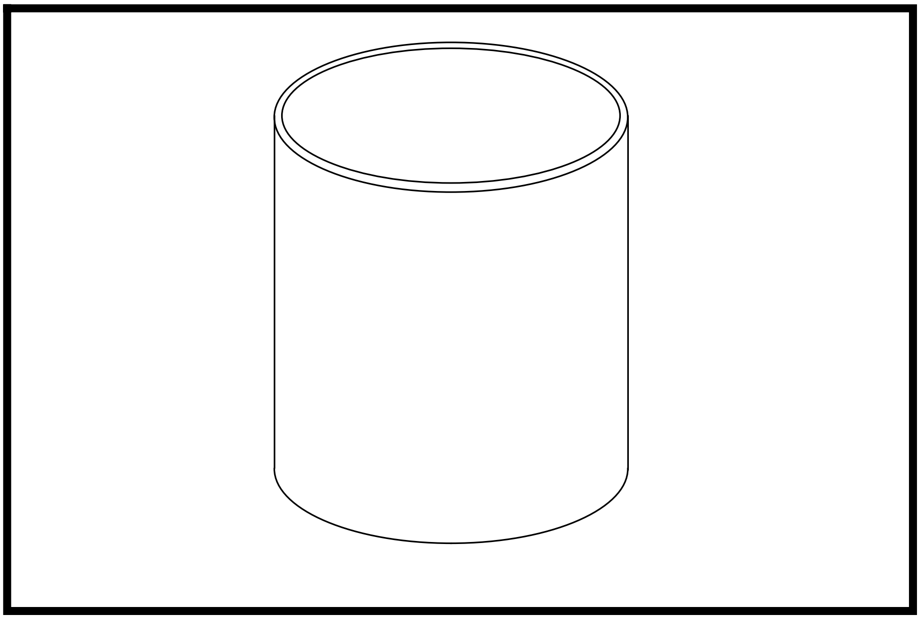
LESSON 14

Buckling Analysis of a Thin-Shell Cylinder

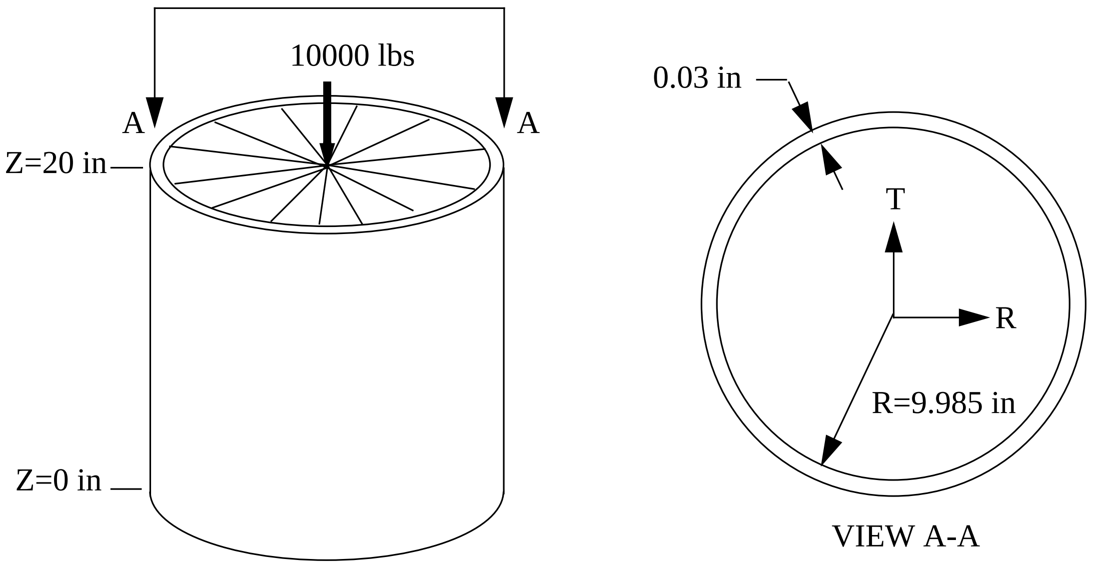


Objectives:

- Create a geometrical representation of a thin-shell cylinder.
- Use the geometry model to define a MSC/NASTRAN analysis model comprised of CQUAD4 and RBE3 elements.
- Prepare a MSC/NASTRAN input file for a Buckling analysis.
- Visualize analysis results.

Model Description:

Below is a geometric representation of the thin-shell cylinder presented on Page 14-1. The cylinder has a 10 inch mean radius, sits 20 inches high and has a constant thickness of 0.03 inches. A compressive force is distributed to the cylinder via an RBE3 element (not shown). The cylinder is simply-supported at its base in all three translational directions. A similar constraint is applied to the top rim except the rim is allowed to translate in the Z-direction.



