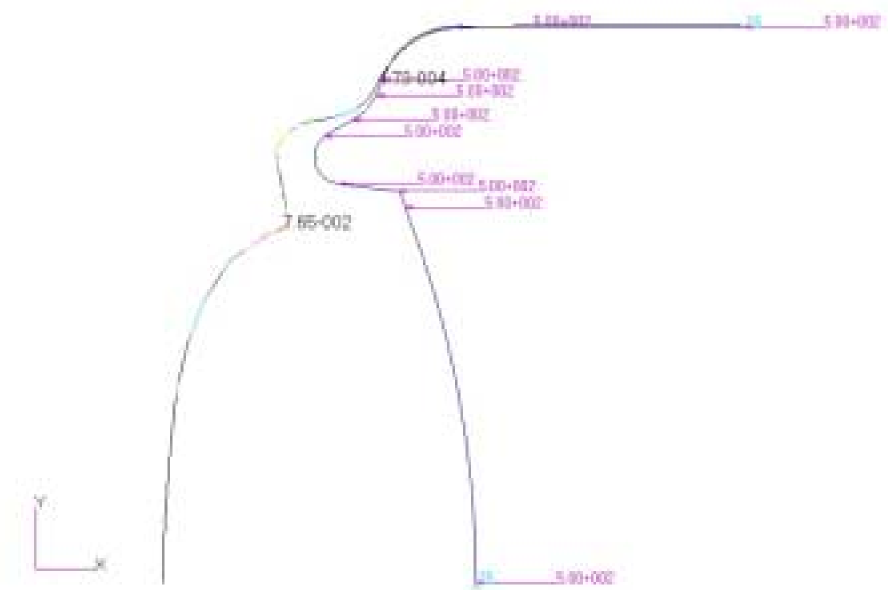
LESSON 7

Soft-Drink Can's Bottom Snap-through



Objectives:

This chapter demonstrates the modeling and analysis of the bottom of an aluminum container under internal pressure. The particular configuration of this container bottom leads to a snap-through problem. Accurate modeling of the geometry is essential since it dramatically influences the snap-through process. The primary goal of this chapter is to show you three Patran functionalities.

- Using nonlinear analysis to solve a snap-through analysis problem.
- Using the TABLES option to specify input data that changes with time, plastic stream, etc.
- Animating the results of an analysis.

Model Description:

The container, a soft drink can, is assumed to be a circle cylinder with a radius of 1.3 inches and a total height of 4.8 inches. The container is made out of aluminum and has a wall thickness of 0.025 inches.

The geometry of this problem is fairly simple due to two factors. The first is that the geometry and the loading of the container are axisymmetric and allow you to perform an axisymmetric analysis. The second factor is that the focus of the analysis is restricted to the phenomena that occurs at the bottom of the container. In this analysis, the height of the container, h, is limited to a length where the edge effects are damped out. The theory behind this assumption is explained below.

IF

$$h = 25\sqrt{rt}$$

where r = the radius of the container, t = the wall thickness, the solution decreases to about 4% of its value at the bottom edge. In this sample, it means you can safely ignore the influence of the top ege since the critical height, h, is equal to 0.4519, calculated as follows:

$$h = 2.5\sqrt{1.307 \times 0.025} = 0.4519$$

An awareness of this decay distance is very important in numerical calculations. If you wish to correctly capture the behavior of the solution in the edge region, the typical finite element size must be small in comparison to the decay distance.

Suggested Exercise Steps:

- Create Geometry and mesh.
- Create pressure load and displacement BCs.
- Setup and run analysis, monitor, attach and plot results.

Exercise Procedure:

1. Create a new file called **container.db**.

File/New ...

Database Name:	**container.db**
OK	
Analysis Code:	**MSC.Marc**
OK	

2. Create the Geometry.

Here we describe the step by step creation of the geometry for the container. As an alternative, you can create the geometry by playing the session file, "soft_drink_can_geom.ses". If you opt to play the session file, play it (File/Session/Play), and skip to Step 3 (Create Mesh Seeds). Otherwise, continue manually creating the geometry: Use 2D ArcAngles method to create arcs.

