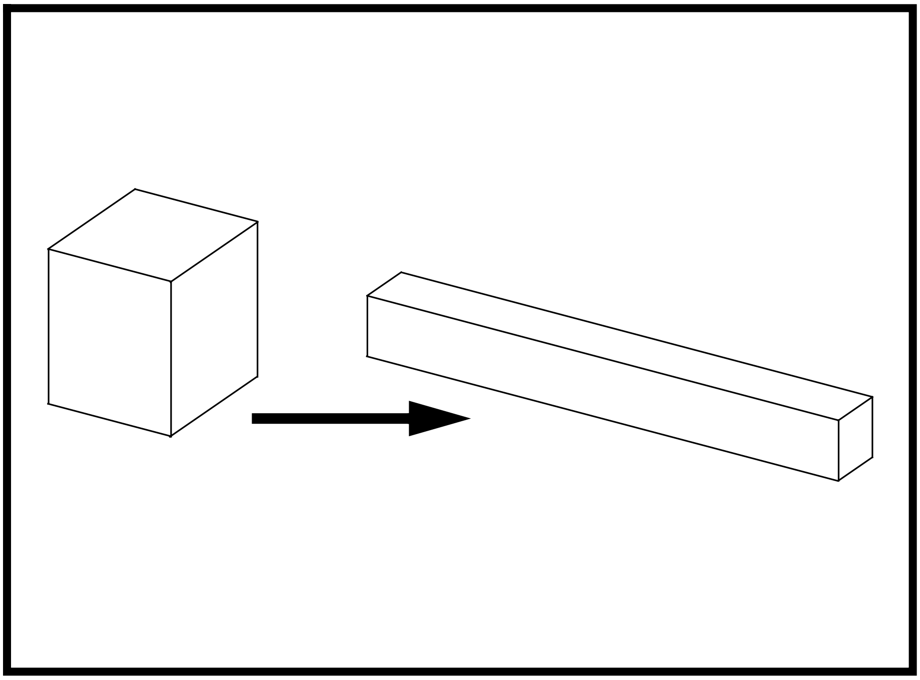
WORKSHOP PROBLEM 5

Large-Scale Deformation of a Hyperelastic Material

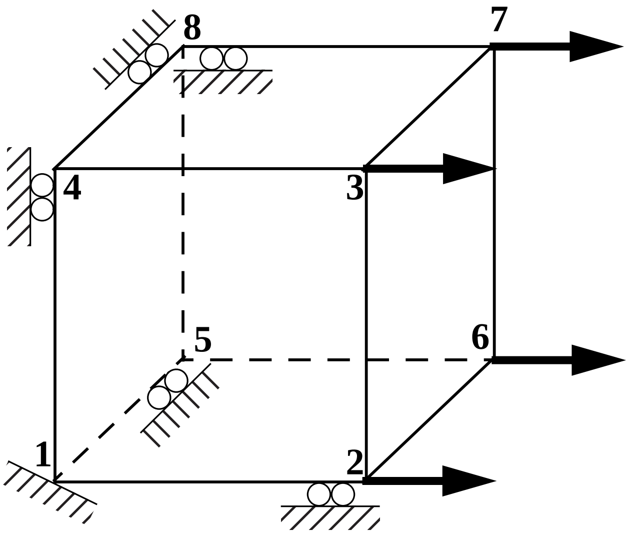


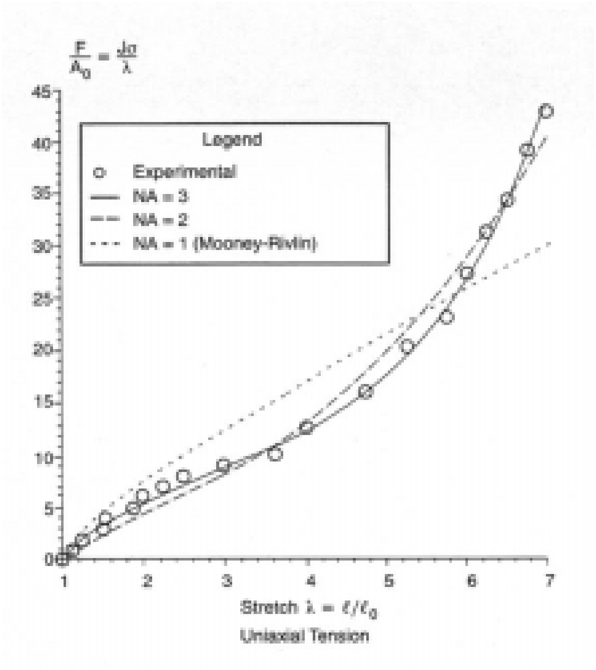
Objectives:

- Demonstrate the use of hyperelastic material properties.
- Create an accurate deformation plot of the model.

Model Description:

For the structure below:





Add Case Control commands and Bulk Data Entries to:

1. Model the hyperelestic behavior of the material.

Suggested Exercise Steps:

- Modify the existing MSC/NASTRAN input file by adding the appropriate nonlinear static analysis control parameters.
- Prepare the model for a nonlinear static analysis (SOL 106).
- Set up the appropriate subcase loading and analysis parameters (LOAD, NLPARM, SPC)
- Input the proper hyperelastic material property for the nonlinear material (MATHP)
- Generate an input file and submit it to the MSC/NASTRAN solver for a nonlinear static analysis.
- Review the results.

Input File for Modification:

prob5.dat

```
ID NAS103 WORKSHOP 5 SOLUTION
SOL 106
TIME 30
CEND
$
TITLE = SIMPLE TENSION
SUBTITLE = DISPLACEMENT CONTROL  $MPC's + 1 SPCD
$                                $AUTO
ECHO=UNSORT
DISP=ALL
STRESS(PLOT)=ALL
GPSTRESS=ALL
FORCE=ALL
MPC=100
SUBCASE 100 $ UNIAXIAL TENSION
$
$
$
BEGIN BULK
PARAM,POST,-1
PARAM,LGDISP,1
NLPARM, 10, 48, , AUTO, 1, , , YES
$
$
$
$ Treloar [1944] data in simple tension.  Nominal stresses in kgcm-2
$
TABLES1, 100, , , , , , , ,
+, 1., 0., 1.125, 1., 1.25, 2., 1.5, 3.,
+, 1.525, 4., 1.875, 5., 2., 6., 2.25, 7.,
+, 2.5, 8., 3., 9., 3.625, 10., 4., 12.5,
+, 4.75, 16., 5.25, 20., 5.75, 23., 6., 27.,
+, 6.25, 31., 6.5, 34., 6.75, 38.75, 7., 42.5,
+, endt
$
```

```
$ Treloar [1944] data in equibiaxial tension.  Nominal stresses in kg-cm-2
$
TABLES1, 200, , , , , , , ,
+, 1.016, 0.83, 1.07, 1.56, 1.15, 2.55, 1.21, 3.25,
+, 1.335, 4.31, 1.44, 5.28, 1.66, 6.65, 1.93, 7.88,
+, 2.46, 9.74, 3., 12.64, 3.4, 14.61, 3.77, 17.33,
+, 4.1, 20.11, 4.32, 22.40, 4.54, 24.41, endt
$
$ Treloar [1944] data in pure shear.  Units of the nominal stress are kgcm-2
$
TABLES1, 400, , , , , , , ,
+, 1.01, 0.24, 1.1, 1.12, 1.2, 1.92, 1.31, 2.87,
+, 1.49, 3.62, 1.86, 5.61, 2.36, 7.2, 2.97, 9.045,
+, 3.45, 10.8, 3.93, 12.45, 4.4, 14.23, 4.714, 15.826,
+, 4.96, 17.46, endt
$
GRID, 1, , 0., 0., 0., , 123456
GRID, 2, , 1., 0., 0., , 23456
GRID, 3, , 1., 1., 0., , 3456
GRID, 4, , 0., 1., 0., , 13456
GRID, 5, , 0., 0., -1., , 12456
GRID, 6, , 1., 0., -1., , 2456
GRID, 7, , 1., 1., -1., , 456
GRID, 8, , 0., 1., -1., , 1456
$
MPC, 100, 2, 1, 1., 7, 1, -1.
MPC, 100, 3, 1, 1., 7, 1, -1.
MPC, 100, 6, 1, 1., 7, 1, -1.
MPC, 100, 3, 2, 1., 7, 2, -1.
MPC, 100, 4, 2, 1., 7, 2, -1.
MPC, 100, 8, 2, 1., 7, 2, -1.
MPC, 100, 5, 3, 1., 7, 3, -1.
MPC, 100, 6, 3, 1., 7, 3, -1.
MPC, 100, 8, 3, 1., 7, 3, -1.
$
CHEXA, 1, 1, 1, 2, 3, 4, 5, 6,
+, 7, 8
SPC, 100, 7, 1,
SPCD, 1000, 7, 1, 6.
ENDDATA
```

