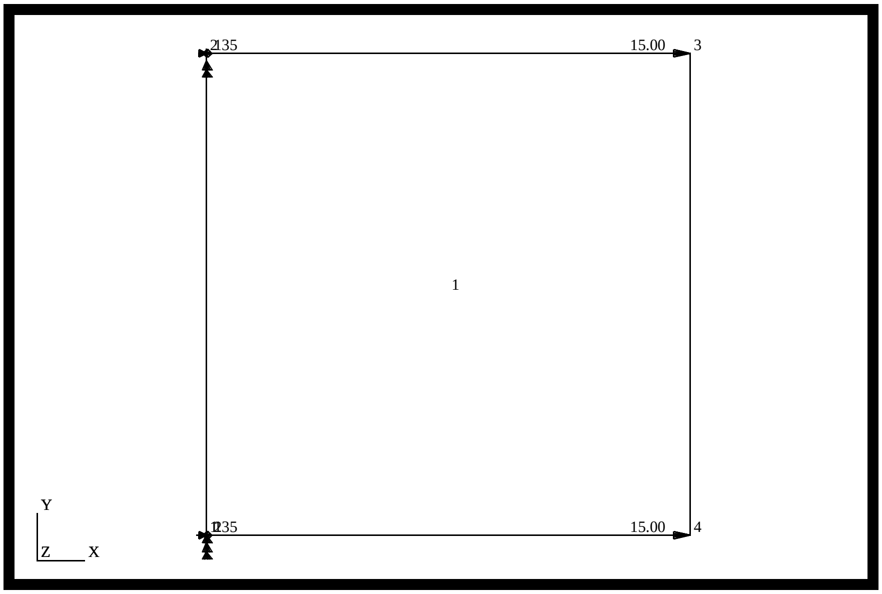
Select Geometry Entities:	**Point 1**

Add
OK

Apply

The model should appear as follows:

Figure 1a-2: Geometry with loads and boundary conditions.



6. Define nodes and element connectivities by generating a mesh over the geometry.

◆ **Finite Elements**

Action: **Create**

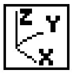
Object: **Mesh**

Type: **Surface**

Global Edge Length **0.5**

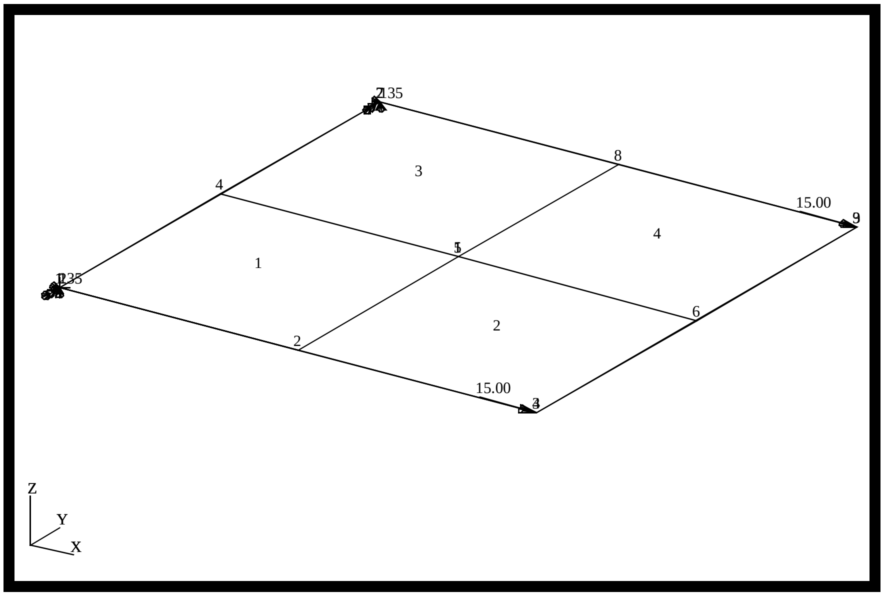
Surface List **Surface 1**

Apply

View geometry:  **Iso 3 View**

The completed model should appear as follows:

Figure 1a-3: The finite element model.



