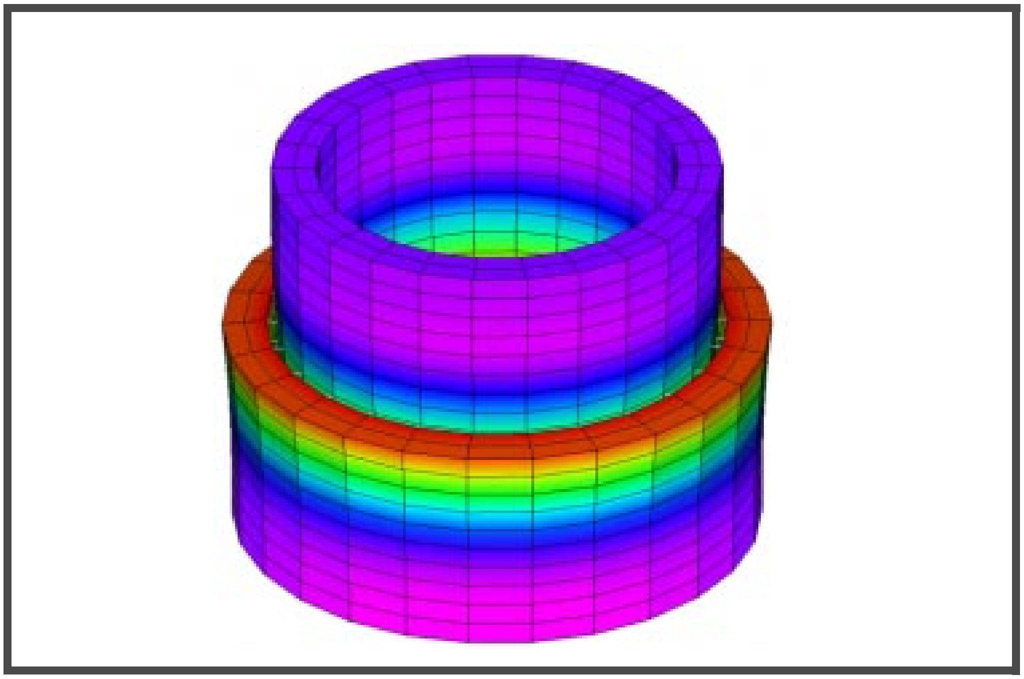
WORKSHOP 12a

Shrink Fit



Objectives

- Create a finite element model of the two steel rings with gap connections at the interface.
- Use groups and clipping planes to isolate the interface nodes.
- Apply loads, boundary conditions, and constraints for a non-linear static analysis.
- Define non-linear analysis parameters.
- Submit the job to MSC.Nastran for analysis.

Model Description:

The model below shows two steel rings with a 0.1 inch interference. Table 12a-1 shows the material properties for the model. Because of the nature of the geometry, all constraints and nodal coordinates will use a cylindrical coordinate system. MSC.Nastran for Windows supports a basic cylindrical coordinate system, which will be used in this exercise. .

This exercise covers creating the finite element model, defining loads and boundary conditions, and setting up non-linear analysis parameters. The second part of the exercise will study the effect of reversing the orientation of the gaps. The third part of the exercise will consider reversing the open stiffness, (K_a), and closed stiffness, (K_b), of the gap elements. Because the gaps properties have been reversed, the initial opening, (U_o), will have to be redefined. The fourth part will take the same model and reverse the gap connectivity. The last part of the exercise will add a rigid connection to the top edge of the model and apply a transverse load to the model.

For convergence purposes, the top edge of the model will be constrained in TR and TZ, and a small enforced displacement of 0.000001 will be applied in the negative Z direction. There will also be a small nodal load of 1×10^{-14} lb. applied on an inconspicuous corner of the model to help with incrementing the load. Finally, nodes along the bottom edge of the model will be pinned.

