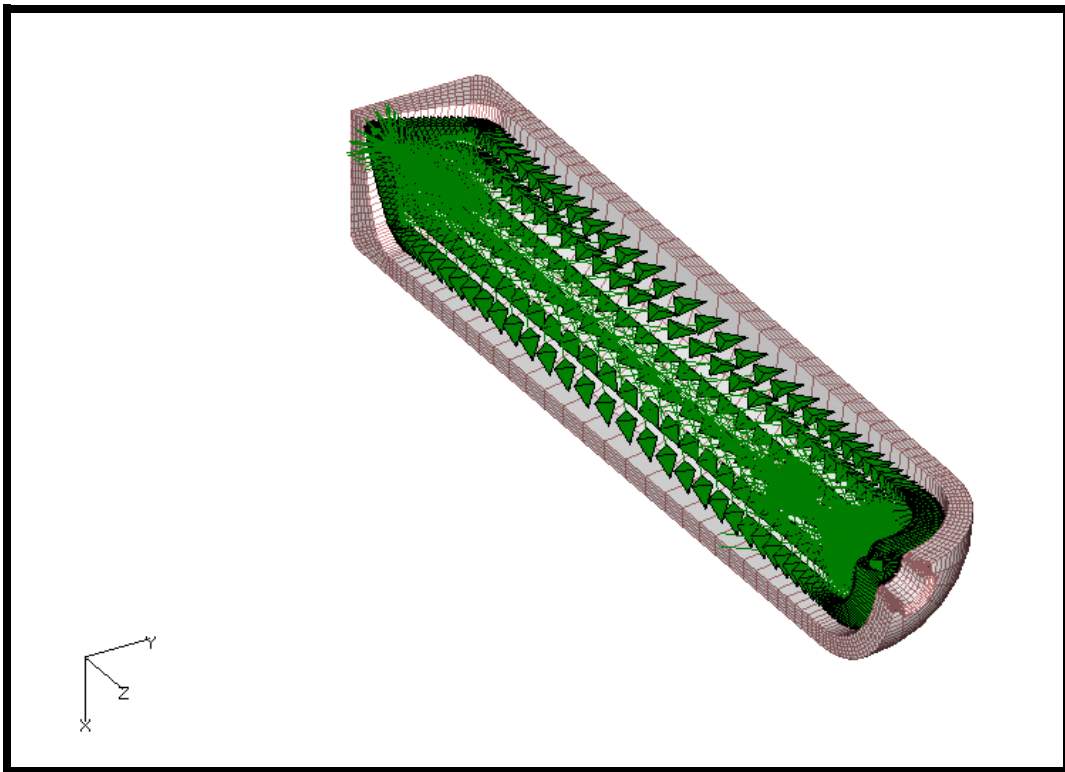

WORKSHOP 21a

Scuba Tank: Solid Mesh



Objectives:

- Create solid tank model.
- Evaluate Results.
- Remesh.
- Evaluate Results.



Suggested Exercise Steps:

- Import the file “tank_outline.NEU”
- Complete the quarter tank geometric model.
- Create the material and property.
- Save these data in a neutral file for later use.
- Solid mesh the model.
- Apply loads and constraints.
- Run analysis and evaluate results.
- Create a new model with a finer mesh.
- Run analysis and evaluate results.

Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0 and begin to create a new model.

Double click on the icon labeled **MSC.Nastran for Windows V4.0**.

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

Open Model File:

New Model

Tools/Advanced Geometry...

Geometry Engine:

Standard

OK

2. The cross-section geometry has already been created and stored in a FEMAP Neutral file.

File/Import/FEMAP Neutral...

Change directory to **c:\training\examples**.

File name:

tank_outline.NEU

Open

OK

The workplane will not be necessary.

Right Click on screen to invoke the pop up menu.

Workplane...

Uncheck **Draw Workplane**.

Draw Workplane

Done

3. Rotate the model.

View/Rotate... <F8>

X:

0

Y:

-90

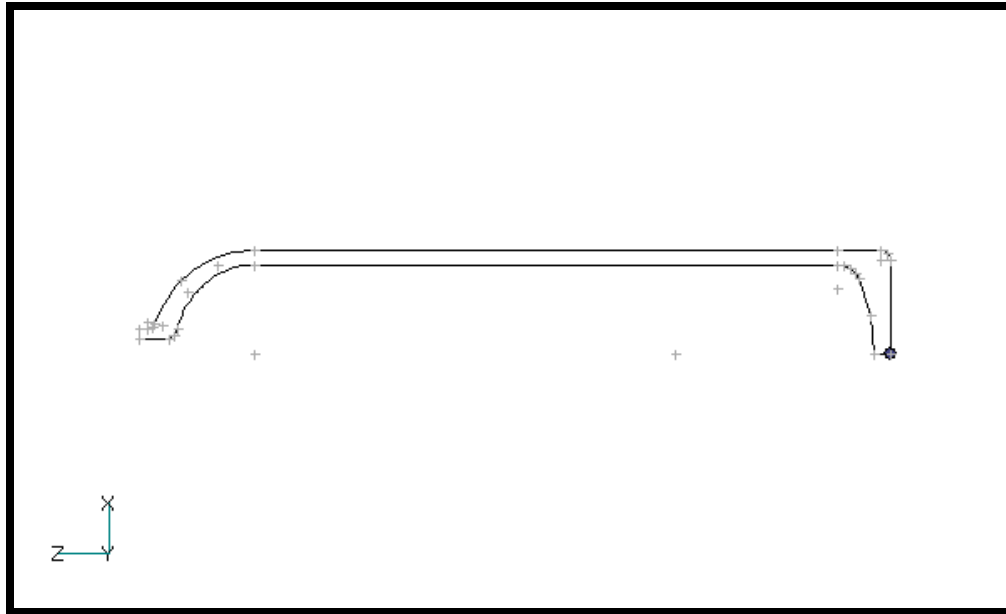
Z:

90

OK

View/Autoscale <Ctrl+A>

Figure 21a.2



4. Turn on Curve IDs.

View/Options... <F6>

Options:

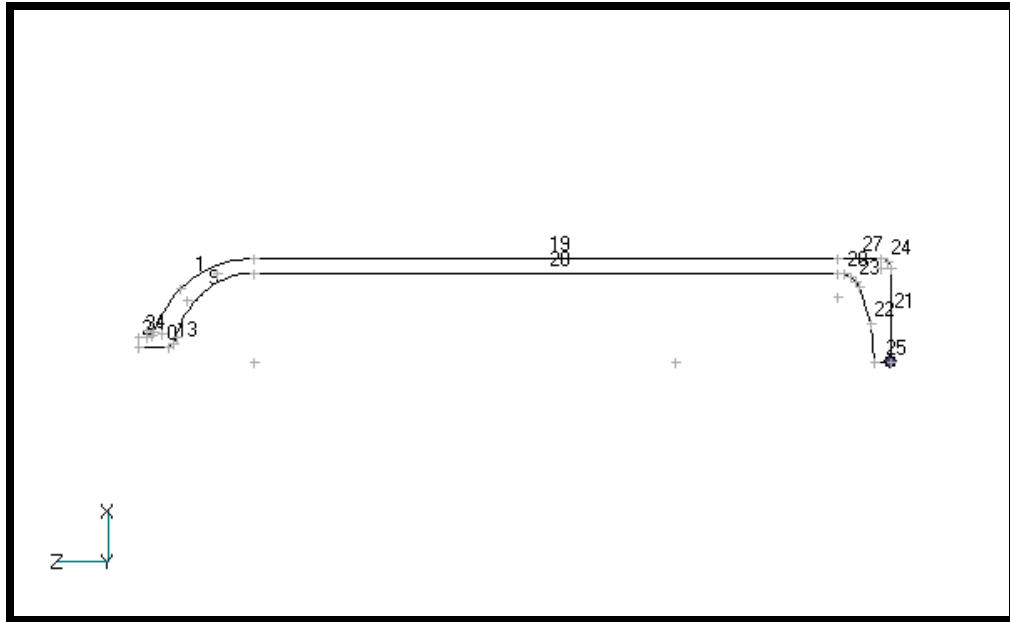
Curve

Label Mode:

1..ID

OK

Figure 21a.3



5. Create several ruled surfaces using the following curve pairs:

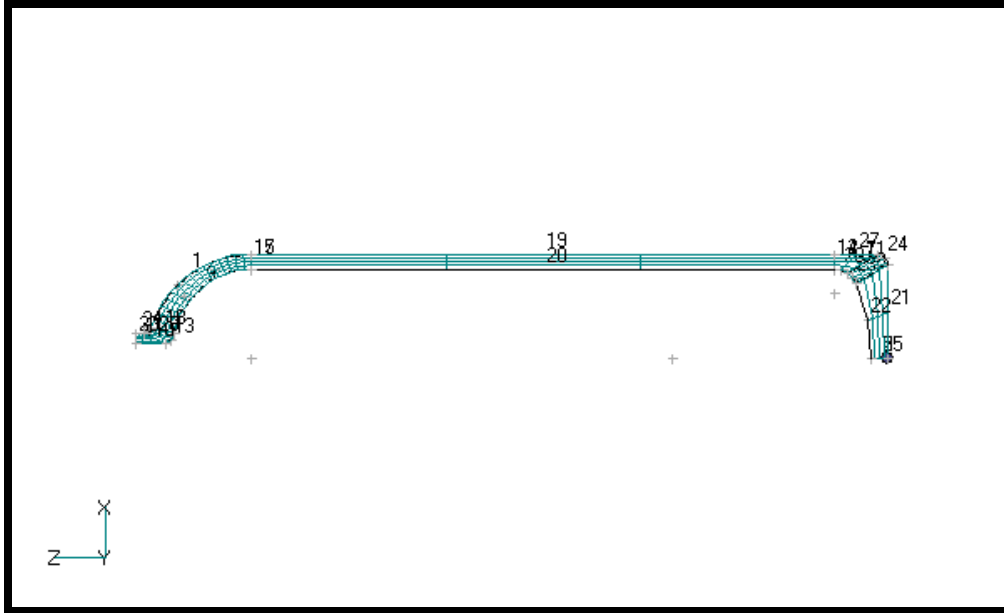
Geometry/Surface/Ruled...

From Curve: To Curve:

Continue for the following combinations:

<i>From Curve:</i>	<i>To Curve:</i>	
<input type="text" value="24"/>	<input type="text" value="23"/>	<input type="text" value="OK"/>
<input type="text" value="27"/>	<input type="text" value="28"/>	<input type="text" value="OK"/>
<input type="text" value="19"/>	<input type="text" value="20"/>	<input type="text" value="OK"/>
<input type="text" value="1"/>	<input type="text" value="9"/>	<input type="text" value="OK"/>
<input type="text" value="4"/>	<input type="text" value="13"/>	<input type="text" value="OK"/>
<input type="text" value="3"/>	<input type="text" value="10"/>	<input type="text" value="OK"/>

Figure 21a.4



6. Merge coincident curves.

Tools/Check/Coincident Curves...

Select All

OK

Options:

Merge Coincident Entities

OK

View/Redraw... <Ctrl+D>

7. Revolve these surfaces to generate volume geometry.

Geometry/Volume/Revolve...