Elastic Modulus:	1.0E7 psi
Poisson Ratio:	0.3

Suggested Exercise Steps:

- Open a new database**.**
- Define a geometric representation of the cylinder using a surface.
- Define an analysis model by meshing the geometry model with shell (CQUAD4).
- Generate an RBE3 element.
- Define material (MAT1) and element properties (PSHELL).
- Define boundary constraints (SPC1) at the base & rim and apply a compressive load (FORCE) to the rim via the RBE3.
- Use the load and boundary condition sets to define a loadcase (SUBCASE).
- Prepare the model for a Buckling analysis (SOL 105 & PARAMs).
- Generate and submit input file for MSC/NASTRAN.
- Post-process results.
- Quit MSC/PATRAN.

Exercise Procedure:

1. Create a new database called **lesson14.db.**

File/New Database...

New Database Name:	**lesson14**
OK	

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

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

2. Use the **Viewing/Angles...** option to change the view of the model.

Viewing/ Angles...

Method: ◆ **Model Absolute**

Angles: **-67., 0, 34.**

Apply

Cancel

2a. Use the **Viewing/Named View Options...** to create a customized view setting.

Viewing/ Named View Options...

Create View...

Create New View: **nas120_iso**

Apply

Close

2b. We will need one other customized view setting.

Viewing/ Angles...

Method: ◆ **Model Absolute**

Angles: **-90., 0, 0.**

Apply

Cancel

Viewing/ Named View Options...

Create View...

Create New View: **nas120_side**

Apply

Also, change the view of the model.

Select Named View:	**nas120_iso**

Close

2c. Create a new point and use this point to define the origin of a local cylindrical coordinate system.

◆ **Geometry**

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

Apply

Action:	**Create**
Object:	**Coord**
Method:	**Euler**
Type:	**Cylindrical**

Rotation Parameters...

Angle of Rotation:	**0.0**
Angle of Rotation:	**0.0**
Angle of Rotation:	**0.0**

OK

Origin:	**[0, 0, 0]**

Apply

2d. Create another point by translating Point 1 a radial distance of 10 inches. Note that the reference coordinate frame is the local cylindrical system.

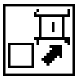
Action:	**Transform**
Object:	**Point**
Method:	**Translate**
Refer. Coordinate Frame:	**Coord 1**
Translation Vector:	**<10, 0, 0>**
■ **Auto Execute**	
Point List:	**Point 1**

There is no need to click the **Apply** button since the **Auto Execute** button is toggled on.

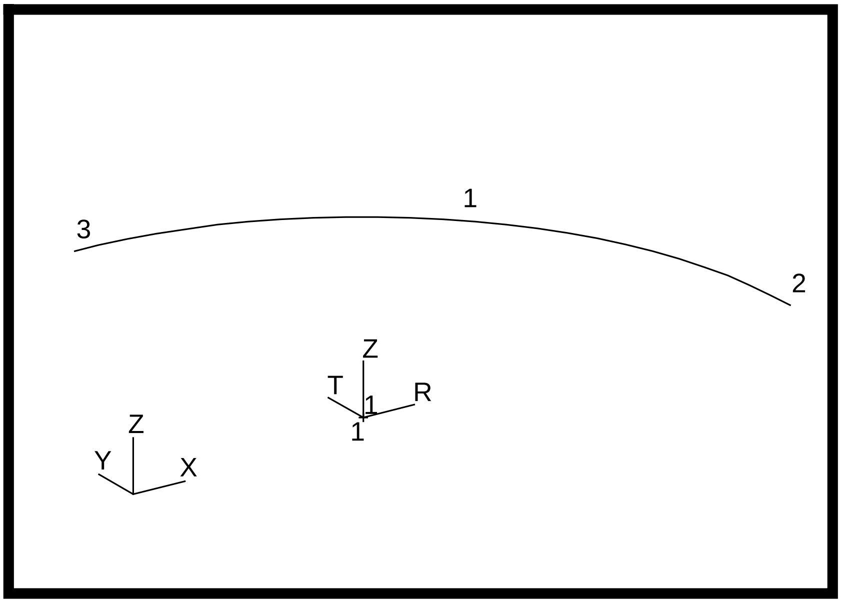
3. Use Point 2 to create a curve by sweeping it 90 degrees about the Z-axis of the local coordinate system.

Action:	**Create**
Object:	**Curve**
Method:	**Revolve**
Refer. Coordinate Frame:	**Coord 1**
Axis:	**Coord 0.3**
Total Angle:	**90.0**
■ **Auto Execute**	
Point List:	**Point 2**

Activate the entity labels by selecting the **Show Labels** icon on the toolbar.

 **Show Labels**

Your model should appear as follows:



3a. Generate one-quarter of the cylinder wall by extruding the curve created in the previous operation.