◼ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**2D ArcAngles**
Radius:	**2.354**
Start Angle:	**0.0**
End Angle:	**22**
Center Point List:	**[0, 0, 0]**
Apply	
Radius:	**0.063**
Start Angle:	**129**
End Angle:	**265**
Center Point List:	**[2.026, 1, 0]**
Apply	
Radius:	**0.125**
Start Angle:	**296**

End Angle:	**334**
Center Point List:	**[2, 1.2, 0]**

Apply

Radius:	**0.18**
Start Angle:	**90**
End Angle:	**162**
Center Point List:	**[2.3, 1.127, 0]**

Apply

Radius:	**0.05**
Start Angle:	**0**
End Angle:	**50**
Center Point List:	**[2.1326, 0.8818, 0]**

Apply

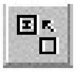
Use the XYZ method to create a Point.

Action:	**Create**
Object:	**Point**
Method:	**XYZ**
Point Coordinates List:	**[3, 1.307, 0]**

Apply

Use Point method to create straight curves.

It might be beneficial to first set the **Show Labels** option.


Show Labels

Action:	**Create**
Object:	**Curve**
Method:	**Point**

Option:	**2 Point**
Starting Point List:	**Point 4**
Ending Point List:	**Point 9**
Apply	
Starting Point List:	**Point 3**
Ending Point List:	**Point 5**
Apply	
Starting Point List:	**Point 6**
Ending Point List:	**Point 8**
Apply	
Starting Point List:	**Point 7**
Ending Point List:	**Point 10**
Apply	

3. Create Mesh Seeds.

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**
Number =:	**8**
Curve List:	**Curve 1 9**
Apply	
Number =:	**6**
Curve List:	**Curve 2 4**
Apply	
Number =:	**4**
Curve List:	**Curve 3 6**
Apply	
Number =:	**3**
Curve List:	**Curve 5 7 8**

Apply

See Figure 7.1 to view the locations of the curves.

Figure 7.1 - Locations of the curves for Mesh Seeds.

4. Mesh the container.

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Curve List:	Curve 1:9

Apply

5. Equivalence.

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**

Apply

6. Create the three required Displacement Boundary Conditions.

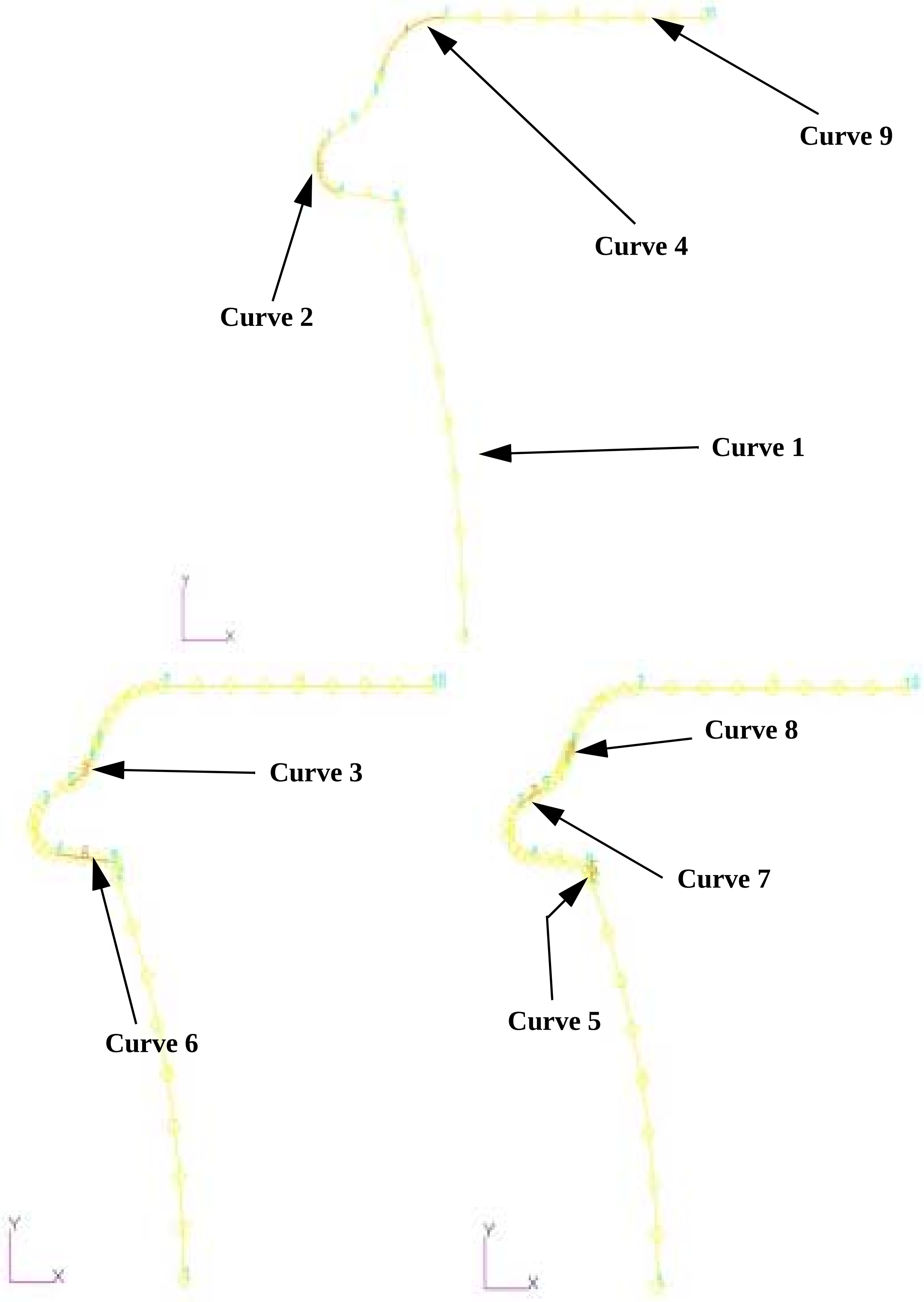
■ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	fixed-y