Select All

OK

Methods ^

Global Axis

Direction:

Positive

Z Axis

OK

Rotation Angle:

90

OK

Cancel

8. Reposition volume.

View/Rotate... <F8>

X:

30

Y:

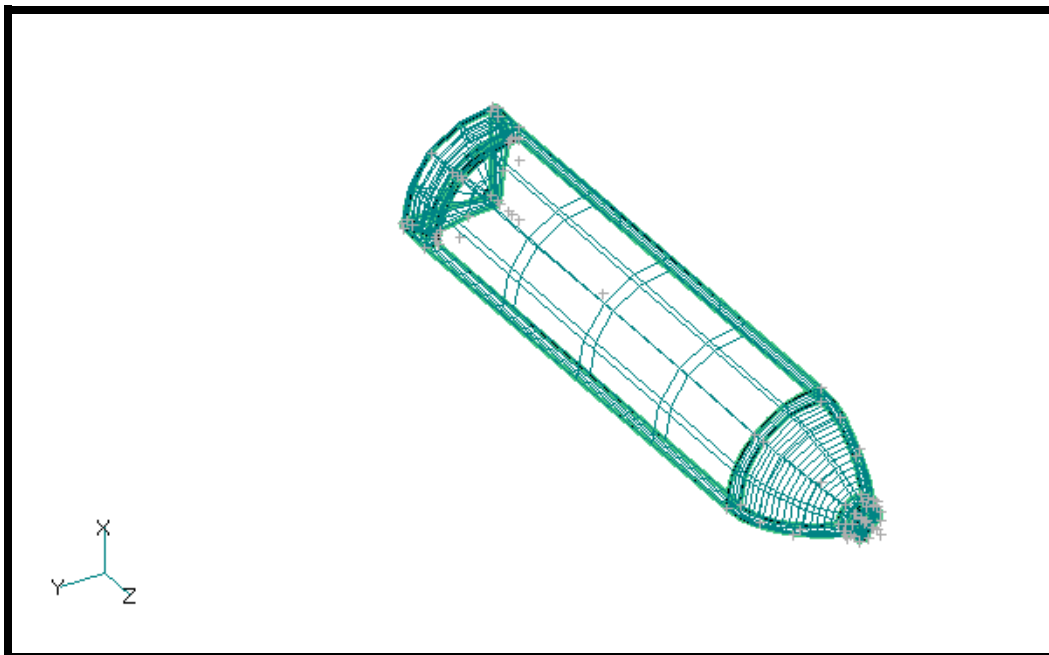
30

Z:

90

OK

Figure 21a.5



The geometry is now complete.

9. Define a material for the model.

Model/Material...

Load...

Library Entry:

17-4PH Stainless H1025

OK

OK

Cancel

10. Define the model property.

Model/Property...

Elem/Property Type...

Volume Elements:

Solid

OK

Title:

Solid_Stainless

Material:

1..17-4PH Stainless H1025

Material Axes:

1..Basic Cylindrical

Align to CSys

OK

Cancel

11. Apply mesh control.

Mesh/Mesh Control/Size Along Curve...

Select all 14 curves radial curves (Curves 60, 59, 66, 61, 58, 65, 53, 63, 62, 54, 64, 57, 56, 55)

OK

Number of Elements:

6

OK

Cancel

View/Rotate... <F8>

X:

30

Y:

30

Z:

-90

OK

12. Turn off curve labels.

View/Options...

Category:

Labels, Entities and Color

Options:

Curve

Label Mode:

0..No Labels

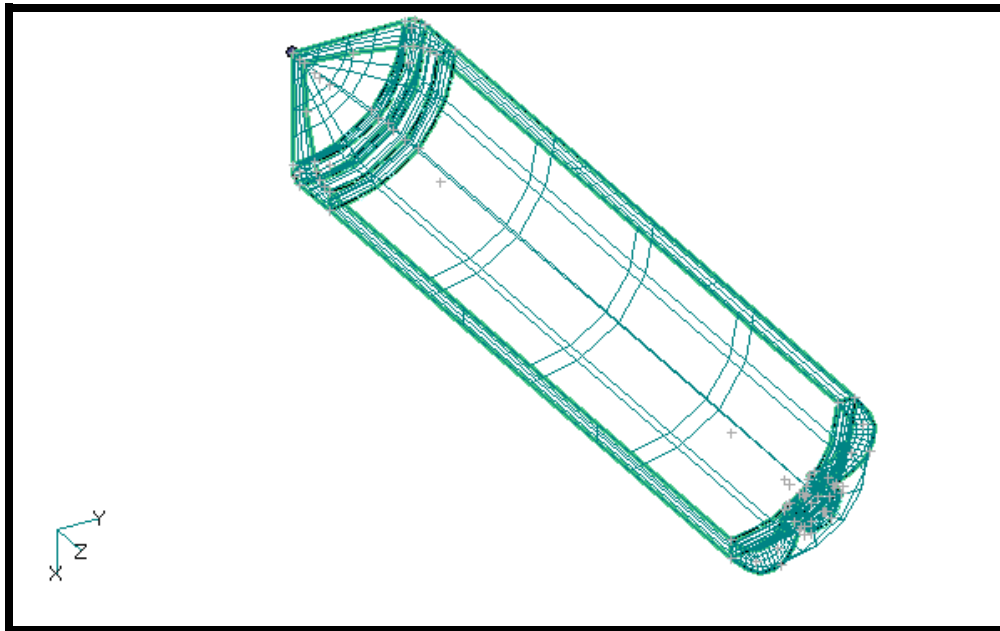
OK

View/Magnify...

Fill View

OK

Figure 21a.6



13. Save this model to a neutral file. This geometry will be used later to generate a finer mesh model to compare results.

File/Export/FEMAP Neutral...

File Name:

tank_quarter.NEU

Write

OK

14. Mesh the geometry.

Mesh/Geometry/Volume...

Select All

OK

Property:

1..Solid Stainless

Node Param...