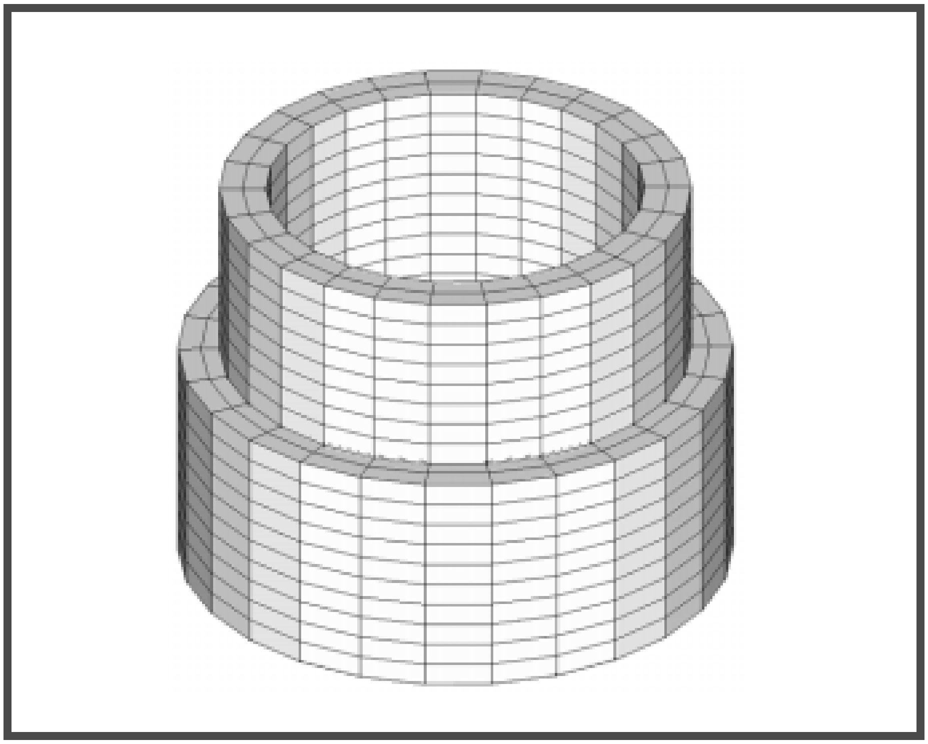
Table 12a.1 - Material Properties

Youngs Modulus:	**30.0 E 6**
Poisson's Ratio:	**0.3**

Table 12a.2 - Gap Element Properties

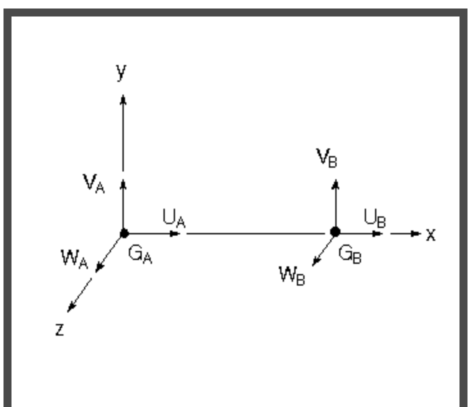
Initial Gap Opening (U_o):	**0.1**
Closed Stiffness (K_a):	**1.0 E-6**
Open Stiffness (K_b):	**1.0 E 8**

Figure 12a.1 - The model



The interface between the rings will be connected with gap elements. Gap elements contain a closed and open stiffness, and contributes that stiffness depending on the relative location of its grid points. The gap coordinate system is shown in Figure 12a.2. Because the open stiffness, or K_b, is much higher than the closed stiffness, or K_a, the gaps will tend to pull the interface nodes to a common location of not interference, as shown in the hand calculation.

Figure 12a.2 - Interface Nodes



Hand Calculation:

When U_a-U_b>Uo, the gap elements exhibits the closed stiffness (K_a). When U_a-U_b<Uo, the gaps exhibit the open stiffness (K_b). The gap elements in this exercise are explained using 3 cases in a hand calculation.

Initial state:

@ $U_a = U_b = 0$, parts interfere.
$U_a - U_b = 0$, which is $< U_0$: K_b kicks in, pull parts to common point of non-interference.

@ $U_a = 0.02$, $U_b = -0.02$, the parts are still pushing into each other.
$U_a - U_b = -0.02 - (-0.02)=0.04$, which is $< U_0$, $\therefore$ Same condition applies, K_b kicks in and separates the parts.

@ $U_a =0.051$, $U_b = -0.051$, the parts no longer interfere with each other
$U_a - U_b = 0.051 - (-0.051) = 0.102$, which is $> U_0$. K_a kicks in which is less stiff than the steel ring. The parts are now free from interference.