Action:	**Create**
Object:	**Surface**
Method:	**Extrude**
Refer. Coordinate Frame:	**Coord 1**
Translation Vector:	**<0, 0, 20>**
■ **Auto Execute**	
Curve List:	**Curve 1**

Now complete the model by rotating this surface about an local Z-axis.

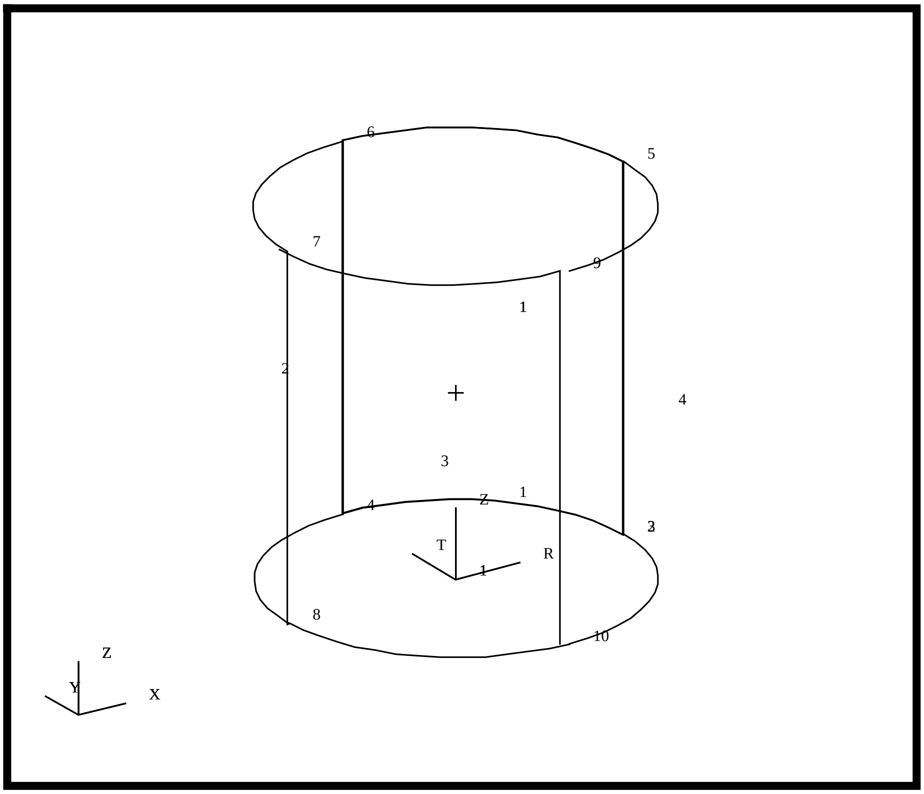
Action:	**Transform**
Object:	**Surface**
Method:	**Rotate**
Refer. Coordinate Frame:	**Coord 1**
Axis:	**Coord 0.3**
Total Angle:	**90.0**

Repeat Count: **3**

■ **Auto Execute**

Surface List: **Surface 1**

Your model should appear as follows:



4. Before you mesh the model, plant mesh seeds to define the size of the elements. Note that the mesh seeds will have precedence over the **Global Edge Length** parameter in the *Create/Mesh* form You may wish to make the nodes that will be generated during the meshing operation more visible. Do this by changing the node size using the **Display/Finite Elements...** option.

Display/Finite Elements...