Input Data...

Translations:	< , 0, >
Rotations:	< , , 0>

OK



Select Application Region...

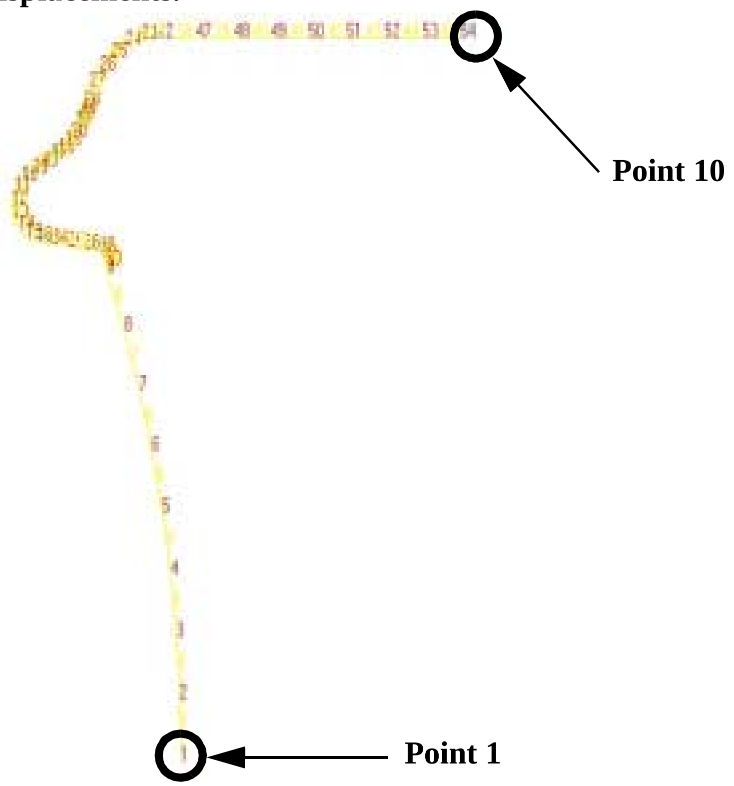
Select Geometry Entities:	select from Figure 7.2 **Point 1**

Add

OK

Apply

Figure 7.2 - Locations of Point 1 and Point 10 for fixed displacements.



■ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**fixed-x**

Input Data...

Translations:	**<0, , >**
Rotations:	**< , , 0>**

OK

Select Application Region...

Select Geometry Entities:	select from Figure 7.2 **Point 10**

Add

OK

Apply

Apply pressure on the curve

Action:	**Create**
Object:	**CID Distributed Load**
Type:	**Element Uniform**
New Set Name:	**pressure**
Target Element Type:	**1D**

Input Data...