Exercise Procedure:

1. **Currently hyperelasticity cannot be modeled using MSC/PATRAN. All users will need to modify the NASTRAN template file manually and submit it for an analysis. Proceed to the next step.**

Generating an input file for MSC/NASTRAN Users:

2. MSC/NASTRAN users can generate an input file using the data from the Model Description. MSC/PATRAN users must modify the Nastran template file before submitting it for an analysis. The result should be similar to the output below (**prob5.dat**):

```
ASSIGN OUTPUT2 = 'prob5.op2' , UNIT = 12
ID NAS103 WORKSHOP 5 SOLUTION
SOL 106
TIME 30
CEND
$
TITLE = SIMPLE TENSION
SUBTITLE = DISPLACEMENT CONTROL  $MPC's + 1 SPCD
$                         $AUTO
ECHO=UNSORT
DISP=ALL
STRESS(PLOT)=ALL
GPSTRESS=ALL
FORCE=ALL
MPC=100
SUBCASE 100 $ UNIAXIAL TENSION
  NLPARM=10
  SPC=100
  LOAD=1000
BEGIN BULK
PARAM,POST,-1
PARAM,LGDISP,1
NLPARM, 10, 48, , AUTO, 1, , , YES
$
$
MATHP, 1, , 1500., , , , , ,
+, , 3, 1, , , , , ,
+, , , , , , , , ,
+, , , , , , , , ,
+, , , , , , , , ,
+, , , , , , , , ,
+,100, 200, , 400
$
$ Treloar [1944] data in simple tension.  Nominal stresses in kgcm-2
$
TABLES1, 100, , , , , , , ,
+, 1., 0., 1.125, 1., 1.25, 2., 1.5, 3.,
+, 1.525, 4., 1.875, 5., 2., 6., 2.25, 7.,
```

```
+, 2.5, 8., 3., 9., 3.625, 10., 4., 12.5,
+, 4.75, 16., 5.25, 20., 5.75, 23., 6., 27.,
+, 6.25, 31., 6.5, 34., 6.75, 38.75, 7., 42.5,
+, endt
$
$ Treloar [1944] data in equibiaxial tension.  Nominal stresses in kg-cm-2
$
TABLES1, 200, , , , , , , ,
+, 1.016, 0.83, 1.07, 1.56, 1.15, 2.55, 1.21, 3.25,
+, 1.335, 4.31, 1.44, 5.28, 1.66, 6.65, 1.93, 7.88,
+, 2.46, 9.74, 3., 12.64, 3.4, 14.61, 3.77, 17.33,
+, 4.1, 20.11, 4.32, 22.40, 4.54, 24.41, endt
$
$ Treloar [1944] data in pure shear.  Units of the nominal stress are kgcm-2
$
TABLES1, 400, , , , , , , ,
+, 1.01, 0.24, 1.1, 1.12, 1.2, 1.92, 1.31, 2.87,
+, 1.49, 3.62, 1.86, 5.61, 2.36, 7.2, 2.97, 9.045,
+, 3.45, 10.8, 3.93, 12.45, 4.4, 14.23, 4.714, 15.826,
+, 4.96, 17.46, endt
$
GRID, 1, , 0., 0., 0., , 123456
GRID, 2, , 1., 0., 0., , 23456
GRID, 3, , 1., 1., 0., , 3456
GRID, 4, , 0., 1., 0., , 13456
GRID, 5, , 0., 0., -1., , 12456
GRID, 6, , 1., 0., -1., , 2456
GRID, 7, , 1., 1., -1., , 456
GRID, 8, , 0., 1., -1., , 1456
$
MPC, 100, 2, 1, 1., 7, 1, -1.
MPC, 100, 3, 1, 1., 7, 1, -1.
MPC, 100, 6, 1, 1., 7, 1, -1.
MPC, 100, 3, 2, 1., 7, 2, -1.
MPC, 100, 4, 2, 1., 7, 2, -1.
MPC, 100, 8, 2, 1., 7, 2, -1.
MPC, 100, 5, 3, 1., 7, 3, -1.
MPC, 100, 6, 3, 1., 7, 3, -1.
MPC, 100, 8, 3, 1., 7, 3, -1.
$
CHEXA, 1, 1, 1, 2, 3, 4, 5, 6,
+, 7, 8
PLSOLID, 1, 1
```

```
SPC, 100, 7, 1
SPCD, 1000, 7, 1, 6.
ENDDATA
```