Suggested Exercise Steps:

1. Create a 2D mesh representing a cross-section of the model.
2. Revolve the mesh to create a solid model.
3. Define a gap property.
4. Use groups and clipping planes to isolate the interface nodes.
5. Create gap elements between the interface nodes.
6. Apply loads, boundary conditions, and non-linear analysis parameters.

Exercise Procedure:

1. Start up MSC.Nastran for Windows 4.0.2 and create a new model.

Double click on the icon labeled MSC.Nastran for Windows V4.0.2.

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

Open Model File: **New Model**

2. Create a material called **steel**.

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

Model/Material...

Title: **steel**

Youngs Modulus: **30.0E6**

Poisson's Ratio: **.3**

OK

Cancel

3. Create the three properties needed for this exercise.

3a. Define a 2D property for the 2D elements. Because the 2D elements are made only to create a revolved mesh, the properties of these elements are trivial. Because of this, plot-only elements will be used.

Model/Property...

Title: **dummy**

Elem/Property Type...

Change the property type to Plot Only.

Plane Elements: ● **Plot Only**

NOTE: Select **Plot Only** for **Plane Elements**, which is different from **Plot Only** for **Line Elements**.

OK

OK

At the prompt, “This Property Type is not usually required. OK to Create?“ choose **Yes**.

Yes

3b. Define a 3D property for the solids.

Title: **solid**

Elem/Property Type...

Volume Elements: ● **Solids**

OK

To select the material, click on the list icon and select **steel**.

Material: **1.steel**

OK

3c. Define a gap property for the gaps.

Title: **gap**

Elem/Property Type...

Line Elements: ● **Gap**

OK

Input the following Property Values.

Initial Gap: **0.1**

Compression Stiffness: **1.0E-6**

Tension Stiffness: **1.0E8**

OK

Cancel

4. Rotate the view from the XY plane to the XZ plane.

View/Rotate... **(F8)**

ZX Front
OK

5. Because geometry is not created in this model, the workplane will not be useful. Enter the **Workplane Management** menu and turn off the workplane.

Tools/Workplane... **(F2)**

☐ **Draw Workplane**

Done

To update this change, regenerate the display.

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

6. Use groups to organize the model. Create a group for the elements representing the upper ring.

Group/Set...

ID:	**1**
Title:	**Upper**

OK

6a. Turn on the automatic add feature to add all created entities into the active group.

Group/Operations/Automatic Add...

● **Active**

OK

7. Create a 2D representation for the cross section of the upper ring, which will be revolved to create a solid mesh.

Mesh/Between...

Property: **1..dummy**

Node Param...

Output Coordinate System: **1..Basic Cylindrical**

OK

	Dir 1:	*Dir 2:*
#Nodes:	**3**	**11**

OK

	X:	*Y:*	*Z:*
Corner 1:	**4.1**	**0**	**4**

OK

	X:	*Y:*	*Z:*
Corner 2:	**5.1**	**0**	**4**

OK

	X:	*Y:*	*Z:*
Corner 3:	**5.1**	**0**	**9**

OK

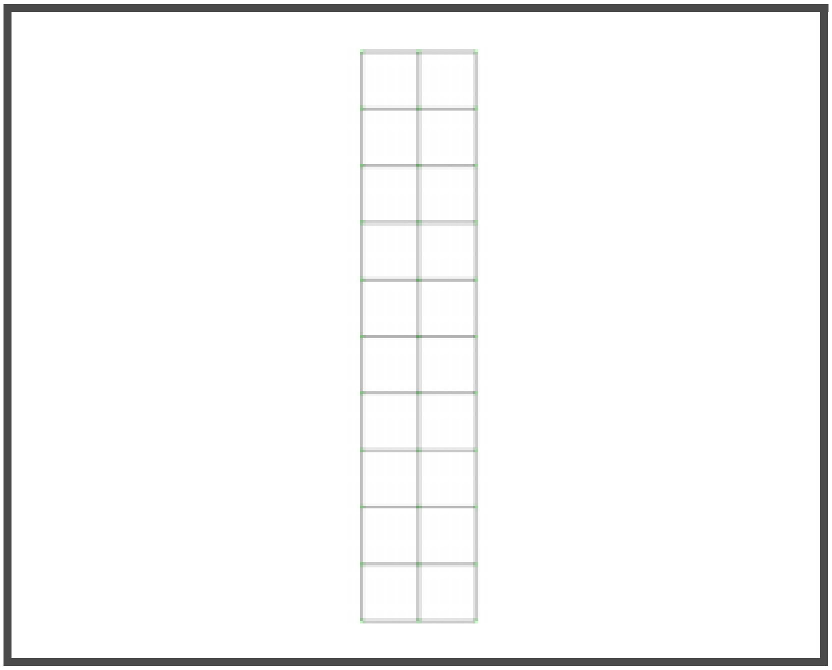
	X:	*Y:*	*Z:*
Corner 4:	**4.1**	**0**	**9**

OK

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale **(Ctrl-A)**

Figure 12a.3 - Upper ring cross section.