Distr Force <F1 F2 F3>:	**<-500, 0, 0>**

OK

Select Application Region...

Select Curves:	select all the curves **Curve 1:9**

Add

OK

Apply

7. Plot Markers.

■ **Loads/BCs**

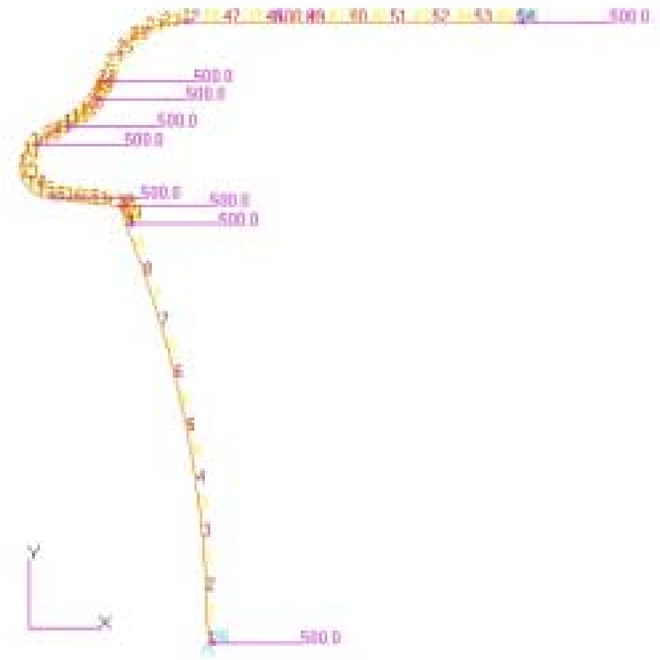
Action:	**Plot Markers**

Assigned Load/BC Sets:	select all sets: **CID D_pressure** **Displ_fixed-x** **Displ_fixed-y**
Select Groups:	**default_group**

Apply

Your screen should show **Figure 7.3**.

Figure 7.3 - Your model, including Plot Markers.



8. Create a field for the elastic-plastic material data.

■ **Fields**

Action:	**Create**
Object:	**Material Property**
Method:	**Tabular Input**
Field Name:	**plastic_s_e**

Active Independent Variable: ❐ **Temperature (T)** ■ **Strain (e)**

Input Data...

The Stress vs. Strain Table will have the following values.

	e	Value
1	**0**	**42000**
2	**0.001748**	**44577**
3	**0.003494**	**45157**
4	**0.06766**	**63665**
5	**0.09531**	**70950**
6	**0.157**	**81315**
7	**0.207**	**88560**
8	**0.2623**	**95216**

OK

Apply

9. Create the Material *aluminum*, with elastic and plastic properties.

■ **Materials**

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Material Name: alumimum

Input Properties...

Constitutive Model: **Elastic**

Elastic Modulus: 11E6

Poisson Ratio: 0.3

OK

Apply

Input Properties...

Constitutive Model:	**Plastic**
Hardening Rule:	**Isotropic**
Stress vs. Plastic Strain:	choose from Field list box, **plastic_s_e**

OK

Apply

10. Define the element Properties.

■ **Properties**

Action:	**Create**
Object:	**1D**
Type:	**Axisym Shell**
Property Set Name:	**container**
Option(s):	**Homogeneous**
	Standard Formulation

Input Properties...

Material Name:	choose from the list box, **aluminum**
Thickness:	**0.025**

OK

Select Members:	select everything on the screen, **Curve 1:9**

Add

Apply

11. Begin setting up the Analysis to run.

■ **Analysis**

Action:	**Analyze**

Object:	**Entire Model**
Method:	**Full Run**
Job Name:	**container_job1**
Translation Parameters...	
Solver Options...	
	■ **Non-Positive Definite**
OK	
OK	
Load Step Creation...	
Job Step Name:	**One**
Solution Type:	**Static**
Solution Parameters...	
Linearity:	**NonLinear**
Nonlinear Geometric Effects:	**Large Displacement/ Large Strains**
	■ **Loads Follow Deformations**
Load Increment Parameters...	
Arclength Method:	**Modified Riks/Ramm**
Initial Fraction of Load Applied in 1st Increment:	**0.05**
Max. Fraction of Load Applied in Any Increment:	**0.1**
Max # of Increments:	**600**
OK	
Iteration Parameters...	
Error Type:	**Relative**
Residual Force:	**0.05**
OK	
OK	

