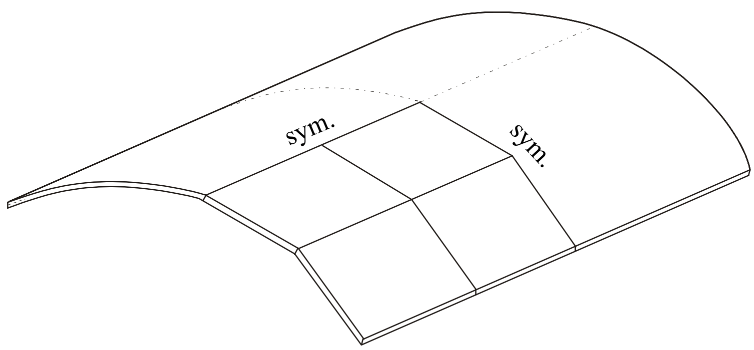
Workshop 6

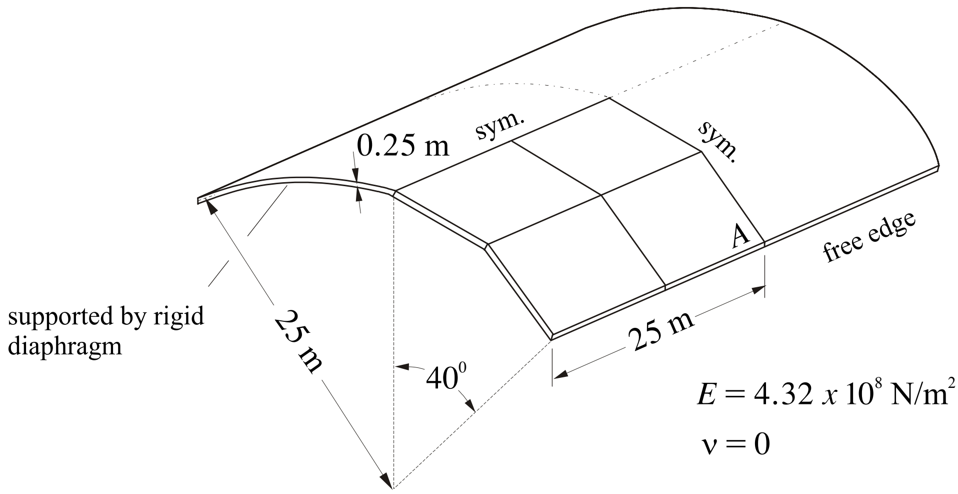
Linear Static Analysis of a Cylindrical Shell Panel



Objectives

- Create a geometric representation of a cylindrical shell panel.
- Use the geometry model to define an analysis model comprised of plate elements.
- Run an MSC/NASTRAN linear static analysis.
- View analysis results.

Model Description



The cylindrical shell roof shown above is loaded by its own weight. Straight edges are free and curved edges have diaphragm support, meaning that translational d.o.f. parallel to the plane containing the curve are prohibited but translational d.o.f. normal to this plane and all rotational d.o.f. are unrestrained. Consider the acceleration due to gravity as 9.81 m/s^2 and 36.7 kg/m^3 for the mass density.

Exercise Procedure

1. Start up **MSC/NASTRAN for Windows 4.5** and begin to create a new model.

   Double click on the icon for the **MSC/NASTRAN for Windows V4.5**.

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

   Turn off the workplane:

   Tools / Workplane (or **F2**) / ☐ **Draw Workplane / Done**

   View / Regenerate (or **Ctrl G**).

2. Create a material called **mat_1**.

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

Title	**mat_1**
Young's Modulus	**4.32e8**
Shear Modulus	**2.16e8**
Mass Density	**36.7**

   Select **OK** / **Cancel**.

   NOTE: In the *Messages Window* at the bottom of the screen, you should see a verification that the material was created. You can check here throughout the exercise to both verify the completion of operations and to find an explanation for errors which might occur.