The surface mesh should resemble Figure 12a.3.

Modify the definition coordinate system of the nodes to reference the basic cylindrical coordinate system.

Modify/Update Other/Node Definition CSys...

Group:	**1..Upper**
OK	
Entity ID:	**1..Basic Cylindrical**
OK	

Revolve the mesh to create a solid model.

Mesh/Revolve/Element...

Group:	**1..Upper**
OK	

Property:	**2..Solid**
Elements along Length	**26**

☒ **Delete Original Elements**

OK

Methods^	**Global Axis**
Coord Sys:	**1..Basic Cylindrical**
Direction:	● **Positive**

● **Z Axis**

	R:	*T:*	*Z:*
Base:	**0**	**0**	**0**

OK

Rotation Angle:	**360**

OK

At the prompt, "OK to Delete 20 Selected Elements" choose **Yes**.

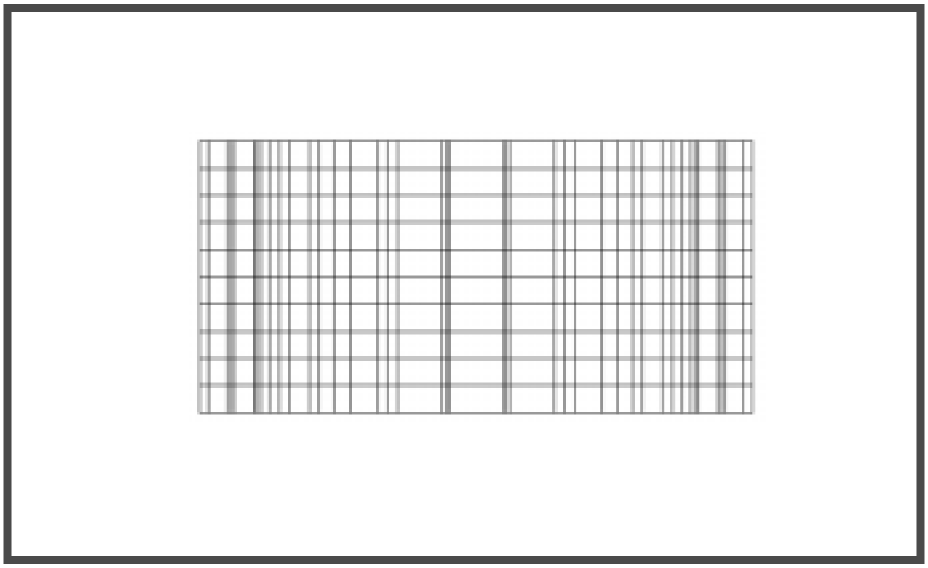
Yes

Fit the display onto the screen, using the Autoscale feature.

View/Autoscale **(Ctrl-A)**

The solid upper ring should look like Figure 12a.4.

Figure 12a.4 - Group 1, a solid mesh of the upper ring.



7a. Generate a mesh for the lower solid.

Group/Set...

ID: **2**

Title: **Lower**

OK

Mesh/Between...

Coord Sys: **0..Basic Rectangular**

Property: **1..dummy**

Node Param...

Output Coordinate System... **1..Basic Cylindrical**

OK

	Dir 1:	*Dir 2:*
#Nodes:	**3**	**11**

OK

	X:	*Y:*	*Z:*
Corner 1:	**5**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 2:	**6**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 3:	**6**	**0**	**5**

OK

	X:	*Y:*	*Z:*
Corner 4:	**5**	**0**	**5**

OK

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale **(Ctrl-A)**

Modify the definition coordinate system of the nodes to reference the basic cyclindrical coordinate system.

Modify/Update Other/Node Definition CSys...

Group: **2..Lower**

OK

Entity ID: **1..Basic Cylindrical**

OK

Revolve the 2D mesh into a solid model of the lower ring.