Submit the input file for analysis:

3. Submit the input file to MSC/NASTRAN for analysis.

   3a. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob5.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

4. When the analysis is completed, edit the **prob5.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing the WARNING messages indicate any modeling errors.

   4a. While still editing **prob5.f06**, search for the word:

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

What is the x-displacement of Node 2 at the end of the analysis?

T1= ____________________

What is the x-displacement of Node 5 at the end of the analysis?

T1= ____________________

Comparison of Results:

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

0
SUBCASE 100 $
UNIAXI
LOAD STEP = 1.00000E+00

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	0.0	0.0	0.0	0.0	0.0
2	G	6.000000E+00	0.0	0.0	0.0	0.0	0.0
3	G	6.000000E+00	-6.170447E-01	0.0	0.0	0.0	0.0
4	G	0.0	-6.170447E-01	0.0	0.0	0.0	0.0
5	G	0.0	0.0	6.170447E-01	0.0	0.0	0.0
6	G	6.000000E+00	0.0	6.170447E-01	0.0	0.0	0.0
7	G	6.000000E+00	-6.170447E-01	6.170447E-01	0.0	0.0	0.0
8	G	0.0	-6.170447E-01	6.170447E-01	0.0	0.0	0.0

6. This ends the exercise for MSC/NASTRAN users. MSC/PATRAN users should proceed to the next step.

7. Create a new database called **prob5.db**.

File/New...

New Database Name:	**prob5**
OK	

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

Tolerance:	● **Default**
Analysis Code:	**MSC/NASTRAN**
Analysis Type:	**Structural**
OK	

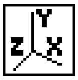
8. Proceed with the Reverse Translation process, that is, importing the **prob5.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

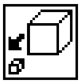
Action:	**Read Output2**
Object:	**Both**
Method:	**Translate**
Select Results File...	
Selected Results File:	**prob5.op2**
OK	
Apply	

9. Post process the results from the analysis.

Alter the view angle to better see the deformation results by clicking on the following toolbar icons:



Iso 1 View



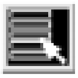
Zoom Out (click twice)

Finally, use the **Results** form to display the deformation of the model.

◆ **Results**

Action:	**Create**
Object:	**Deformation**

Now click on the **Select Results** icon.

 **Select Results**

Select Result Case(s)	*(Select all cases.)*
Select Fringe Result	**Displacements, Translational**
Show As:	**Resultant**

Click on the **Display Attributes** icon.

 **Display Attributes**

In order to see the deformation results accurately, set the Scale Interpretation to True Scale with a Scale Factor of 1.

Scale Interpretation	● *True Scale*
Scale Factor	**1.0**

■ *Show Undeformed*

Now click on the **Plot Options** icon.

 **Plot Options**

Coordinate Transformation:	**None**
Scale Factor	**1.0**

Apply

Notice how drastically the shape of the hyperelastic element changed. The height and the wideth of the element shrinked by more thean half of their original dimensions in order to compensate for the deformation in the x-dirctiion.

Quit MSC/PATRAN when you have completed this exercise.