Output Requests...	
Select Nodal Results...	**ROTATION (2)** **EXTERNAL FORCE (3)**
OK	
Select Element Results...	**STRAIN, PLASTIC EQUIVALENT (27)**
Number of Integration Pts. thru Section:	**5**
Write Results for Integration Pts. (list):	**1 2 3 4 5**
OK	
OK	
Apply	
Cancel	
Load Step Selection...	
Selected Job Steps:	**One**
OK	
Apply	

12. Monitor the analysis by monitoring the **.sts, .log**, or **.out** files. Also, the **Monitor** option is readily available.

Action:	**Monitor**
Object:	**Job**
View Status File...	

After the job has finished, a successful completion will result with:
Job ends with exit number: 3004.

13. Read back the Results.

■ **Analysis**

Action:	**Read Results**

Object: **Result Entities**

Method: **Attach**

Select Results File...

Selected Results File: **Container.t16**

OK

Apply

14. Display Quick Plots of von Mises Stress & Displacements.

■ **Results**

Action: **Create**

Object: **Quick Plot**

Selected Result Cases: select the last results case, **One, A1:...Time=1.0s**

Selected Fringe Result: **Strain, Total**

Quantity: **von Mises**

Selected Deformation Result: **Displacement, Translation**

Apply

Figure 7.4 - The displacement of the can bottom.

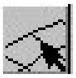
15. Display Graph by using the X vs Y method.

Action: **Create**

Object: **Graph**

Method: **Y vs X**

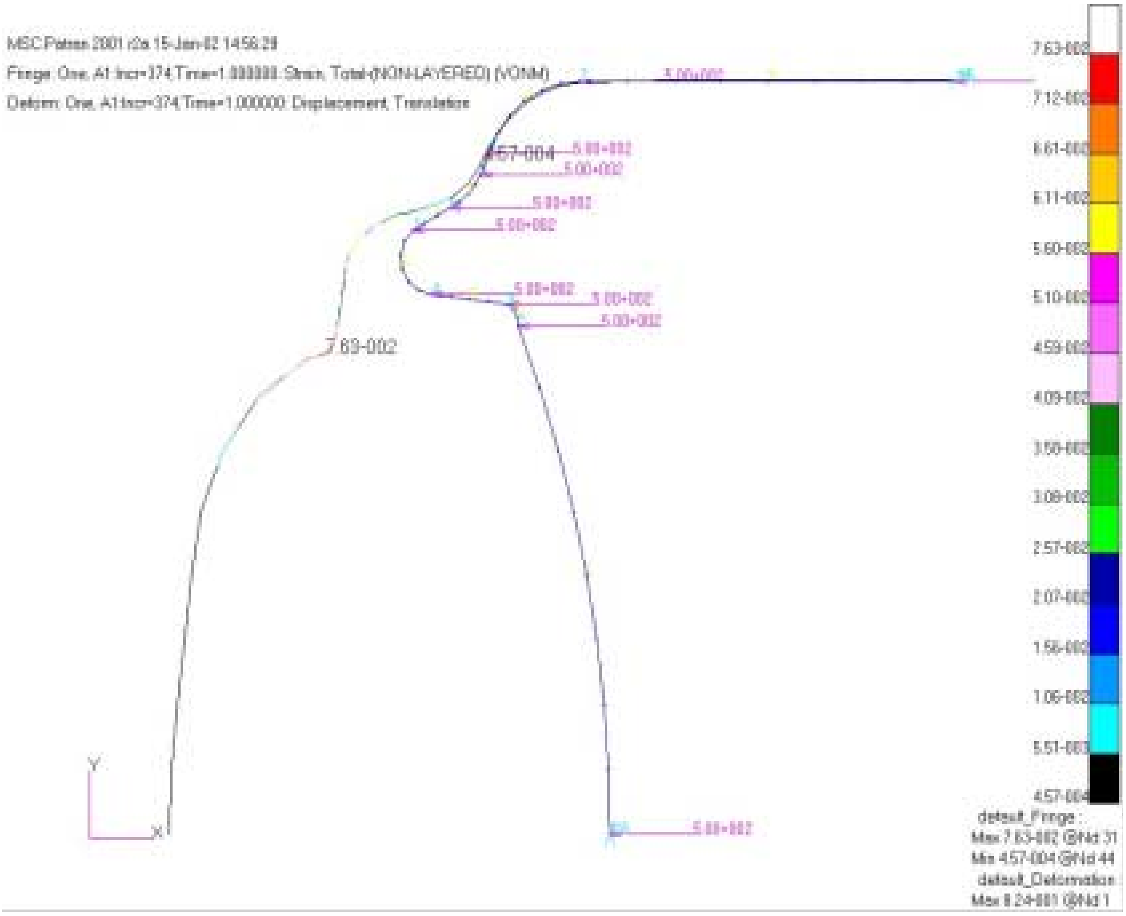
Use the **Target Entities** function for the entities to be graphed.