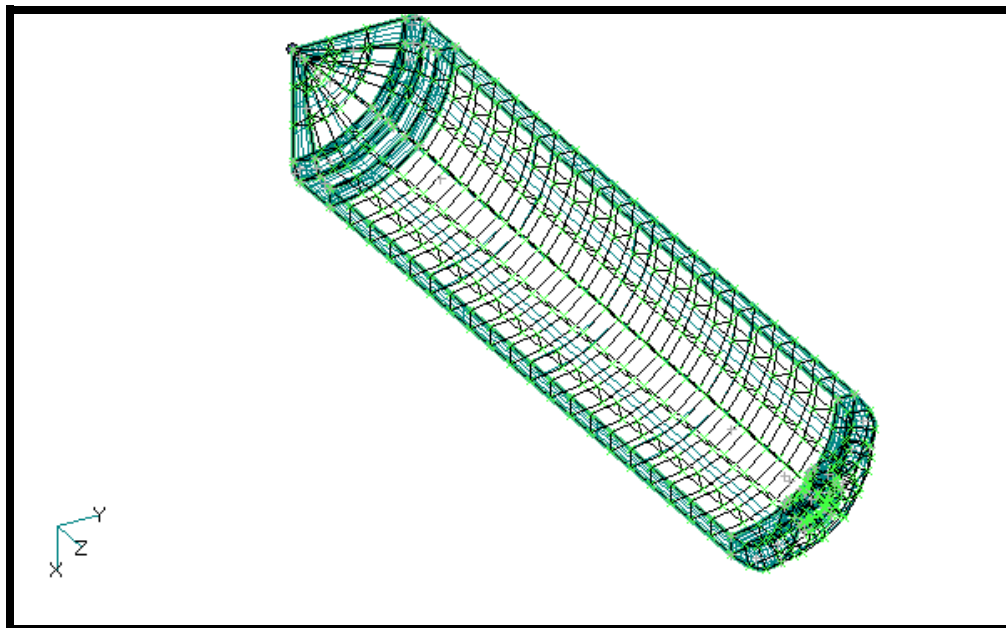
Output Coordinate System:

1..Basic Cylindrical

OK

OK

Figure 21a.7



15. Equivalence the finite element mesh.

Tools/Check/Coincident Nodes...

Select All

OK

No

Options:

Merge Coincident Entities

OK

Click the *Quick Options* icon and turn off the geometry.

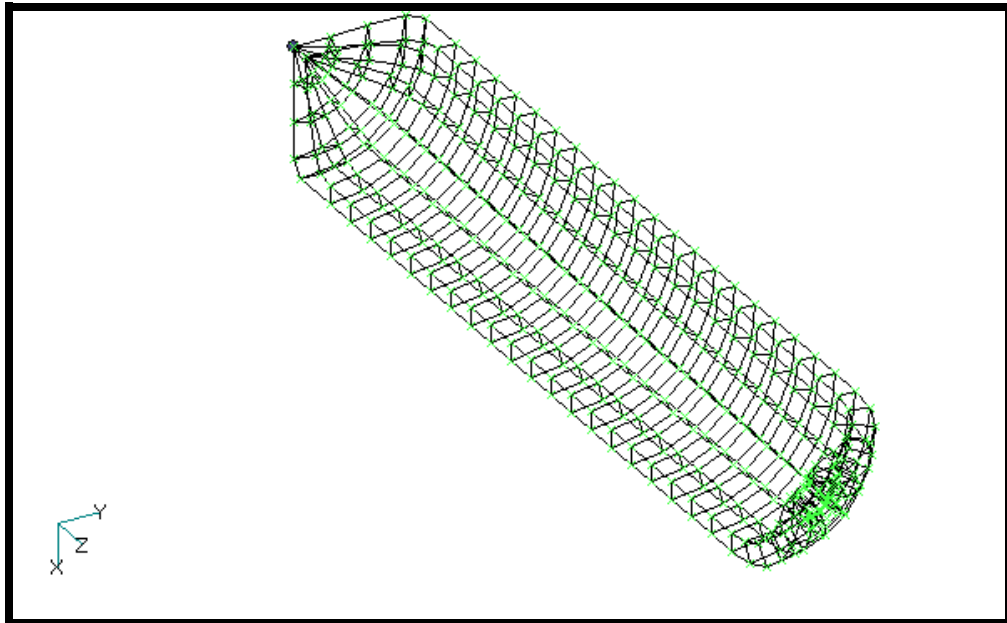


Quick Options

Geometry Off

Done

Figure 21a.8



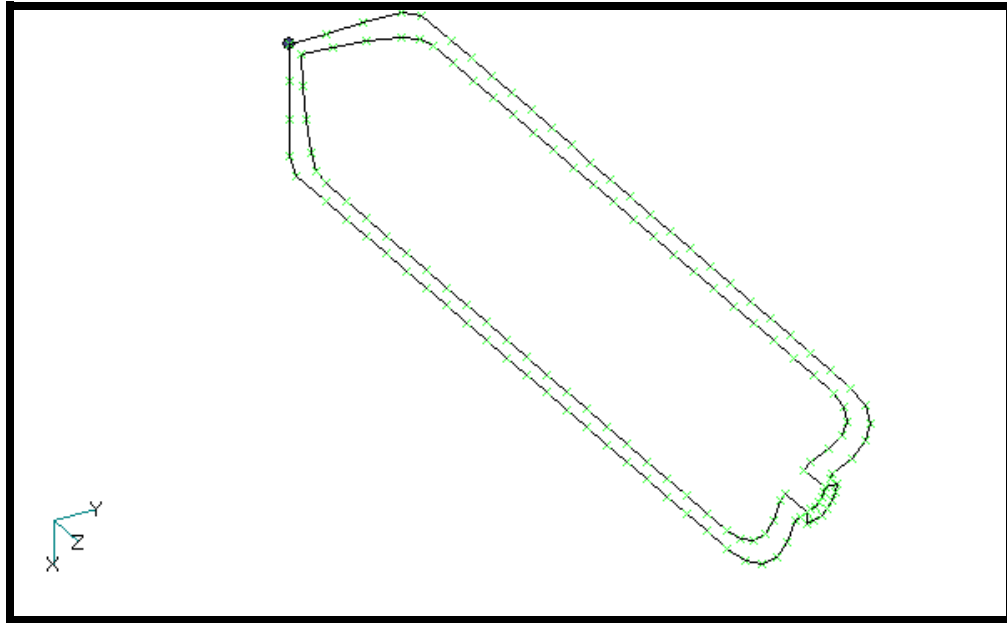
View/Select... <F5>

Model Style:

● **Free Edge**

OK

Figure 21a.9



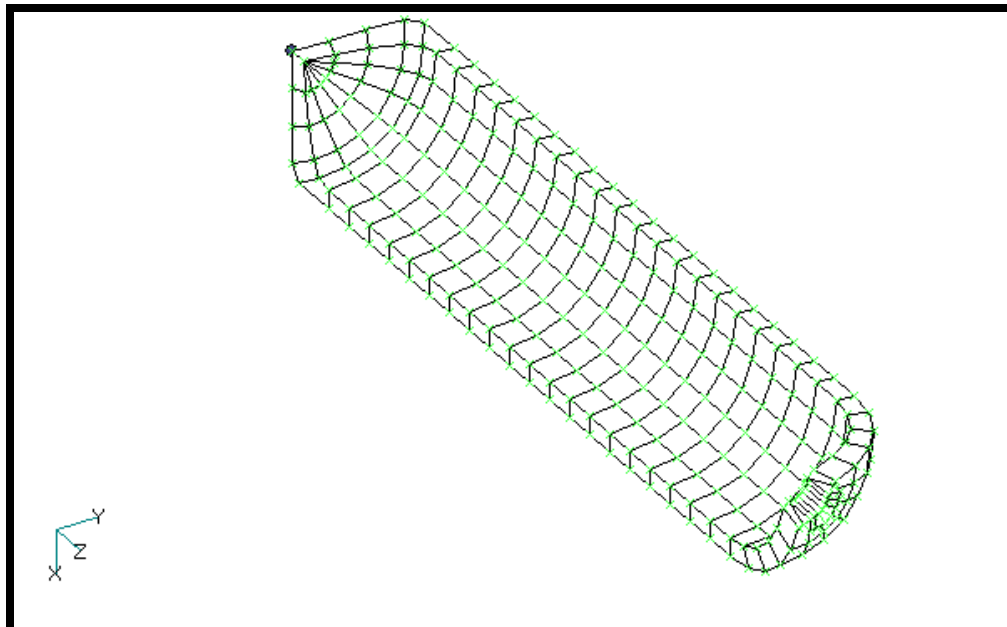
View/Select... <F5>

Model Style:

● **Full Hidden Line**

OK

Figure 21a.10



16. Apply constraints to the model.

Model/Constraint/Set... <Shift+F2>

Title:

Symmetric

OK

Apply symmetric constraints.

Model/Constraint/Nodal...

Method ^

on Surface

ID: 1

to: 14

by: 1

OK

DOF:

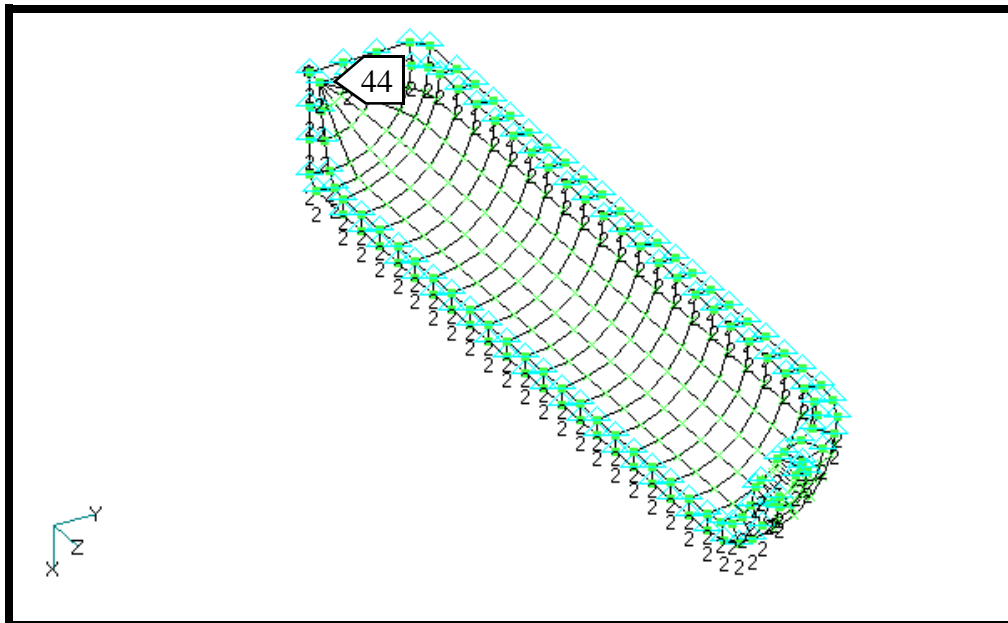
TT

OK

Select Node 44

(see Fig. 21a.11)

Figure 21a.11



OK

DOF:

TR

OK

When asked, "OK to Overwrite (No=Combine)?," select **No**.

No

Now restrain nodes on the opening in the z-direction.

