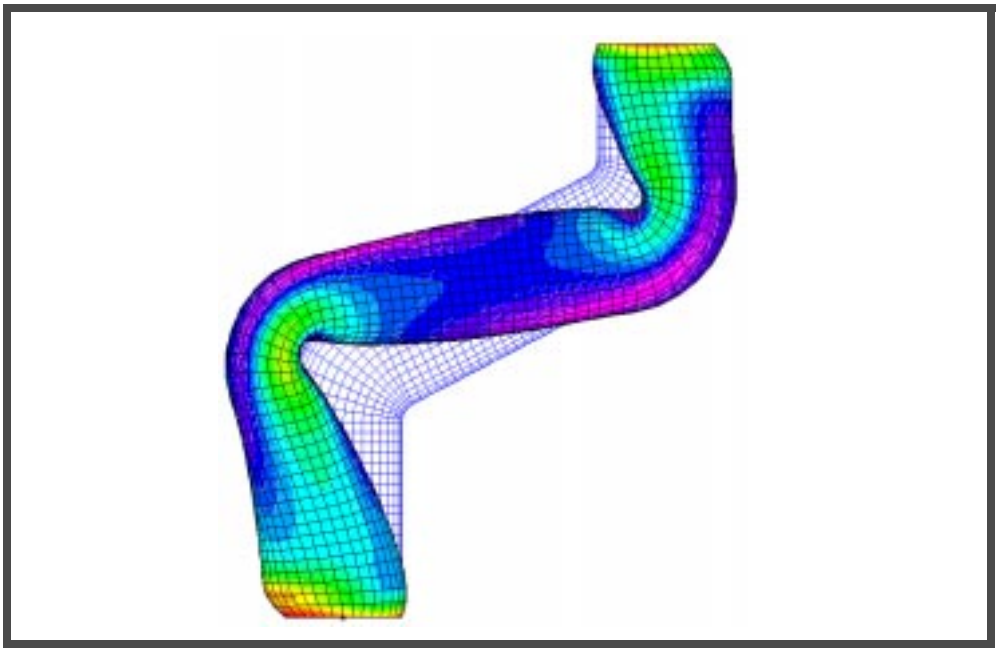
WORKSHOP 4

Curve Sweeping (V4.0)



Objectives:

- Create curves representing the cross-section and centerline.
- Sweep the curves to create surfaces representing the model.
- Create a mesh and apply appropriate LBCs.
- Run a Linear Static Analysis using MSC.Nastran.
- Create a Deformed/Contour plot.

Model Description:

Below is a proposed design for a pipe linkage, which will facilitate the transport of high pressure and temperature steam. Create the geometry for the model below and apply the appropriate loads, constraints, and element properties.

