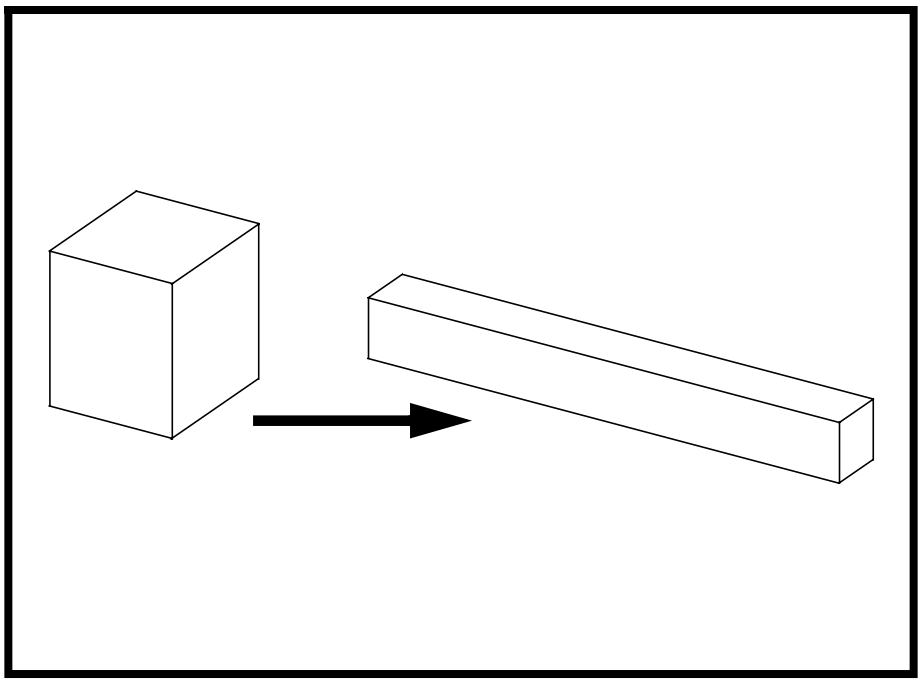
WORKSHOP PROBLEM 5

Large-Scale Deformation of a Hyperelastic Material



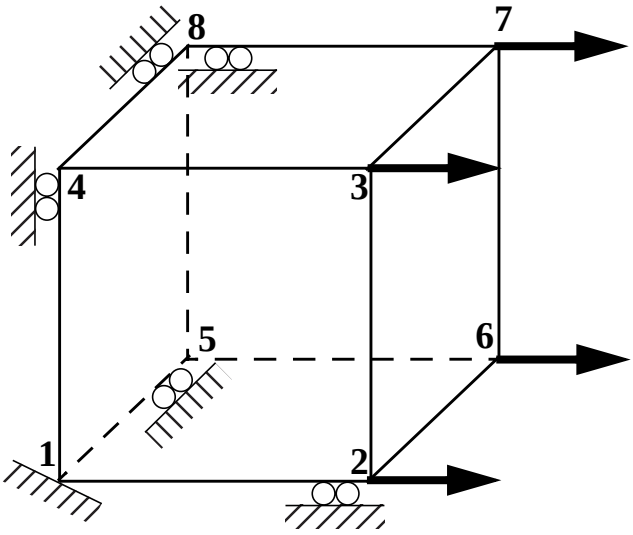
Objectives:

- Create a model with hyperelastic material properties.
- Submit an MSC/NASTRAN nonlinear analysis.
- Generate an accurate deformation plot of the model.

Model Description:

Below is a finite element representation of a hyperelastic material under load. A nonlinear analysis with load increments will be performed to obtain the deformation of this material under load.