3. Create a property called **prop_1** to apply to the members of the shell.

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

Title	**prop_1**
Material	**mat_1**

   Note that the default element type is **Plate** element, **not parabolic**.

Thickness, Tavg or T_1	**0.25**

Select **OK** / **Cancel**.

4. Create the geometry for a circular edge.

Due to symmetry considerations, only half of the geometry needs to be created.

Create a 40° arc with radius of 25 m:

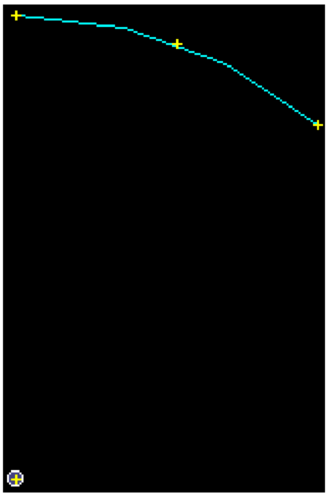
Geometry / **Curve-Arc** / **Radius-Start-End**

CSys		**Basic Cylindrical**						
Start of Arc	R	**25**	T	**50**	Z	**0**		**OK**
End of Arc	R	**25**	T	**90**	Z	**0**		**OK**
Radius		**25**		**OK**				

Select **Cancel**.

To fit the display onto the screen, select **View** / **Autoscale** / **Visible** (or **Ctrl A**).

On your display, there should now be half of a circular edge.



5. Create the geometry for the surface of the shell.

The cylindrical surface may be created by extruding the above curve along a vector.

Geometry / Surface / Extrude

Select the curve / **OK**.

Specify a vector to extrude de curve along:

Base	R **25**	T **90**	Z **0**
Tip	R **25**	T **90**	Z **-25**

Select **OK** / **Cancel**.

View / Rotate / Trimetric / OK.

To fit the display onto the screen, select **View / Autoscale / Visible** (or **Ctrl A**).

6. Define the number of elements along the edges.

Mesh / Mesh Control / Size Along Curve

Select the four edges of the surface / **OK**.

Number of Elements	**8**
Node Spacing	✓ **Equal**

Select **OK** / **Cancel**.

7. Generate the finite elements (8×8 mesh of QUAD4).

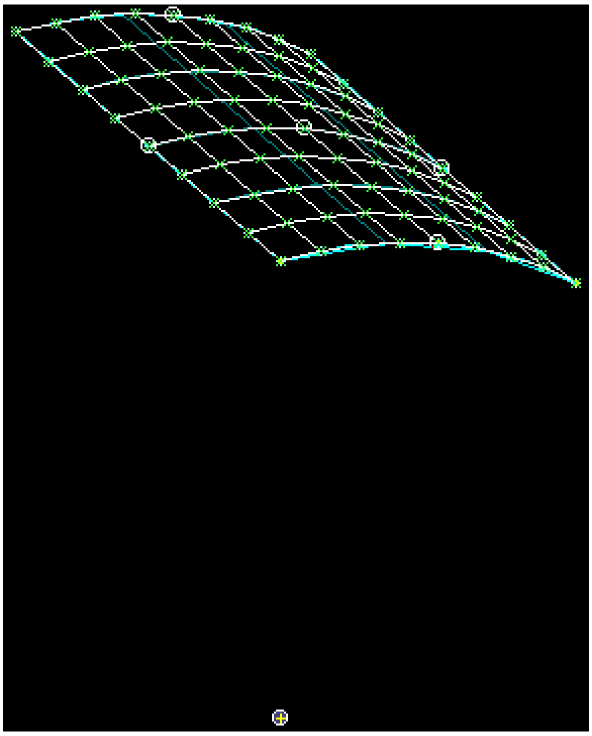
Mesh / Geometry / Surface

Select the surface / **OK**.

Property	**prop_1**

Select **OK**.

Note that **Quad** is the default element shape. So, **Plate** + **not parabolic** (linear) + **Quad** = **QUAD4**. Due to symmetry considerations, just one quarter of the shell is modelled.