Figure 4.1

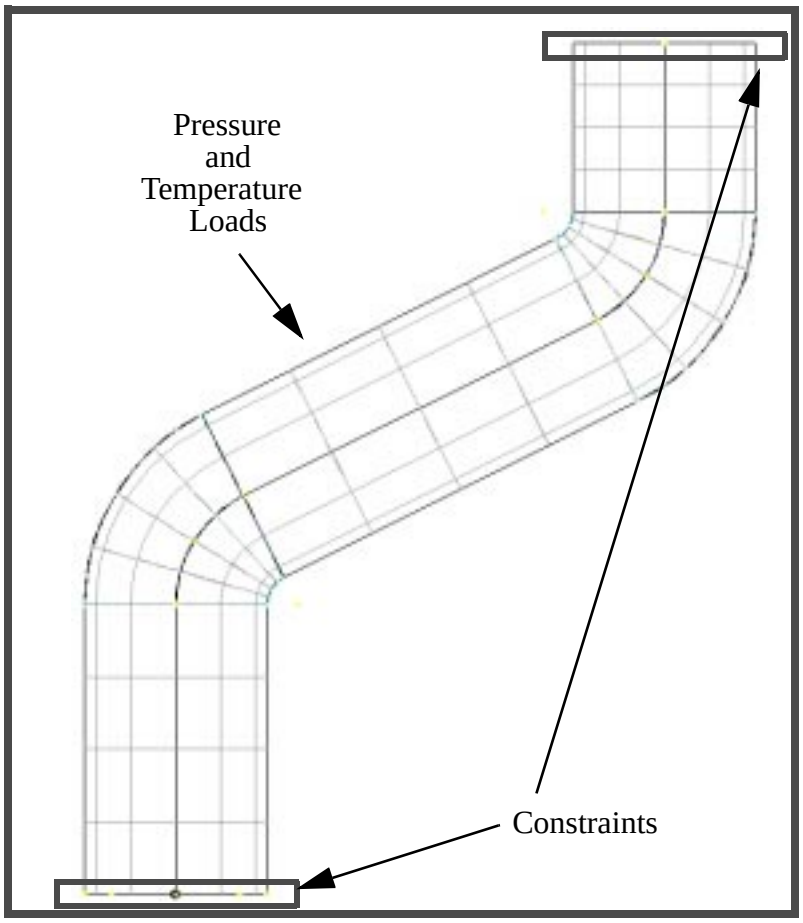


Table 4.1 - Material and Properties

Youngs Modulus:	**30E6 psi**
Poisson's Ratio:	**0.3**
Thermal Coefficient of Expansion:	**6.6E-6 in/in ^{o}F**
Reference Temp:	**75 ^{o}F**
Thickness:	**0.15 in**

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**

2. Create the basic lines for the geometry.

   Create the first line of the model by doing the following:

   Geometry/Curve-Line/Coordinates...

   First Location: *X:* *Y:* *Z:*

   0 **0** **0**

   OK

   Second Location: *X:* *Y:* *Z:*

   0 **6** **0**

   OK

   Repeat the process for the other 2 lines.

   HINT: MSC/N4W repeats the previous coordinate, to help minimize user input.

   First Location: *X:* *Y:* *Z:*

   0 **6** **0**

   OK

   Second Location *X:* *Y:* *Z:*

   8 **10** **0**

   OK

   First Location: *X:* *Y:* *Z:*

   8 **10** **0**

OK

Second Location: *X:* *Y:* *Z:*

8 **14** **0**

OK

Cancel

3. Clean up the view

View/Autoscale... **<Ctrl+A>**

4. Create fillets between the curves.

Modify/Fillet... **<Ctrl+F>**

Curve 1: **1**

Curve 2: **2**

Radius: **2**

With center near: *Click in the X box, then select a point on the screen near point A (see Figure 4.2)*