Figure 5.1



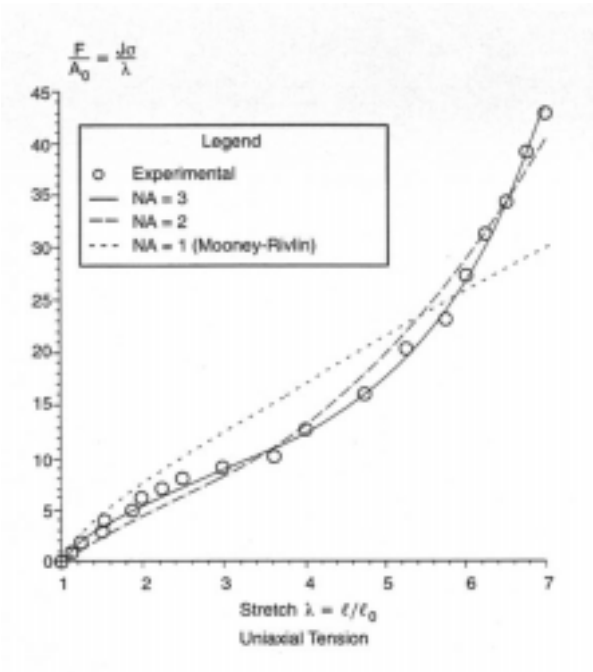


Table 5.1 - Hyperelastic Properties

Distortional Deformation Constants, A_{10}:	**0**
Distortional Deformation Constants, A_{01}:	**1500**
Distortional Strain Energy, N_A:	**3**
Volumetric Strain Energy, N_D:	**1**

Exercise Procedure:

1. Start up MSC/NASTRAN for Windows V3.0 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.

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

Open Model File: **New Model**

(Optional) For users who wish to remove the default rulers in the work plane model, please do the following:

View/Options...

● **Tools and View Style**

Category: **Workplane and Rulers**

Draw Entity ☐

Apply

Cancel

2. Create functions to define the hyperelastic material properties.

From the pulldown menu, select **Model/Function**.

Model/Function...

Title: **Simple_Tension**

Type: **4..vs Stress**

Load the data from the functions library.

Load...

Library Entry: **Treloar[1944]SimpleTen**

OK

Remove the first two entries of the function and replace with the new data point.

Delete

Delete

Data Entry: ● **Single Value**

X:	*Y:*	
1	**0**	**More**
1.125	**1**	**OK**

Create the second function.

ID: **2**

Title: **Equibiaxial_Tension**

Type: **4..vs Stress**

Load the data from the functions library.

Load...

Library Entry: **Treloar[1944]Equibiaxial**

OK

OK

Create the third function.

ID: **3**

Title: **Pure_Shear**

Type: **4..vs Stress**

Load the data from the functions library.

Load...

Library Entry: **Treloar[1944]PureShear**

OK

OK

Cancel

3. Create a material called **mat_1**.

From the pulldown menu, select **Model/Material**.

Model/Material...

Title: **mat_1**

Type...

● **Hyperelastic**

OK

Distortional Deformation Constant (A_{ij}); i=row, j=column:

A_{10}: **0**

A_{01}: **1500**

Strain Energy Polynomial Order:

Distortional: **3**

Volumetric: **1**

To select the function from the functions created in the previous step, select the desired field and depress **Ctrl-F**.

Experimental Data Functions:

Simple Ten/Comp: **Ctrl-F**

Entity ID: **1..Treloar [1944] Simple Ten**

OK

Equibiaxial Tension: **Ctrl-F**

Entity ID: **2..Treloar[1944] Equibiaxial**

OK

Pure Shear: **Ctrl-F**

Entity ID: **3..Treloar [1944] Pure Shear**

OK

OK

Cancel

4. Create the property that will define the hyperelastic element.

Model/Property...

Elem/Property Type...

Volume Elements: ● **Solid**

OK

Title: **Solid**

To select the material, click on the list icon next to the databox and select **mat_1**.

Material: **1..mat_1**

OK

Cancel

5. Create the nodes for the structure.

Model/Node...

	X:	*Y:*	*Z:*
Coordinates:	**0**	**0**	**0**

Parameters...

Permanent Constraints: ☒ **TX** ☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

OK

Repeat the process for the other 7 nodes.

Node 2.

X:	*Y:*	*Z:*		
1	**0**	**0**	**Parameters...**	
☐ **TX**	☒ **TY**	☒ **TZ**		
☒ **RX**	☒ **RY**	☒ **RZ**	**OK**	**OK**

Node 3.

X:	*Y:*	*Z:*		
1	**1**	**0**	**Parameters...**	
☐ **TX**	☐ **TY**	☒ **TZ**		
☒ **RX**	☒ **RY**	☒ **RZ**	**OK**	**OK**

Node 4.

X:	*Y:*	*Z:*		
0	**1**	**0**	**Parameters...**	
☒ **TX**	☐ **TY**	☒ **TZ**		
☒ **RX**	☒ **RY**	☒ **RZ**	**OK**	**OK**

Node 5.

X:	*Y:*	*Z:*		
0	**0**	**-1**	**Parameters...**	
☒ **TX**	☒ **TY**	☐ **TZ**		
☒ **RX**	☒ **RY**	☒ **RZ**	**OK**	**OK**

Node 6.