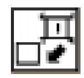
7. Next, define the ply material describing the constituent layers of the composite using the specified modulus of elasticity, shear modulus, and poisson ratio.

◆ **Materials**

Action:	**Create**
Object:	**2d Orthotropic**
Method:	**Manual Input**
Material Name:	**ply_mat**
Input Properties ...	
Constitutive Model:	**Linear Elastic**
Elastic Modulus 11 =	**20e6**
Elastic Modulus 22 =	**2e6**
Poisson Ratio 12 =	**.35**
Shear Modulus 12 =	**1e6**
Apply	
Cancel	

7a. Ply direction or angle is the direction of the 1 axis of the ply coordinate system. Since E_{11} equals 10 times E_{22}, it is inferred that the fibers are going in the direction of the 1 axis of the ply coordinate system.

7b. NASTRAN defines Ply 1 as being the most negative ply in the z direction of the element coordinate system (Z_e). The element z axis can be displayed using the normal display as shown below:

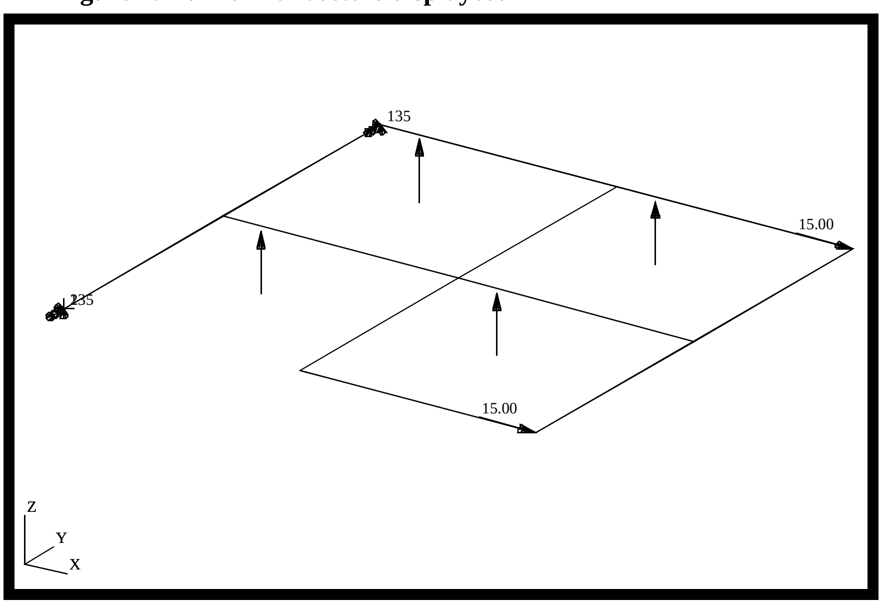
Hide Labels:  **Hide Labels**

◆ **Finite Elements**

Action:	**Verify**
Object:	**Element**
Test:	**Normals**
Display Control:	● **Draw Normal Vectors**
Test Control:	**Display Only**

Apply

Figure 1a-4: Normal vectors displayed.



Looking back, Figure 1a-1 shows that Ply 1 has fibers lying in the direction of the global y axis (Y_g) and layer 2 has fibers in the direction of the global x axis (X_g).

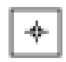
7c. NASTRAN defines the zero ply orientation angle as being in the direction of the x axis of the material coordinate system (X_m).

X_m will be defined by the projection of the x axis of a user defined coordinate system (X_u) onto the surface of the elements.