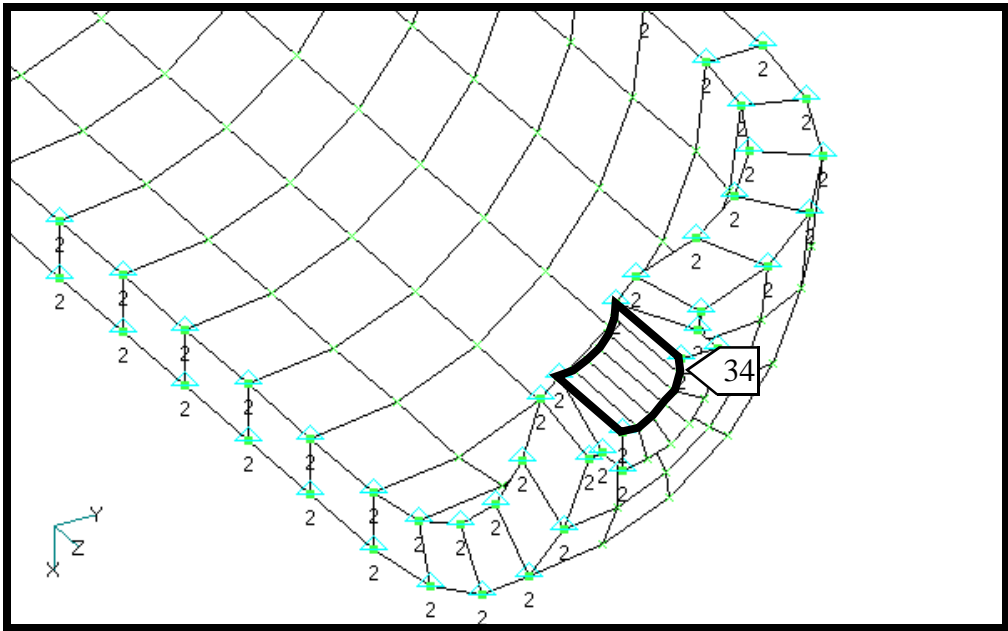
Method ^

on Surface

Select Surface 34

(see Fig. 21a.12)

Figure 21a.12



OK

DOF:

TZ

OK

No

Cancel

17. Create a load of 500 psi internal pressure.

Model/Load/Set... <Ctrl+F2>

Title:

5_Ksi

OK

Model/Load/Elemental...

Select All

OK

(highlight)

Pressure:

Pressure

5000

OK

Method:

Adjacent Faces

Pick a face element on the inside face of the tank.

Element:

100

Tolerance:

45

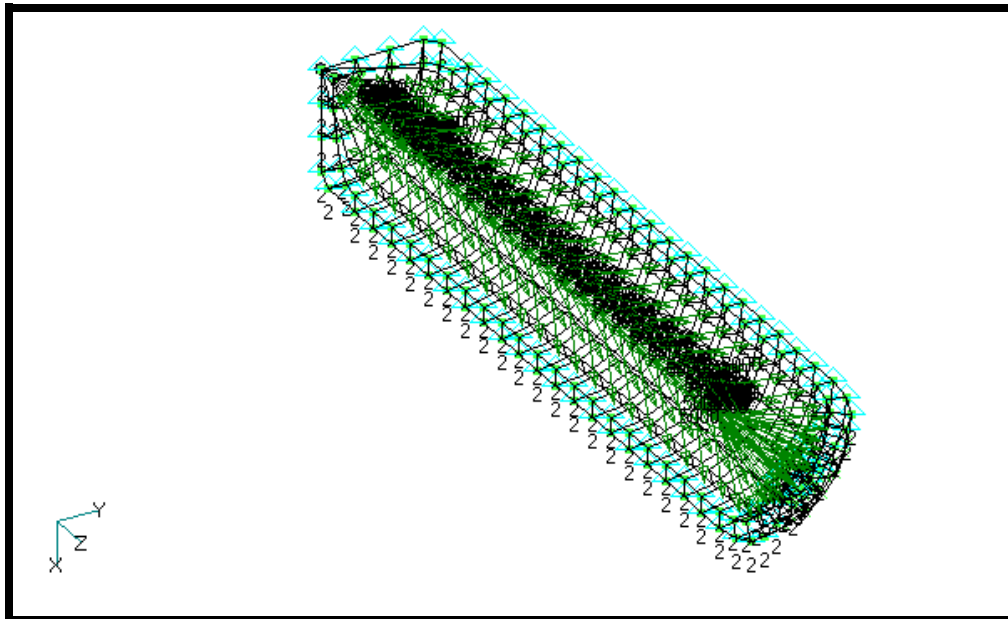
Face:

5

OK

Cancel

Figure 21a.13



18. Run an analysis.

File/Export/Analysis Model... <Ctrl+T>

Type:

File name:

Additional Info: Run Analysis

Output Types:

File name:

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 message(s), you could select Show Details. Since the analysis ran successfully, we will not bother with the details this time.

19. Review results.

Clean up the viewscreen by clicking on the following icon:



Draw: Element

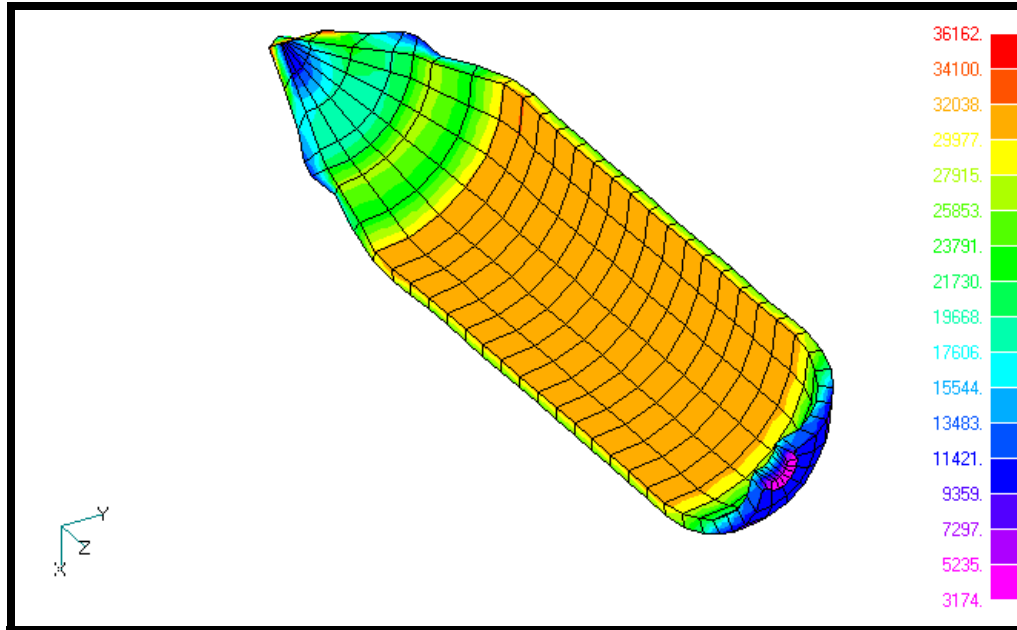
View/Select... <F5>

Deformed Style: Deform

Contour Style: Contour

OK

Figure 21a.14



View/Select... <F5>

Deformed Style:

None - Model Only

Contour Style:

Contour

Deform and Contour Data...