X:	*Y:*	*Z:*		
1	**0**	**-1**	**Parameters...**	
☐ **TX**	☒ **TY**	☐ **TZ**		
☒ **RX**	☒ **RY**	☒ **RZ**	**OK**	**OK**

Node 7.

X:	*Y:*	*Z:*		
1	**1**	**-1**	**Parameters...**	
☐ **TX**	☐ **TY**	☐ **TZ**		
☒ **RX**	☒ **RY**	☒ **RZ**	**OK**	**OK**

Node 8.

X:	*Y:*	*Z:*		
0	**1**	**-1**	**Parameters...**	
☒ **TX**	☐ **TY**	☐ **TZ**		
☒ **RX**	☒ **RY**	☒ **RZ**	**OK**	**OK**

Cancel

To bring the model into the viewable area, use the Autoscale feature and the Rotate feature.

View/Autoscale...

View/Rotate...

Trimetric

OK

6. Create the element of the structure.

Model/Element...

Type...

Volume Elements: ● **Solid**

OK

Property: **1..Solid**

Nodes: **1** **2** **3** **4** **5** **6** **7** **8**

● **Brick**

OK

NOTE: When selecting the nodes, you may (if you wish) manually type in the endpoint nodes of the solid elements. However, it is easier to use the graphic interface and select the nodes on the screen using the mouse. Click in the first *Nodes* box and then select the nodes on the screen in the following order. (Note that the node nearest to the cursor is highlighted by a large yellow X - you don't have to click precisely on the node!)

Click **OK** when MSC/NASTRAN returns the message "Element or Region Top Face Below Bottom Face. Switching."

OK

Cancel

7. Create the model constraints.

Before creating the appropriate constraints, a constraint set must be created. Do so by performing the following:

Model/Constraint/Set...

Title: **constraint_1**

OK

Now define the relevant constraint for the model.

Model/Constraint/Nodal...

Select **Node 7**.

OK

In the *DOF* box, check the following boxes:

☒ **TX** ☐ **TY** ☐ **TZ**
☐ **RX** ☐ **RY** ☐ **RZ**

OK

Cancel

Now create the second load set that defines the connectivity between **Node 7** and the remaining nodes.

Model/Constraint/Set...

Constraint Set ID: **2**

Title: **constraint_2**

OK

Now define the relevant constraint for the model.

Model/Constraint/Equation...

Enter the following under the *Constraint Equation* form.

Coeff **1**

Node ID **2**

DOF ☒ **TX**

Add

Coeff **-1**

Node ID **7**

DOF ☒ **TX**

Add

OK

Create the remaining 8 constraint equation.

ID **2**
Coeff **1**
Node ID **3**
DOF ☒ **TX**
Add
Coeff **-1**
Node ID **7**
DOF ☒ **TX**
Add
OK

ID **3**
Coeff **1**
Node ID **6**
DOF ☒ **TX**
Add
Coeff **-1**
Node ID **7**
DOF ☒ **TX**
Add
OK

ID **4**
Coeff **1**
Node ID **3**

DOF ☒ **TY**

Add

Coeff **-1**

Node ID **7**

DOF ☒ **TY**

Add

OK

ID **5**

Coeff **1**

Node ID **4**

DOF ☒ **TY**

Add

Coeff **-1**

Node ID **7**

DOF ☒ **TY**

Add

OK

ID **6**

Coeff **1**

Node ID **8**

DOF ☒ **TY**

Add

Coeff **-1**

Node ID **7**

DOF ☒ **TY**

Add

OK

ID **7**
Coeff **1**
Node ID **5**
DOF ☒ **TZ**
Add
Coeff **-1**
Node ID **7**
DOF ☒ **TZ**
Add
OK

ID **8**
Coeff **1**
Node ID **6**
DOF ☒ **TZ**
Add
Coeff **-1**
Node ID **7**
DOF ☒ **TZ**
Add
OK

ID **9**
Coeff **1**
Node ID **8**
DOF ☒ **TZ**
Add
Coeff **-1**

Node ID **7**

DOF ☒ **TZ**

Add

OK

Cancel

8. Create the model loading.

Like the constraints, a load set must first be generated before creating the appropriate model loading.

Model/Load/Set...

Title: **load_1**

OK

Since this is a nonlinear analysis, the nonlinear analysis load set options must first be defined.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Defaults...

Number of Increments: **48**

Stiffness Updates / Method: **1..AUTO**

Iterations Before Update: **1**

Output Control / Intermediate: **1..YES**

OK

Next, create the displacement load.