Node Sizes: **5**

Apply

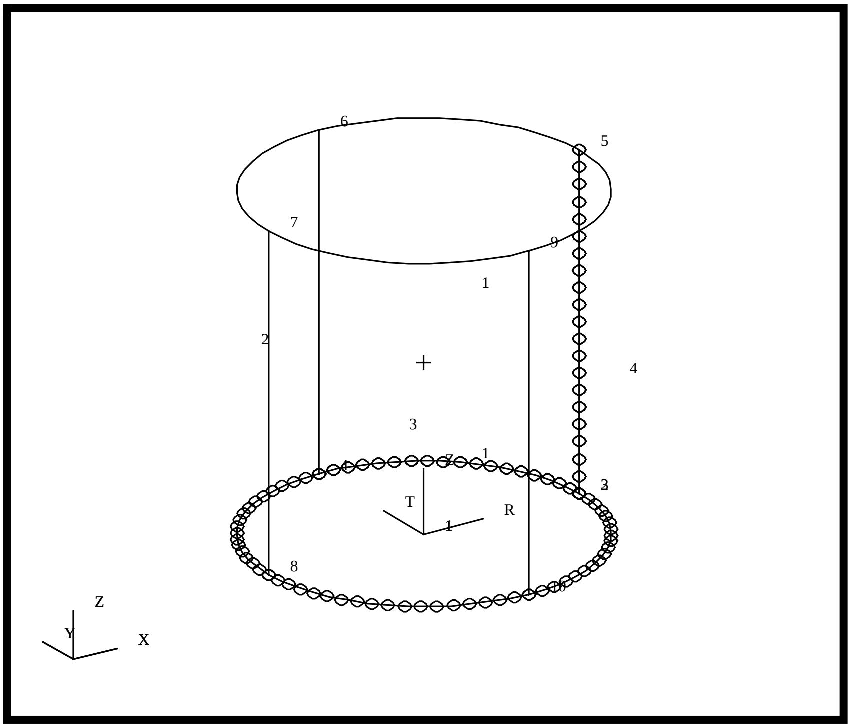
Cancel

4a. Create 18 elements along the bottom of the surfaces and 20 elements along the height.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**
Number =	**18**
Curve List:	**Surface 1:4.4**
Apply	
Number =	**20**
Curve List:	**Surface 1.1**
Apply	

The mesh seeds should appear as follows:



5. Generate an analysis model by meshing the geometry model.

Note: The node locations are to be specified with respect to our local cylindrical system (i.e. in r-θ–z coordinates). Hence, we control this by designating the local coordinate system as the Reference Coordinate frame.

However, it is desired to have our displacement results be generated with respect to our Global coordinate system. Therefore our Analysis Coordinate Frame references the Global coordinate system.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Element Topology:	**Quad4**
Mesher:	◆ **IsoMesh**

Node Coordinate Frames...

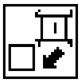
Analysis Coordinate Frame:	**Coord 0**
Refer. Coordinate Frame:	**Coord 1**

OK

Surface List:	**Surface 1:4**

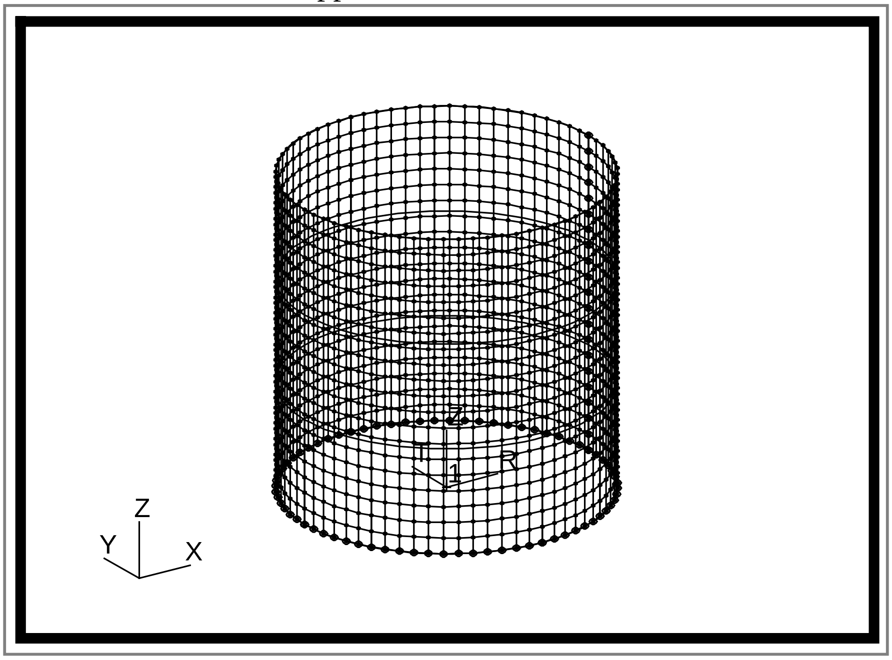
Apply

Dectivate the entity labels by selecting the **Hide Labels** icon on the toolbar



Hide Labels

You model should appear as follows:



Remember to equivalence the model to remove duplicate nodes at common surface edges.

◆ **Finite Elements**

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

Apply

6. Use the **Viewing/Named View Options...** to change the view of the model.

Viewing/ Named View Options...

Select Named View:	**default_view**

Close

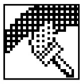
7. **Erase All FEM** for clarity using the option in **Display/ Plot/Erase....**.

Display/Plot/Erase...

Erase All FEM

OK

Refresh the display when needed using the brush icon on the Top Menu Bar.



Refresh Graphics

7a. Create a new node at Z=20 inches. The compressive load will be applied to this node. The Analysis & Reference coordinate frames are set in accordance with the previous Step.

◆ **Finite Elements**

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node ID List:	**5000**
Analysis Coordinate Frame:	**Coord 0**
Refer. Coordinate Frame:	**Coord 1**

❑ **Associate with Geometry**

Node Location List:	**[0, 0, 20]**

Apply

7b. Turn on node labels.

Display/Entity Color/Label/Render...