Contour:

60010..Solid X Normal Stress

OK

OK

View/Select... <F5>

Deform and Contour Data...

Contour:

60011..Solid Y Normal Stress

OK

OK

Model/Output/Error Estimate...

Output Vector:

60031..Solid Von MisesStresses

OK

View/Select... <F5>

Deform and Contour Data...

Output Vector:

300000..ERROR ESTIMATE 60031

OK

OK

20. Now a finer mesh model will be created.

File/New...

Yes

Tools/Advanced Geometry...

Geometry Engine:

Standard

OK

21. The quarter tank geometry was created and stored in a FEMAP Neutral file in the first part of this exercise.

File/Import/FEMAP Neutral...

Change directory to **c:\Mscn4w40\examples**.

File name:

tank_quarter.NEU

Open

OK

22. Mesh the geometry.

Mesh/Mesh Control/ Default Size...

Min Elem:

6

OK

View/Options... <F6>

Category:

Tools and View Style

Options:

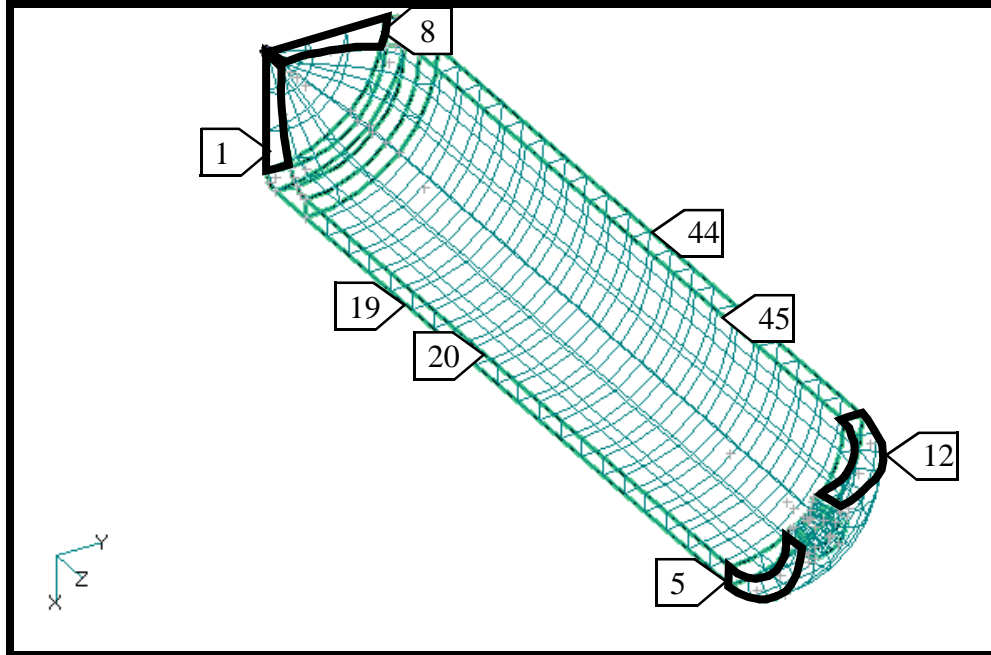
Curve and Surface Accuracy

Surface Divisions:

2..Show Mesh Size

OK

Figure 21a.15



23. Apply mesh control on surfaces.

Mesh/Mesh Control/Mapped Divisions on Surface...

Select Surfaces 1 and 8 (see Fig. 21a.15)

OK

	<i>s:</i>	<i>t:</i>
Number of Elements:	<input type="text" value="16"/>	<input type="text" value="6"/>
Bias:	<input type="text" value="1"/>	<input type="text" value="1"/>

OK

Select Surfaces 5 and 12 (see Fig. 21a.15)

OK

s: *t:*

Number of Elements:
Bias:

24. Apply a mesh control along curves.

Mesh/Mesh Control/Size Along Curve...

Select Curves 19, 20, 44, and 45 (see Fig. 21a.15)

Mesh Size:

Number of Elements

Node Spacing:

Biased

Bias Factor:

Node Spacing:

Small Elements at Both Ends

Mesh/Geometry/Volume...

Property:

Output Coordinate System:

CSys:

25. Equivalence the finite element mesh.

Tools/Check/Coincident Nodes...

No

Options:

Merge Coincident Entities

OK

View/Select... <F5>

Model Style:

Free Edge

OK

View/Select... <F5>

Model Style:

Full Hidden Line

OK

Click the *Quick Options* icon and turn off the geometry.



Quick Options

Draw:

Node

Geometry Off

Done

26. Apply constraints to model.

Model/Constraint/Set... <Shift+F2>

Title:

Symmetric

OK

Model/Constraint/Nodal...