OK

Curve 1: **2**

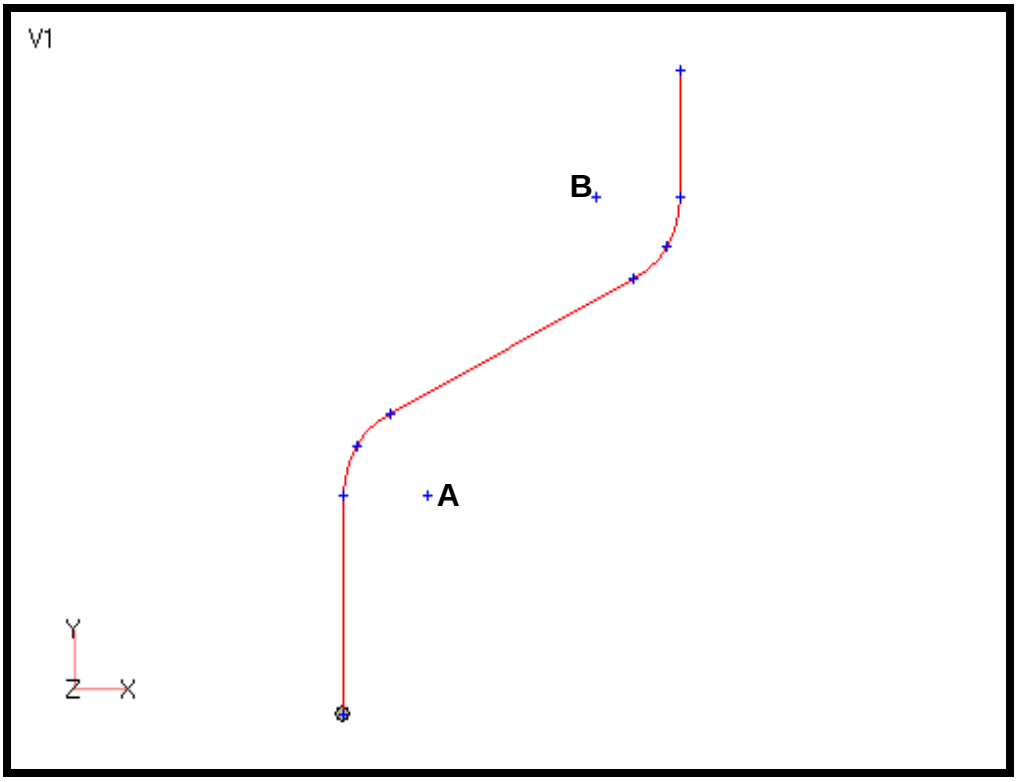
Curve 2: **3**

With center near: *Select point on the screen near point B (see Figure 4.2)*

OK

Cancel

Figure 4.2



5. Rotate the workplane to create the cross-section.

Tools/Workplane...	**<F2>**
Move Plane:	**Rotate...**
Methods ^	**Global Axis**
OK	
Rotation angle:	**90**
OK	

6. Choose *View/Regenerate* to regenerate the graphics.

View/Regenerate...	**<Ctrl+G>**

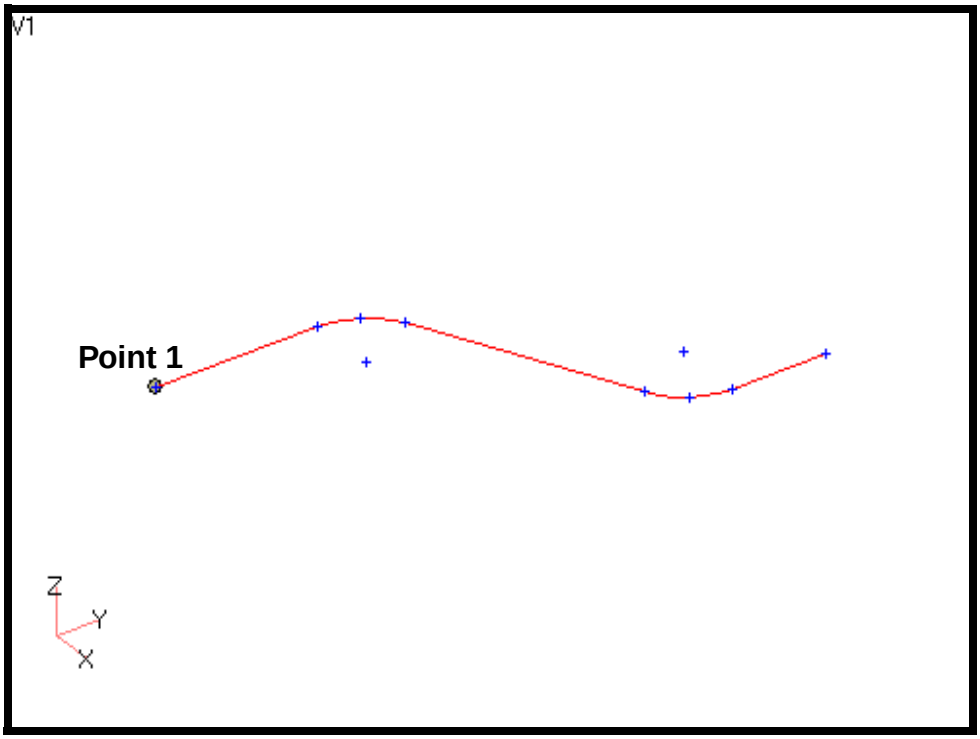
7. Rotate the model to a *Dimetric* view.