Mesh/Revolve/Element...

Group:	**2..Lower**
OK	
Property:	**2..Solid**
Element along Length	**26**

☒ **Delete Original Elements**

OK

Coord Sys:	**1..Basic Cylindrical**
Methods^	**Global Axis**
Direction:	● **Positive**

● **Z Axis**

	R:	*T:*	*Z:*
Base:	**0**	**0**	**0**

OK

Rotation Angle:	**360**

OK

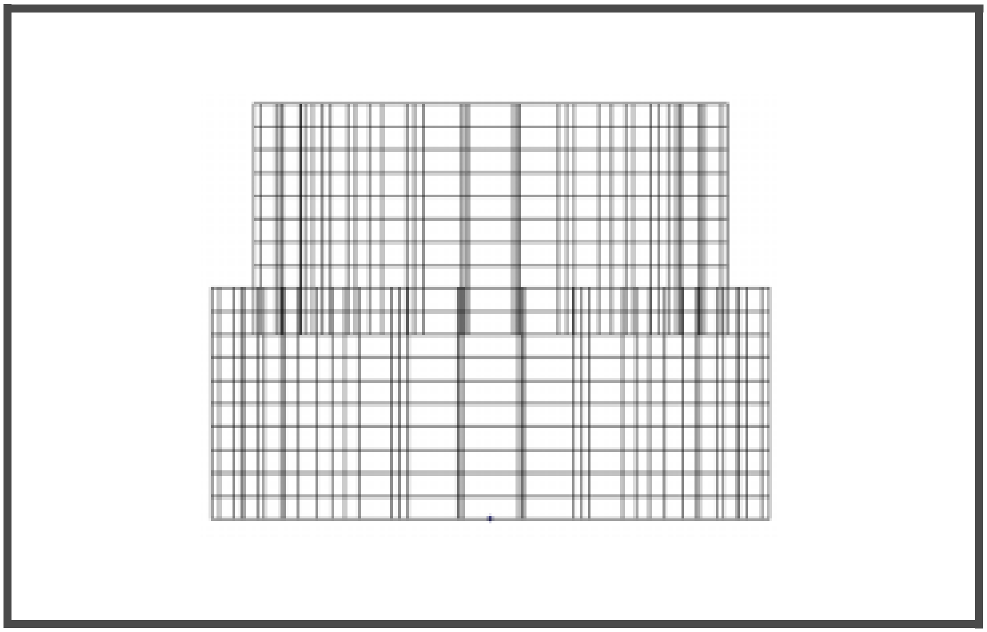
At the prompt, “OK to Delete 20 Selected Elements” choose **Yes**.

Yes

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale **(Ctrl-A)**

Figure 12a.5 - The complete solid model is shown below



8. Because the mesh was revolved a full revolution. The nodes that lie on the positive half of the XZ plane will not be connected together. To fix this, merge all coincident nodes.

Tools/Check/Coincident Nodes...

Select All

OK

At the prompt, "OK to Specify Additional Range of Nodes to Merge?" choose **NO**.

NO

☒ **Merge Coincident Entities...**

OK

NOTE: Since the interface nodes are a distance of .1 apart, the merging tolerance is only small enough to merge the nodes on the XZ plane. The nodes between the two solids will not be merged.

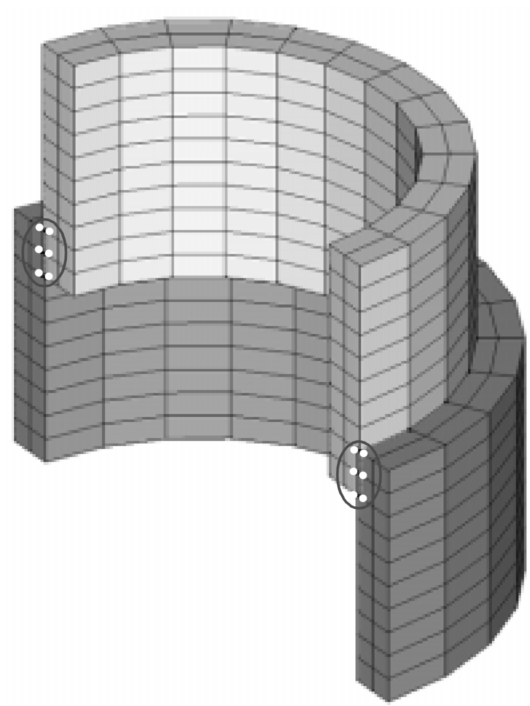
9. To make the next steps easier, change the screen to show only the active group. Right click anywhere on an empty area of the screen.