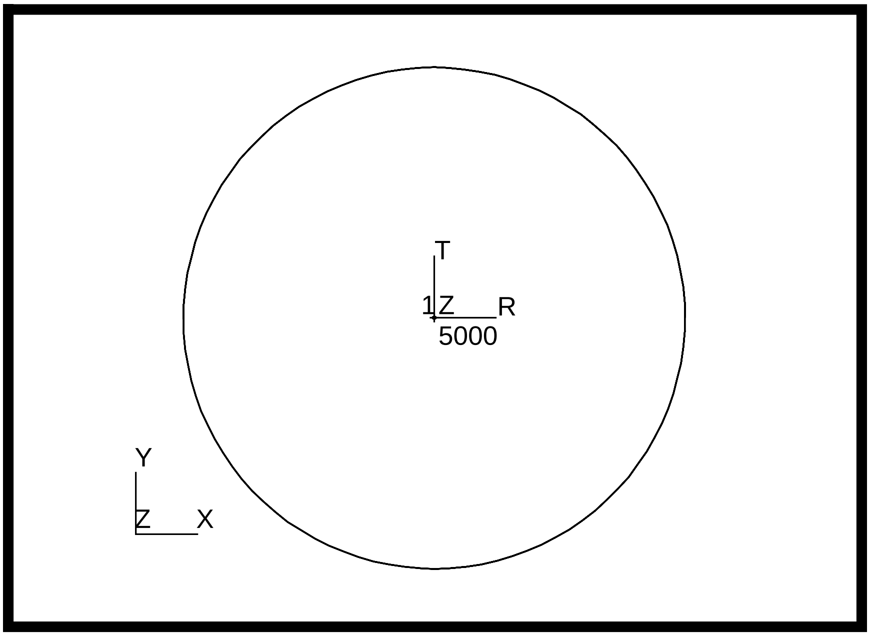
(Scroll Down)

Node: ■ **Label**

Apply

Cancel

The model should appear as follows:



8. Next, create an RBE3 element to distribute the load from the load application node to the cylinder walls.

Display/Plot/Erase...

Plot All Posted FEM

Erase All Geometry

OK

8a. Use your mouse to place the top row of elements and their top most nodes into a new group called **rbe3_region**. Make this group current and post only this group to the display.

Viewing/ Named View Options...

Select Named View: **nas120_side**

Close

Preferences/Picking...

Single Picking: ◆ **Entity**

Rectangle/Polygon Picking: ◆ **Enclose any portion of entity**

Close

Group/Create...

New Group Name: **rbe3_region**

■ **Make Current**

■ **Unpost All Other Groups**

Entity Selection:

Node 381:399 781:798 1180:1197 1579:1595 5000 Elm 360:399 703:720 1063:1080 1423:1440

Apply

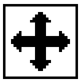
Cancel

Viewing/ Named View Options...

Select Named View: **nas120_iso**

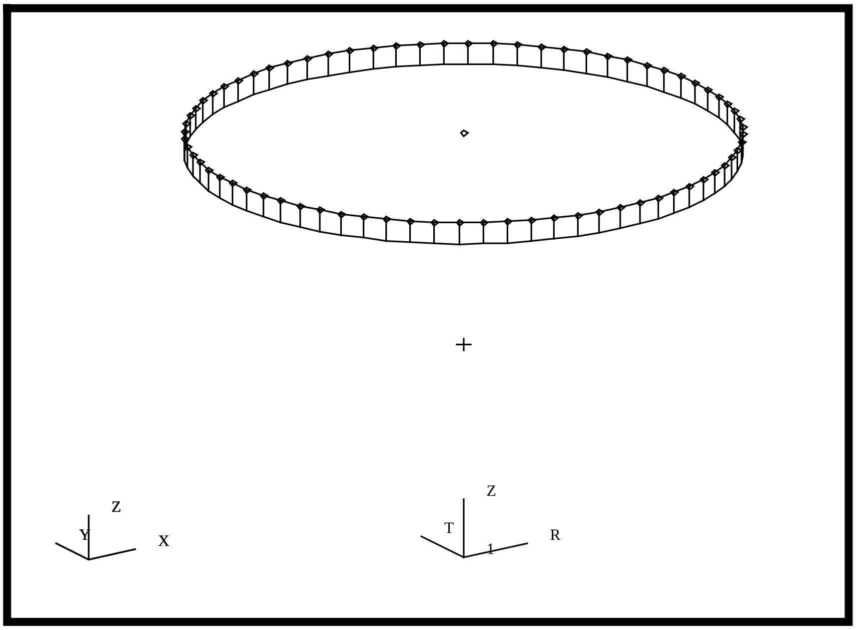
Close

To scale the view to fit the window, click the **Fit View** icon in the Main Form.



Fit View

8b. At this point, your model should appear as follows:



◆ **Finite Elements**

Action:	**Create**
Object:	**MPC**
Type:	**RBE3**

Define Terms ...