Choose View/Rotate and select a *Dimetric* view.

View/Rotate... **<F8>**

Dimetric
OK

Figure 4.3



8. Create the cross-sectional geometry.

Geometry/Curve-Arc/Angle-Center-Start...

Methods^	**On Point**
Point ID:	**1**
OK	
Methods^	**Locate in Workplane**
X:	**1.5**
Y:	**0**

OK

Angle: **180**

OK

Methods^ **On Point**

Point ID: **1**

OK

Point ID: **42**

OK

Angle: **180**

OK

Cancel

Clean up the view

View/Autoscale... **<Ctrl+A>**

9. Break the arcs in half.

Modify/Break... **<Ctrl+K>**

select the upper arc, ***Curve 6***

OK

Methods ^ **Midpoint**

Curve ID: **6**

OK

select the lower arc, ***Curve 7***

OK

Curve ID: **7**

OK

Cancel

10. Sweep to create the modal geometry.

Geometry/Surface/Sweep...

select the cross-sectional curves ***Curves 6, 7, 8, 9***

OK

select the curve to sweep along ***Curve 1***

OK

Cancel

View/Regenerate... **<Ctrl+G>**

Geometry/Surface/Sweep...

select the end of the newly created surface ***Curves 12, 16, 20, 24***

OK

select the curve to sweep along ***Curve 4***

OK

Cancel

View/Regenerate... **<Ctrl+G>**

Continue this pattern for the rest of the curves.

11. Create the material for the model.

Model/Material...

Title:	**Steel**
Youngs Modulus:	**30E6**
Poisson's Ratio:	**0.3**
Expansion Coeff:	**6.6E-6**
Reference Temp:	**75**

OK

Cancel

12. Create the element property.

Model/Property...

Title:	**pipe_wall**
Material:	**1..Steel**
Thickness:	**0.15**

OK

Cancel

13. Create a mesh.

Mesh/Mesh Control/Default Size...

Size:	**0.25**

OK

Mesh/Geometry/Surface...

select first cylinder ***Surfaces 1, 2, 3, 4***

OK

Property:	**1..pipe_wall**

OK

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

select 'elbow' ***Surfaces 5, 6, 7, 8***

OK

	s:	*t:*
Number of Elements:	**9**	**8**
Bias:	**1**	**1**

OK

Cancel

Mesh/Geometry/Surface...

select 'elbow' ***Surfaces 5, 6, 7, 8***

OK

Property: **1..pipe_wall**

OK

Continue this pattern to the rest of the geometry.

14. Check for coincident nodes.

Tools/Check/Coincident Nodes...

Select All

OK

When asked, "OK to Specify an Additional Range of Nodes to Merge?," select **No**.

No

Options: ☒ **Merge Coincident Entities**

Reporting: ☐ **Make Group to Keep**

OK

15. View the normal directions of the elements.

View/Options...

Category: ● **Labels, Entities and Colors**

Options: **Element-Directions**

☒ **Show Direction**

Normal Style: **1..Normal Vectors**

OK

Notice that the middle and end sections, as well as the 'elbow' sections, have different normal vector directions.

16. Reverse vector directions.

Modify/Update Elements/Reverse...

Methods^	**On Surface**

Select All

OK

Update Element Directions: ● **All Normal Inward**

OK

17. Choose *View/Regenerate* to regenerate the graphics.

View/Regenerate... **<Ctrl+G>**

18. Remove the element directions from the screen.

View/Options... **<F6>**

Options: **Element-Diections**

☐ **Show Direction**

OK

19. Rotate the model to a *XY Top* view.

View/Rotate... **<F8>**

XY Top

OK

20. Create the boundary constraints for the model.

Model/Constraint/Set...

Title: **end_constraint**

OK

Model/Constraint/Nodal...

select the nodes on the ends of the pipe (see Fig. 4.1)

OK

Pinned

OK

Cancel

21. Create the loading for the model.

Model/Load/Set...