We will define a material coordinate system with an x axis in the direction of Y_g as shown below:

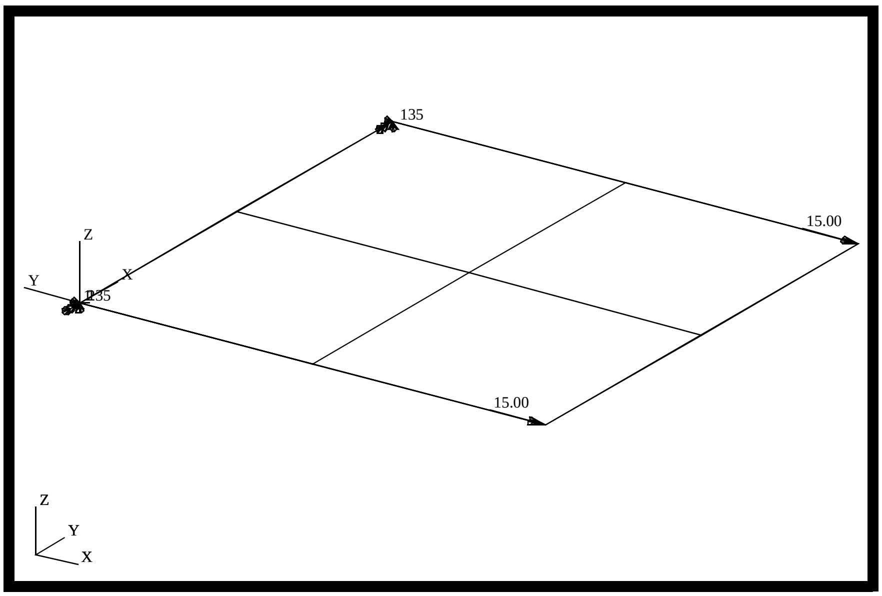
◆ **Geometry**

Action:	**Create**
Object:	**Coord**
Method:	**3Point**
Point on Plane 1-3:	**Node 4**

(Remember...“**Node**” icon to select node.)  **Node**

Apply

Figure 1a-5: Material coordinate system created.



By projecting X_u onto the elements, a ply with fibers going in the direction of Y_g has a ply orientation angle of 0 degrees, such as Ply 1 (most negative Z_e), and plies in the direction of X_g will have a ply orientation angle of 90 degrees.

8. Using the material **ply_mat**, create a composite.

◆ **Materials**

Action:	**Create**

Object: **Composite**

Method: **Laminate**

Material Name: **composite1**

To enter material layers for the composite: Click on **ply_mat** in the "*Existing Materials*" databox twice.

To enter layer thickness: (1) Click "*Thickness For All Layers of...*" databox in the *Laminated Composite* form, (2) type **.0054**, and (3) hit **Enter**.

To enter layer orientations: (1) Under *Orientation*, click cell on the first row. (2) Click *Overwrite Orientations* databox, (3) type **0**, **Enter**, and **90, Enter**. Then click on the following:

Load Text Into Spreadsheet

The form should look like the following:

	Material Name	*Thickness*	*Orientation*
1	**ply_mat**	**.0054**	**0**
2	**ply_mat**	**.0054**	**90**

Apply

9. Next, create an element property for referencing the composite created in the previous step.

◆ **Properties**

Action: **Create**

Dimension: **2 D**

Type: **Shell**

Property Set Name: **plate**

Option(s): *(pull-down menu)* **Laminate**

Input Properties ...

Material Name **m:composite1**

Material Orientation: **Coord 1**

OK

Select Members: **Surface 1**

Add

Apply

9a. In order to check the X_m direction (orientation angle), do the following:

◆ **Properties**

Action: **Show**

Select Property: **Orientation Angle**

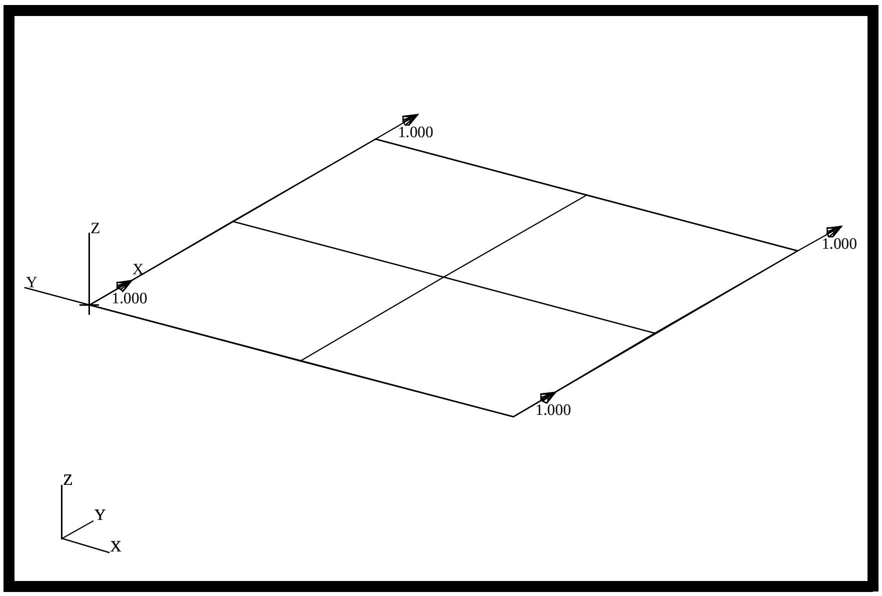
Display Method: **Vector Plot**

Select Groups: ● **Current Viewport**

default_group

Apply

Figure 1a-6: Material orientation angle shown.



10. Submit model for linear static 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:	**prob1a**

Apply

An MSC/NASTRAN input file called **prob1a.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.

Generating an input file for MSC/NASTRAN Users:

11. The generated input file (prob1a.bdf) should be similar to the following:

(Type "**more prob1a.bdf**" at UNIX shell window to compare.)

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator (  MSC/PATRAN Version 7.5 ) on January   15, 1998 at
$ 19:10:09.
ASSIGN OUTPUT2 = 'prob1a.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 15-Jan-98 at 19:09:52
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
   SUBTITLE=Default
   SPC = 2
   LOAD = 2
   DISPLACEMENT(SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
   STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
BEGIN BULK
PARAM    POST    -1
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
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : plate
$ Composite Property Record created from P3/PATRAN composite material
$ record : composite1
$ Composite Material Description :
PCOMP    1                                       0.      0.
+      A
+      A 1       .0054   0.      YES     1       .0054   90.     YES
CQUAD4   1       1       1       2       5       4       1
CQUAD4   2       1       2       3       6       5       1
CQUAD4   3       1       4       5       8       7       1
CQUAD4   4       1       5       6       9       8       1
$ Referenced Material Records
```