Model Data...

Group: ● **Active**

OK

Figure 12a.6 - Half view of model with interfaces nodes highlighted



Note: The nodes on the upper ring which come into contact with the lower ring can be identified with the coordinates 4 ≤ Z ≤ 5, and R=5.1. The interface nodes on the lower ring can be identified with cylindrical coordinates, 4 ≤ Z ≤ 5, and R = 5.0. Because there is no geometry, and the interface nodes are beneath layers of elements, the easiest way to isolate these nodes is using groups and clipping planes. A clipping plane uses coordinates or a 2D plane to cut, or clip out a part of the model. Anything that isn't clipped is included in the group. In this case, each group will use two clipping planes, one at z=4, another and z=5, and a third clipping rule where R=5.0 or 5.1.

10a. Isolate the interface nodes.

Group/Set...

ID:	3
Title:	inner interface nodes
OK	

Group/Clipping/Plane...

Clip:	● **Positive Side**
Clipping Plane:	● **1**
OK	

Coord Sys:	**1..Basic Cylindrical**
Method^	**Global Plane**
Direction:	● **Positive**
	● **XY Plane**
Z:	5
OK	

Group/Clipping/Plane...

Clip:	● **Positive Side**
Clipping Plane:	● **2**
OK	

CSys:	**1..Basic Cylindrical**
Method^	**Global Plane**
Direction:	● **Negative**
	● **XY Plane**
Z:	4
OK	

Group/Clipping/Coordinate...

● **Outside**

Coord Sys: **1..Basic Cylindrical**

Coordinate Value: ● **R**

Minimum: **4.99**

Maximum: **5.01**

OK

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale **(Ctrl-A)**

NOTE: Although the display may show elements on the top and bottom ring, only the interface nodes of the upper ring exist in this group.

10b. Create a group for the outer interface nodes.

Group/Set...

ID: **4**

Title: **outer interface nodes**

OK

Group/Clipping/Plane...

Clip: ● **Positive Side**

Clipping Plane: ● **1**

OK

CSys: **1..Basic Cylindrical**

Method^ **Global Plane**

Direction: ● **Positive**

● **XY Plane**

Z: **5**

OK

Group/Clipping/Plane...

Clip: ● **Positive Side**

Clipping Plane: ● **2**

OK

Coord Sys: **1..Basic Cylindrical**

Method^ **Global Plane**

Direction: ● **Negative**

● **XY Plane**

Z: **4**

OK

Group/Clipping/Coordinate...

● **Outside**

Coord Sys: **1..Basic Cylindrical**

Coordinate Value: ● **R**

Minimum: **5.09**

Maximum: **5.11**

OK

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale **(Ctrl-A)**

NOTE: Although the display may show elements on the top and bottom ring, only the interface nodes of the lower ring exist in this group.

11. Reset the view to look at the entire model. Right click anywhere on an empty area of the screen.

Model Data...

Group: ● **None**

OK

12. Create gap element between the interface nodes.

Mesh/Connection/Closest Link...

Group: **3..inner interface nodes**

OK

Group: **4..outer interface nodes**

OK

Connection Type: ● **Line Elements**

Property: **3..Gap**

OK

Coord Sys: **1..Basic Cylindrical**

Method^ **Global Plane**

● **Z Axis**

	R:	*T:*	*Z:*
Base:	**0**	**0**	**0**

OK

NOTE: This step create gap element from grid A (inner grids) to grid B (outer grids).

13. Create the model constraints.

Model/Constraint/Set...