Observe that the mesh is different from that shown in page 6-2.

8. Create the model constraints.

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

   Model / Constraint / Set

Title	**constraint_1**

   Select **OK**.

   This constraint set will have 3 different constraints.

   Now, define the first relevant constraint for the model.

   Model / Constraint / Nodal / Method^/ on Curve

   Select the edge $Z = 0$ / **OK**.

   On the *DOF* box, select

☑ **TX** ☑ **TY**

Select **OK**.

Next, define the second relevant constraint for the model.

Method^/ on Curve

Select the left edge / **OK**.

On the *DOF* box, select **X Symmetry / OK**.

A warning message will appear: *Selected Constraints Already Exist. OK to Overwrite (No = Combine)?* Select **No** to combine.

Finally, define the third relevant constraint for the model.

Method^/ on Curve

Select the edge $Z = -25$ / **OK**.

On the *DOF* box, select **Z Symmetry / OK**.

A warning message will appear: *Selected Constraints Already Exist. OK to Overwrite (No = Combine)?* Select **No** to combine and, then, **Cancel**.

View / Regenerate (or **Ctrl G**).

9. Create the model loading.

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

Model / Load / Set (or **Ctrl F2**)

Title **load_1**

Select **OK**.

Now, define the body load.

Model / Load / Body

Acceleration

✓ *Active*	*X* **0**	*Y* **-9.81**	*Z* **0**

Select **OK**.

10. Run the analysis.

File / Analyze

Analysis Type	**Static**
Loads	✓ **load_1**
Constraints	✓ **constraint_1**
	✓ **Run Analysis**

Select **OK**.

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

Be sure to set the desirable working directory.

File Name	**work_6**

Select **Save**.

When the MSC/ NASTRAN manager is through running, MSC/ NASTRAN for Windows 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 detail this time. Then select **Continue**.

11. Display the deformed plot on the screen.

You may now display the deformed plot. First, however, you may want to remove the boundary constraint markers.

View / Options / Quick Options (or **Ctrl Q**)

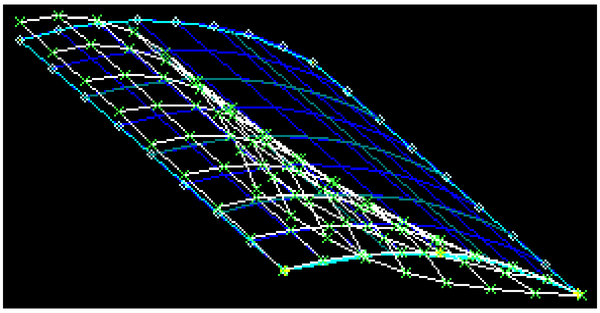
☐ **Constraint / Done / OK**.

Plot the deformation of the shell.

View / Select (or **F5**)

Deformed Style	**Deform**
	Deformed and Contour Data
Output Vectors / Deformation	**Total Translation**

Select **OK / OK**.



What is the vertical displacement at point A?

The answer is easily given clicking **Off**, at the right bottom side of the screen, and then selecting **Node**.

View / Rotate / ZX Front / OK.

The required deflection will come out when the cursor is left next to the node at point A.

This concludes the exercise.

File / Save

File / Exit.

Answer

Vertical displacement at point A	-0.3069

The value -0.3024 m is recommended as the correct vertical displacement at point A (MacNeal, R. H., and Harder, R. L., 1985, "A proposed standard set of problems to test finite element accuracy", *Finite Elements in Analysis and Design*, Vol. 1, pp. 3-20).

In this workshop a continuously curved surface is represented by a surface built up of flat elements. Common advice is that a flat shell element should span no more than roughly 10° of the arc of the actual shell. The use of smaller elements improves the result not only because the domain of the interpolated variables is reduced, but also because a better representation of the geometry is obtained. Would you like to improve the result by refining the mesh?