◆ **Create Dependent**

❑ **Auto Execute**

Node List:	**Node 5000**
DOFs	UX UY UZ RX RY RZ

Apply

◆ **Create Independent**

Node List:	**(Select remaining nodes)**
	See Note Below

NOTE: Select your independent nodes by using the polygon pick feature of your mouse (Ctrl-mouse click)

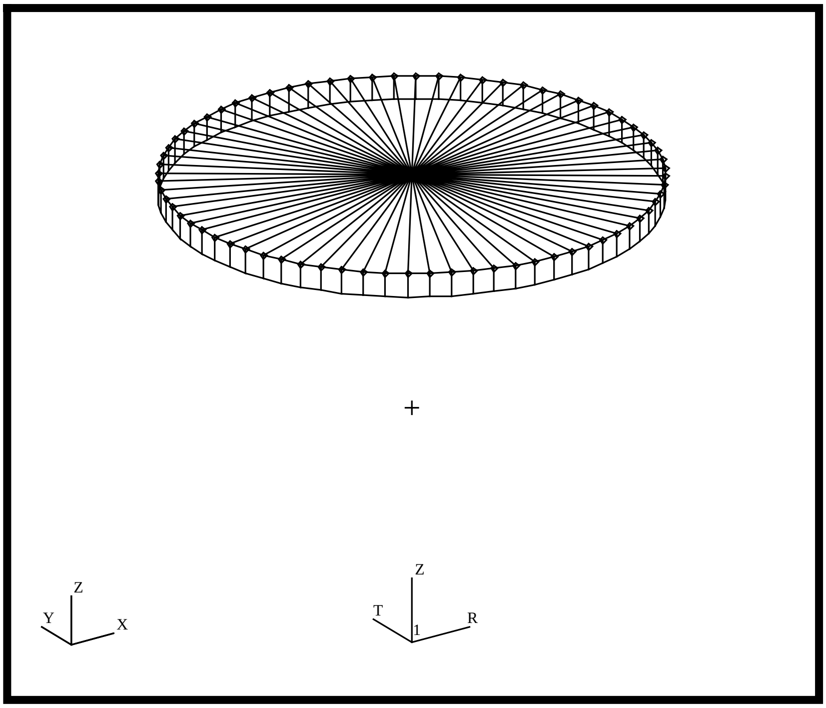
DOFs UX UY UZ

Apply

Cancel

Apply

This operation should yield the following display:



8c. Create a new group consists of all the geometry called **geometry_only** and make it current.

Group/Create...

New Group Name: **geometry_only**

■ **Make Current**

❐ **Unpost all other groups**

Group Contents: **Add All Geometry**

Apply

9. Define a material using the specified Modulus of Elasticity, Poisson ratio & density.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**aluminum**

Input Properties...

Constitutive Model:	**Linear Elastic**
Elastic Modulus =	**1.0e7**
Poisson Ratio =	**.3**

Apply

The *Current Constitutive Models* form should appear as below:

Linear Elastic - [,,,,] - [Active]

Cancel

10. Now, reference the material you just defined when you specify element properties for your analysis model. Remember to use the specified cylinder thickness.

◆ **Properties**

Action:	**Create**
Object:	**2 D**
Method:	**Shell**
Property Set Name:	**cylinder_prop**

Input Properties...

Material Name:	**m:aluminum**

Thickness: **0.03**

OK

Select Members: **Surface 1:4**

Add

Apply

11. Define the boundary constraints for the model. Refresh the display as needed before continuing.

11a. Create displacement constraints and apply it to the model. Recall that the **top** and **base** of the cylinder have different boundary conditions. We will apply these constraints to the geometry model instead of the analysis model. First, we define the base constraints:

◆ **Load/BCs**

Action: **Create**

Object: **Displacement**

Method: **Nodal**

New Set Name **base**

Input Data...

Translation < T1 T2 T3 > **< 0, 0, 0>**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Select Geometry Entities: **Surface 1:4.4**

Add

OK

Apply

11b. Next, we define the rim constraints:

◆ **Load/BCs**

Action: **Create**

Object: **Displacement**

Method: **Nodal**

New Set Name: **top**

Input Data...