```
$ Material Record : ply_mat
$ Description of Material : Date: 15-Jan-98           Time: 17:40:54
MAT8     1       2.+7    2.+6    .35     1.+6
$ Nodes of the Entire Model
GRID     1               0.      0.      0.
GRID     2               .5      0.      0.
GRID     3               1.      0.      0.
GRID     4               0.      .5      0.
GRID     5               .5      .5      0.
GRID     6               1.      .5      0.
GRID     7               0.      1.      0.
GRID     8               .5      1.      0.
GRID     9               1.      1.      0.
$ Loads for Load Case : Default
SPCADD   2       1       3
LOAD     2       1.      1.      1
$ Displacement Constraints of Load Set : simple_constraint
SPC1     1       135     1       4       7
$ Displacement Constraints of Load Set : y_constraint
SPC1     3       2       1
$ Distributed Loads of Load Set : uniform_load
FORCE    1       3               3.75    1.      0.      0.
FORCE    1       6               3.75    1.      0.      0.
FORCE    1       6               3.75    1.      0.      0.
FORCE    1       9               3.75    1.      0.      0.
$ Referenced Coordinate Frames
CORD2R   1               0.      0.      0.      0.      0.      1.
+      B
+      B 0.      1.      0.
ENDDATA f64cb716
```

SUBMITTING THE INPUT FILE FOR MSC/NASTRAN and MSC/PATRAN USERS:

12. Submit the input file to MSC/NASTRAN for analysis by finding an available UNIX shell window. At the command prompt enter **nastran prob1a.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

13. Proceed with the Reverse Translation process, that is, importing the **prob1a.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

Action: **Read Output 2**

Object: **Result Entities**

Method: **Translate**

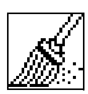
Select Results File...

Selected Results File **prob1a.op2**

OK

Apply

Before postprocessing the results, clear the LBC markers from the screen by selecting the following main menu icon:



Reset Graphics

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

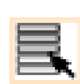
14. Create Fringe plot.

◆ **Results**

Action: **Create**

Object: **Fringe**

Click Select Results icon:



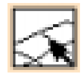
Select Results

Select Result Cases: **Default, Static Subcase**

Select Fringe Result: **Displacements,Translational**

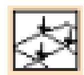
Position...((NON-LAYERED))

Quantity: **Z Component**

Click Target Entities icon:  **Target Entities**

Target Entity: **Current Viewport**

(Note: **Target Entity** allows you to view fringe plots of entities of your choice.)

Click Display Attributes icon:  **Display Attributes**

Style: **Discrete/Smooth**

Display: **Free Edges**

(Note: **Display Attributes** form allows you to change the displayed graphics of fringe plots.)

Now click Plot Options icon.  **Plot Options**

Coordinate Transfomation: **None**

Filter Values: **None**

Apply

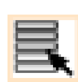
You can press **Apply** whenever you choose; after working on *each* icon menu or, as we have done here, after working on *all* of them.

15. Create Deformation plot.

◆ **Results**

Action: **Create**

Object: **Deformation**

Click Select Results icon:  **Select Results**

Select Result Cases: **Default, Static Subcase**

Select Deformation Result: **Displacements,Translational**

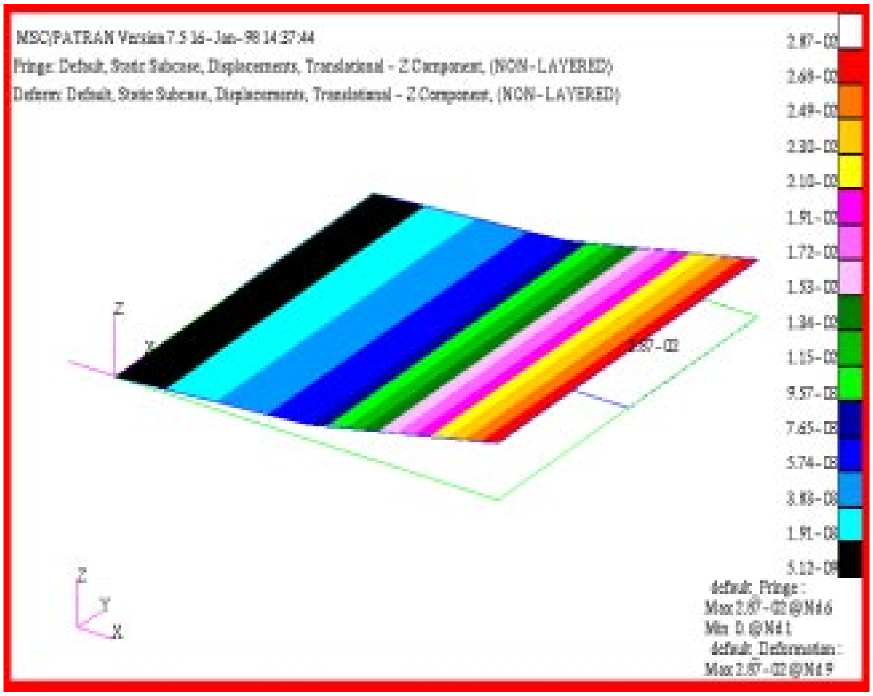
Position...((NON-LAYERED))

Show As: **Resultant**

Go through all the icon menus as we have done with the Fringe plot (if you wish), and then:

Apply

Figure 1a-7: Fringe and Deformation plots of our model.



Notice that the z displacement of the model dominates the deformation. This is due to the effects of the ply orientation.

The first layer's fibers (the bottom layer) are oriented at 90 degrees to the load, while the second layer's fibers (top) are oriented at 0 degrees. The ply modulus in the primary direction (0 degrees orientation) is ten times that of the secondary direction. Therefore, under tension, the bottom layer will translate more than the top layer and "bow" the material upwards.

Quit MSC/PATRAN when you have completed this exercise.