Model/Load/Nodal...

Select **Node 7**.

OK

Highlight **Displacement**.

TX ☒ **6**

Method: ● **Constant**

OK

Cancel

9. Submit the job for analysis.

File/Export/Analysis Model...

Analysis Type: **10..Nonlinear Static**

OK

Change the directory to **C:\temp**.

File name: **prob5**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Large Deformation of Hyperelastic Material**

OK

Under *Output Requests*, change the output to:

1..PostProcess Only

Also deselect all the boxes except the following:

☒ **Displacement**

Under *Analysis Case Requests*, enter the following:

☒ **Loads =** **1..load_1**

☒ **Constraints (SPC) =** **1..constraint_1**

☒ **Constraint Eqns (MPC) =** **2..constraint_2**

OK

OK

When asked if you wish to save the model, respond **Yes**.

Yes

File name: **prob5**

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 the analysis ran smoothly, we will not bother with the details this time.

Continue

When asked if it is "OK to Begin Reading File C:\TEMP\prob5.xdb," respond **Yes**.

Yes

10. List the results of the analysis.

To list the results, select the following

List/Output/Query...

Output Set: **48..Case 48 Step 1.000000**

Category: **1..Displacement**

Entity: ● **Node**

ID: **2**

OK

NOTE: You may want to expand the message box in order to view the results. To do this, double click on the message box. Adjust the size of the box to your preference by dragging the top border downward.

Answer the following questions using similar procedure. The answers are listed at the end of the exercise.

What is the T1 displacement of **Node 2** at load step = 1.0?

T1 Translation @ Node 2 = _____

What is the T1 displacement of **Node 5** at load step = 1.0?

T1 Translation @ Node 5 = _____

11. Display the deformed plot on the screen.

First, you may want to remove the labels and LBC markers in order to give a better view of the deformation.

View/Options...

Quick Options...

Labels Off

☐ **Node - Perm Constrain**

☐ **Load - Displacement**

☐ **Constraint**

☐ **Constraint Equation**

Done

OK

Plot the deformation of the structure.

View/Select...

Deformed Style: ● **Deform**

Deformed and Contour Data...

Data Selection/Category: **1..Displacement**

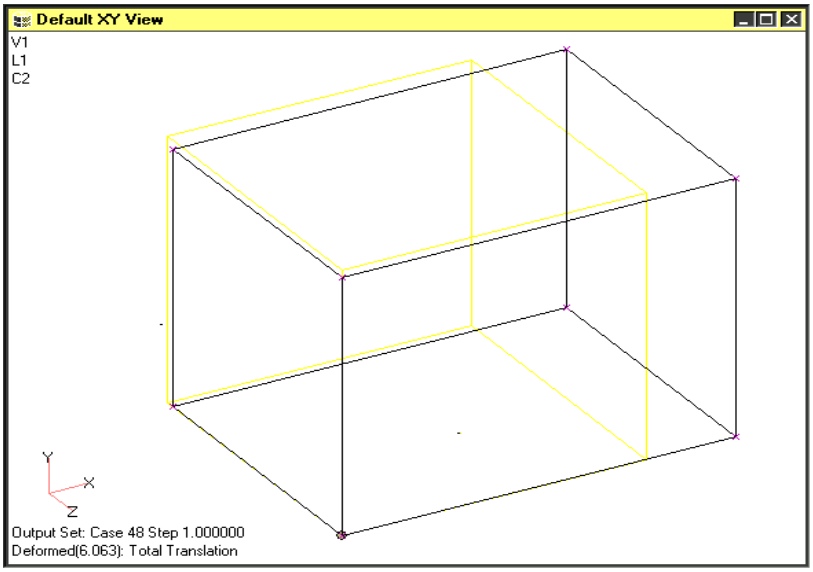
Output Set: **48..Case 48 Step 1.000000**

Output Vectors/Deformation: **1..Total Translation**

OK

OK

The XY view should appear as follows:



In order to see the deformation results accurately, you will need to turn off the display scaling of the actual deformation.

View/Options...

Category: ● **PostProcessing**

Options: **Deformed Style**

% of Model (Actual) ☐

OK

NOTE: You may need to decrease the magnification of the model in order to see deformation of the model.

View/Magnify...

Notice how drastically the shape of the hyperelastic element changed. The height and the width of the element shrinked to less than half of their original dimensions in order to compensate for the deformation in the x direction.

This concludes the exercise.

Disp X @ Node 2:	**6.00000**
Disp X @ Node 5:	**0.00000**