Translation < T1 T2 T3 > **< 0, 0, >**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

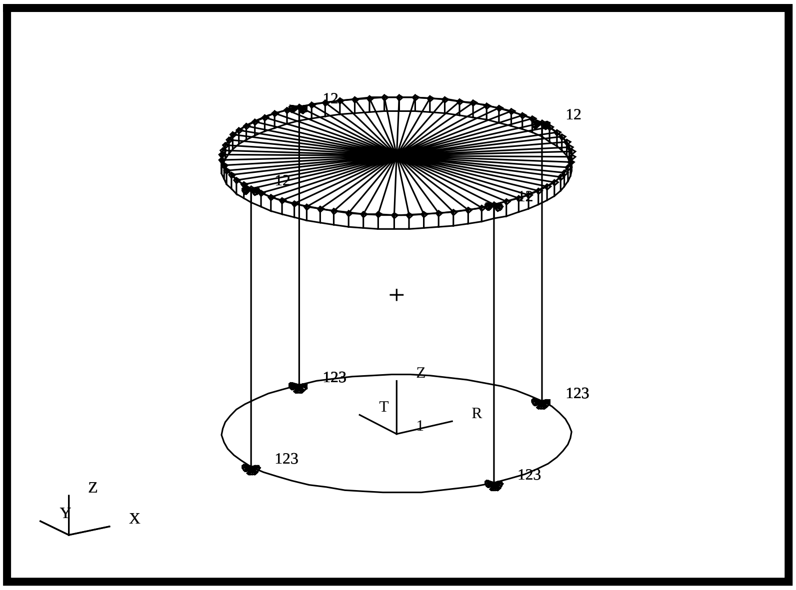
Select Geometry Entities: **Surface 1:4.2**

Add

OK

Apply

The displacement constraints should appear as follows:



Display/Entity Color/Label/Render...

(Scroll Down)

Node: ■ **Label**

Apply

Cancel

Display/Plot/Erase...

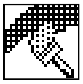
Erase All FEM

Selected Entities: **Node 5000**

Plot

OK

Refresh the display when needed using the brush icon on the Top Menu Bar.

 **Refresh Graphics**

12. Apply the compressive force along the local Z-axis of Node 5000.

◆ **Load/BCs**

Action:	**Create**
Object:	**Force**
Method:	**Nodal**
New Set Name:	**load**

Input Data...

Force <F1 F2 F3>	**< , , -10000>**

OK

Select Application Region...

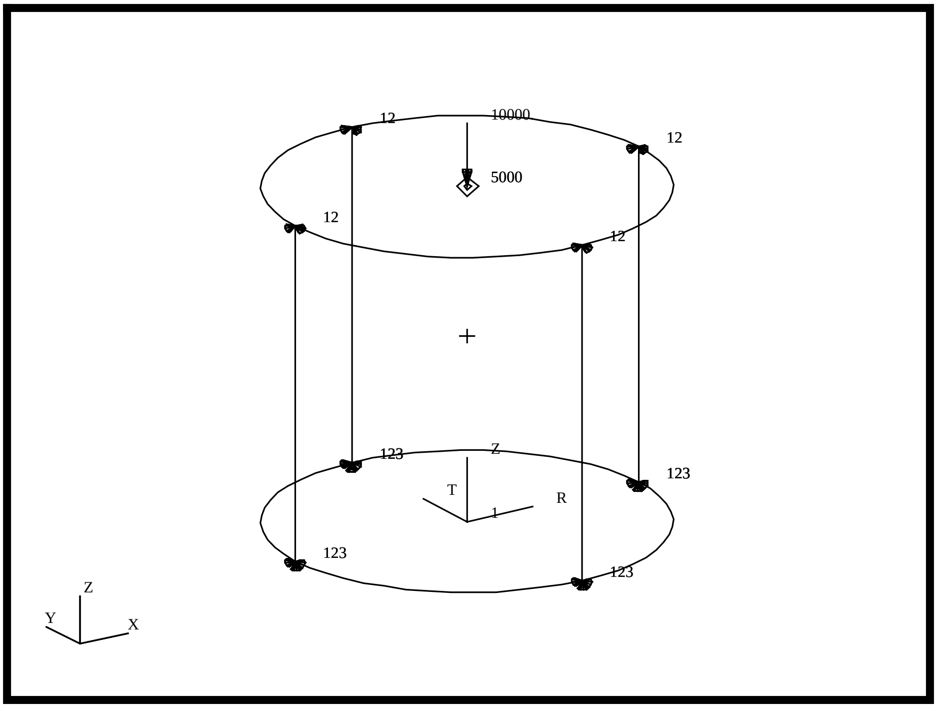
Geometry Filter:	◆ **FEM**
Select Nodes:	**Node 5000**

Add

OK

Apply

The load and boundary constraints should appear as follows:



13. For clarity, create a new group called **fem_only** that consists only of analysis model entities.

Group/Create...

New Group Name: **fem_only**

■ **Make Current**

■ **Unpost All Other Groups**

Group Contents: **Add All FEM**

Apply

Cancel

13a. Since the boundary & load sets were applied to the geometry model, the display of the load sets will only appear on the geometry model. This can be changed using **Display/Load/BC/Elem. Props...** option.

Display/Load/BC/Elem. Props...

■ **Show on FEM Only**

Apply

13b. Also, for clarity, you may wish to disable the display of vector values.

Display/Load/BC/Elem. Props...