Title: **constraint**

OK

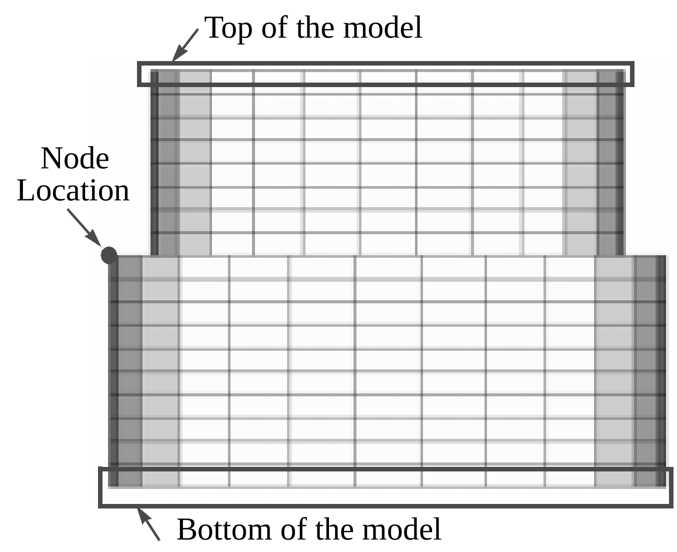
Pin the nodes at the bottom of the model.

Model/Constraint/Nodal...

Select the base nodes using the box method, as shown in Figure 12a.7.

OK

Figure 12a.7 - Application regions for constraints in the model.



On the *DOF* box, select these boxes.

CSys: **1..Basic Cylindrical**

Pinned

OK

Next, select the top of the model using the box method, as shown in Figure 12a.7.

OK

On the *DOF* box, select these boxes.

Coord Sys:	**1..Basic Cylindrical**

☒ **TT** ☒ **TZ**

OK

Cancel

14. Apply loads to the model.

Model/Load/Set...

Title:	**loads**

OK

Apply a small enforced displacement to the top of the top ring. This help with convergence for the non-linear analysis.

Model/Load/Nodal...

Select the top of the model, as shown in Figure 12a.7.

OK

Select the following options.

(highlight Displacement)	**Displacement**
Coord Sys:	**1..Basic Cylindrical**
TZ:	**-1.0E-6**

OK

Apply a small symmetrical load at top part of the model, as shown in Figure 12a.7.

OK

Select the following options.

(highlight Displacement)	**Force**
Coord Sys:	**1..Basic Cylindrical**

FZ: ☒ **1.0E-14**

OK

Cancel

15. Define the options for a nonlinear analysis.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Defaults...

OK

16. Submit the job for analysis.

File/Export/Analysis Model...

Type: **10..Nonlinear Static**

OK

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

File name: **prob12a**

Write

☒ **Run Analysis**

OK

When asked to save the model, choose **Yes**.

Yes

File name: **prob12a**

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

17. Review the results of the analysis.

View/Select **(F5)**

Under the *Deformed Style* window, make the following selection:

Model Style:

● **Quick Hidden Line**

☒ **Render**

Deformed Style: ● **Deform**

Deformed and Contour Data...

Output Set: **1..Case 1 Time 1**

Deformation: **1.. Total Translation**

OK

OK

18. Switch to a Dimetric view and a scale factor of 10X the actual deformation.

View/Rotate... **(F8)**

Dimetric

OK

View/Options... **(F6)**

● **PostProcessing**

Options: **Deformed Style**

% of Model (Actual) ☐

Scale %: **0.75**

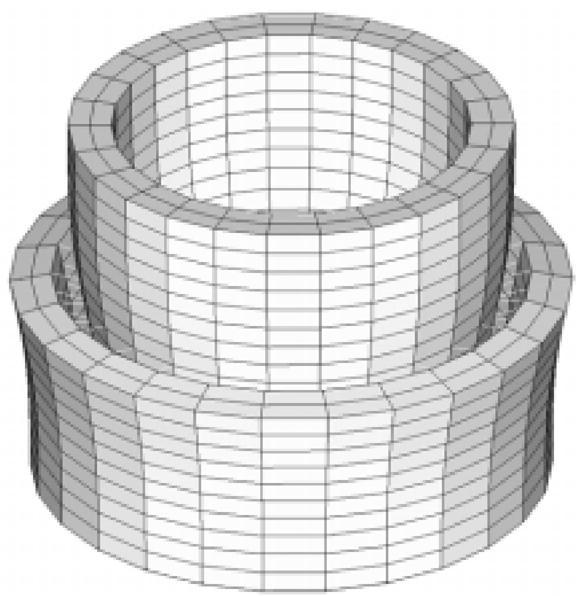
Scale Act: | **10**

OK

19. Save the model.

File/Save... **(F4)**

Figure 12a.8 - Model with displacement



What is the maximum deformation?

Value = ___________

Displacement	**0.0685**