 **Target Entities**

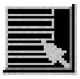
Target Entity: **Elements**

Select Elements: input these values (random elements not close to each other): **Elm 1 13 18 27 45**

Soft-Drink Can’s Bottom Snap-through



Click on the Select Results icon



Select Results

Select Result Case(s): **One**

Choose all cases from the form that automatically opens.

Filter

Apply

Close

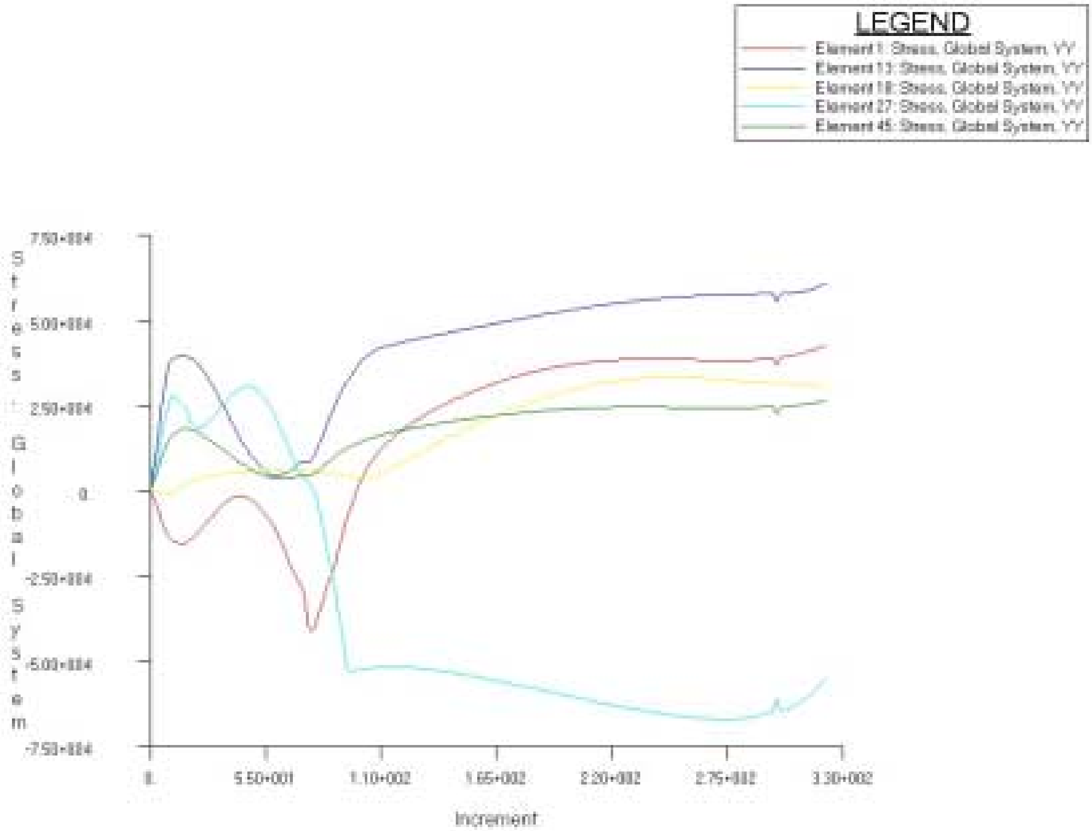
Y: **Result**

Select Y Result: **Stress, Global System**

Position...(At Layer 1)

Select:	**At Layer 3**
Close	
Quantity:	**Y Component**
Apply	

Figure 7.5 - Random Elements selected for Y results of Stress.



Close the database and quit PATRAN.

This concludes this exercise.