❑ **Show LBC/El. Prop Vectors**

Apply
Cancel

Turn off all node labels as well.

Display/Finite Elements...

Node: ❑ **Label**

Apply
Cancel

13c. Display all of the loads & boundary markers on the analysis model.

◆ **Load/BCs**

Action: **Plot Markers**

Select all the Load/BC sets in the *Assigned Load/BCs Sets* box by highlighting all of them. And post the markers onto the current group.

Assigned Load/BCs Sets: **Displ_base**
Displ_top
Force_load

Select Groups: **fem_only**

Apply

14. Change the *Render Style* of your model from **Wireframe** to **Hidden Line**.

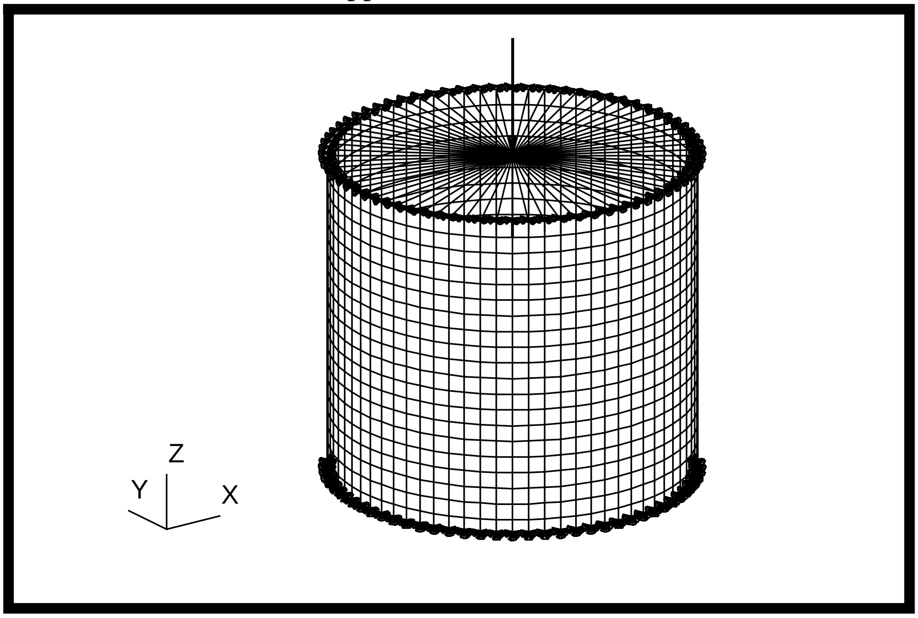
Display/Entity Color/Label/Render...

Render Style: **Hidden Line**

Apply

Cancel

Your model should appear as follows:



15. Now you are ready to generate an input file for 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:	**cylinder**

Translation Parameters...

OUTPUT2 Format:	**Binary**
MSC/NASTRAN Version:	*set accordingly, here it is **70***

OK

Solution Type...

Solution Type: ◆ **Buckling**

Solution Parameters...

■ **Database Run**

■ **Automatic Constraints**

Data Deck Echo: **Sorted**

Plate Rz Stiffness Factor = **10000.0**

Wt.- Mass Conversion = **0.00259** *(For English units)*

Eigenvalue Extraction

Number of Desired Roots = **2**

OK

OK

OK

Apply

An input file named **`cylinder.bdf`** will be generated. The process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green.

16. If all is well, you will then submit the input file to MSC/NASTRAN for analysis. To do this, find an available xterm window and at the prompt enter:

nastran cylinder.bdf scr=yes

Monitor the run using the UNIX **`ps`** command.

16a. When the run is completed, edit the **`cylinder.f06`** file and search for the word **FATAL.** If none exists, search for the word **WARNING**. Determine whether or not existing WARNING messages indicate modeling errors.

16b. While still editing **`cylinder.f06`**, search for the word:

R E A L (spaces are necessary)

What are the eigenvalues associated with the 2 buckling modes for our structure?

mode 1 = ____________

mode 2 = ____________

What is the critical buckling load for this structure?

P_{cr} = ____________

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

◆ **Analysis**

Action:	**Read Output2**
Object:	**Result Entities**
Method:	**Translate**

Select Results File...

Filter

Selected Results File:	*select the desired* ***.op2*** *file*

OK

Apply

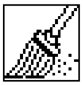
When the translation is complete and the Heartbeat turns green, bring up the *Results* form.

◆ **Results**

Action:	**Create**
Object:	**Quick Plot**

Choose the desired result case in the *Select Result Cases* list and select the result(s) in the *Select Fringe Result* list and/or in the *Select Deformation Result* list. And click **Apply** to view the result(s) in the viewport.

If you wish to reset your display graphics to the state it was in before you began post-processing your model, remember to select the broom icon.

 **Reset Graphics**

Quit MSC/PATRAN when you have completed this exercise.