Method ^

on Surface

ID: 1

to: 14

by: 1

OK

DOF:

TT

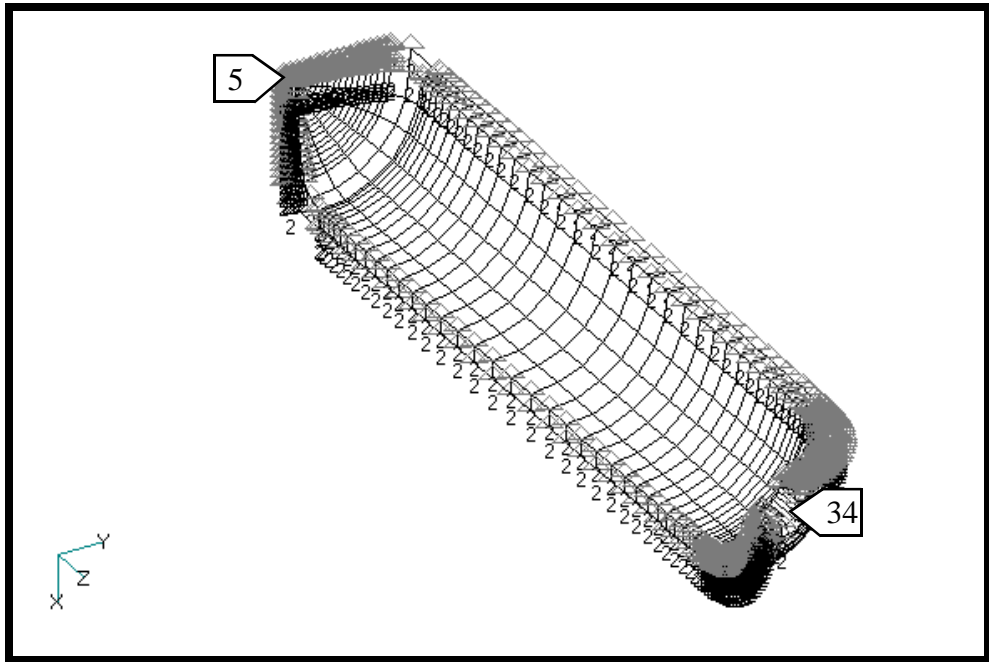
OK

Method ^

on Surface

Select Surface 34 (see Fig. 21a.16)

Figure 21a.16



OK

DOF:

TZ

OK

No

Method ^

on Curve

Select Curve 5 (see Fig. 21a.16)

OK

DOF:

TR

OK

No

Cancel

Click the *Quick Options* icon.



Quick Options

Draw:

Constraint

Done

27. Create loads.

Model/Load/Set... <Ctrl+F2>

Title:

5_Ksi

OK

Model/Load/Elemental...

Select All

OK

(highlight)

Pressure

Pressure:

5000

OK

Method:

Adjacent Faces

Select and element face on the inside face of the tank.

Element:

1640

Tolerance:

20

Face:

5

OK

Cancel

Click the *Quick Options* icon and turn off the geometry.



Quick Options

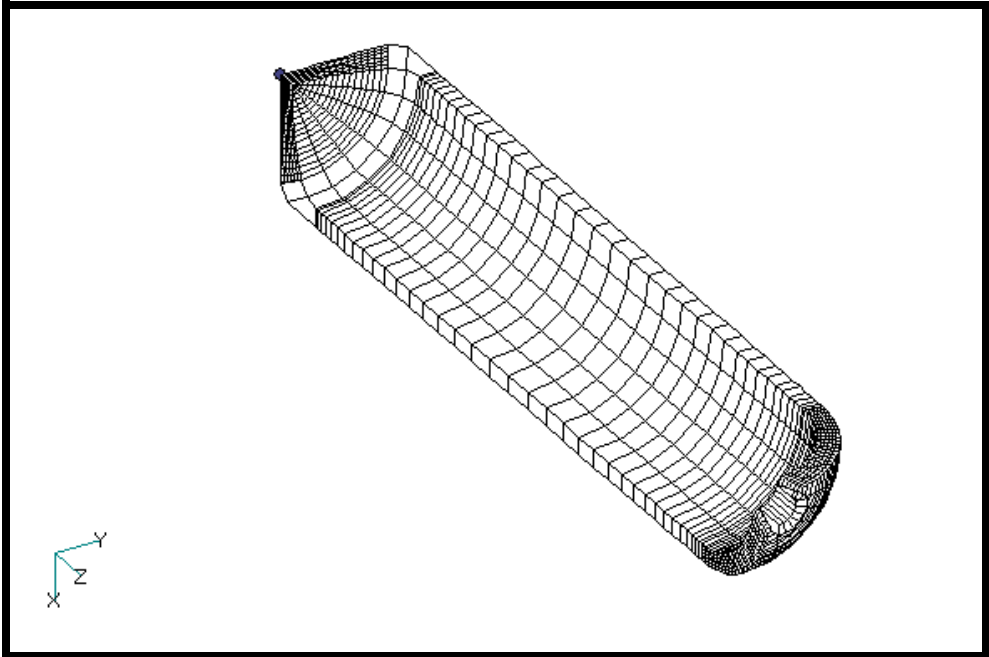
All Entities Off

Draw:

Element

Done

Figure 21a.17



28. Run analysis.

File/Export/Analysis Model...	<Ctrl+T>
<i>File Name:</i>	1..Static
OK	
<i>File Name:</i>	tank_fine
Write	
<i>Additional Info:</i>	<input checked="" type="checkbox"/> Run Analysis
<i>Output Types:</i>	2..Displacements and Stresses
OK	
Yes	
<i>File Name:</i>	tank_fine
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 message(s), you could select Show Details. Since the analysis ran successfully, we will not bother with the details this time.

Continue

29. Review and compare results.

View/Select... <F5>

Deformed Style: **Deform**

Contour Style: **Contour**

Deform and Contour Data...

OK

OK

View/Select... <F5>

Deformed Style: **None - Model Only**

Contour Style: **Contour**

Deform and Contour Data...

Contour: **60010..Solid X Normal Stress**

OK

OK

View/Select... <F5>

Deform and Contour Data...

Contour: **60011..Solid Y Normal Stress**

OK

OK

Model/Output/Error Estimate...

Output Vector: **60031..Solid Von MisesStresses**

OK

View/Select... <F5>

Deform and Contour Data...

Output Vector:

300000..ERROR ESTIMATE 60031

OK

OK

This concludes the exercise.

File/Save

File/Exit