Title: **load_1**

OK

Model/Load/Elemental...

Select All

OK

(highlight) **Pressure**

Pressure/Value: **110**

OK

Method: ● **Face ID**

Face: **2**

OK

Select All

OK

(highlight) **Temperature**

Temperature/Value: **450**

OK

Cancel

Submit the model for analysis.

File/Export/Analysis Model...

Analysis Format/Type: **1..Static**

OK

Save the new file under a temporary (TEMP) directory.

File Name: **prob4**

Write

Additional Info: ☒ **Run Analysis**

OK

When asked, "OK to Save Model Now?," select **Yes**.

Yes

Save the new file under a temporary (TEMP) directory.

File Name: **prob4**

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

22. Turn off the workplane.

Choose *Tools/Workplane* and uncheck the *Draw Workplane* box.

Tools/Workplane...

☐ **Draw Workplane**

Done

23. Display only the elements.

View/Options... **<F6>**

Quick Options

All Entities Off

Draw: ☒ **Element**

Done

OK

24. Display the results.

View/Select... **<F5>**

Model Style: ● **Quick Hidden Line**

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

Output Set: **1..MSC/NASTRAN Case 1**

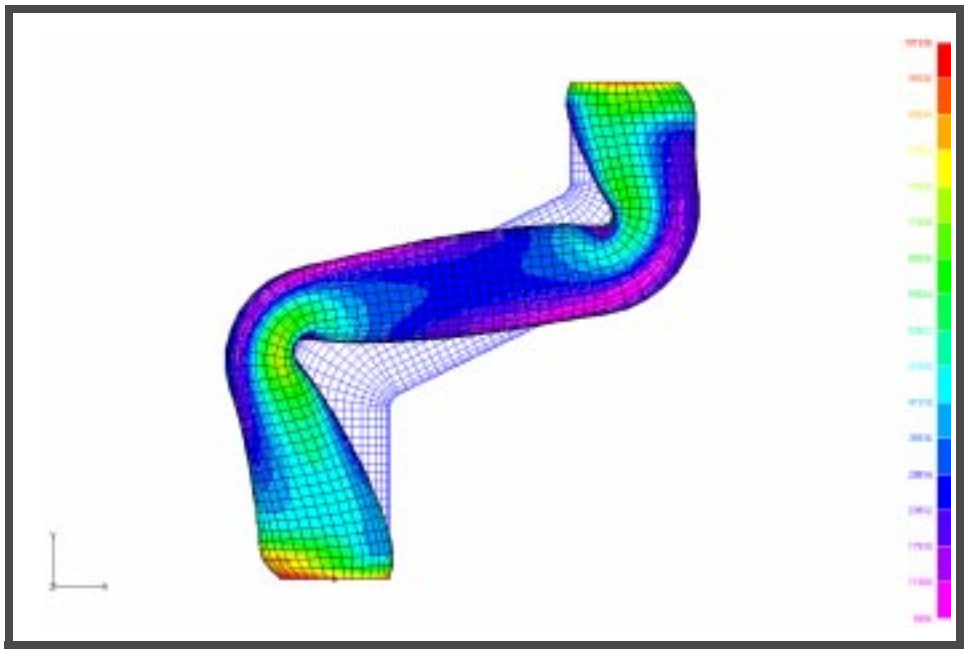
Deformation: **1..Total Translation**

Contour: **7033..Plate Top VonMises Stress**

OK

OK

Figure 4.4



Notice that the locations of highest stress are at the ends. This is because the model expands due to thermal loading and is constrained from movement (expansion) at these locations. In addition, the model should be expanding due to internal pressure.

Also, the model appears to undergo large deformations. This is because the deformation is scaled to 10% of the model size (to make viewing easier). This scale factor can be adjusted, if desired, under View/ Options..., the category set to Post-Processing and option Deformed Style..

This concludes the exercise.

File/Save

File/Exit