Basic Modeling

3

Modeling is the process of creating an accurate representation of the physical object and is the first step in analysis. This chapter describes the geometry creation and finite element modeling capabilities of MSC/NASTRAN for Windows (MSC/N4W).

The following section includes several brief examples of the geometry creation and finite element modeling commands available with MSC/N4W. This chapter is divided into the following sections:

3.1 Points, Lines and Circles

3.2 Splines

3.3 Surfaces

3.4 Finite Element Modeling

Modeling examples can be found in this chapter as well as in the preprocessing sections of many of the finite element analysis examples.

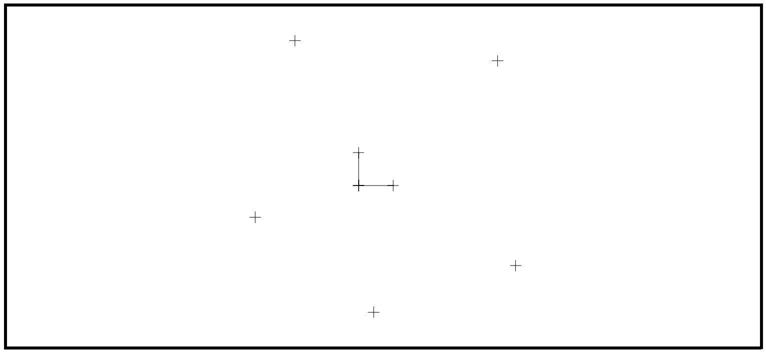
3.1 Creating Points, Lines and Circles

Geometric entities such as point, lines, circles, splines and surfaces form the foundation for fast and efficient generation of finite element models. The entities can either be built using the geometry creation capabilities provided within MSC/N4W or imported from your CAD system.

The following example demonstrates the creation of simple MSC/N4W geometry, involving lines, points, and circles.

To begin, follow these steps:

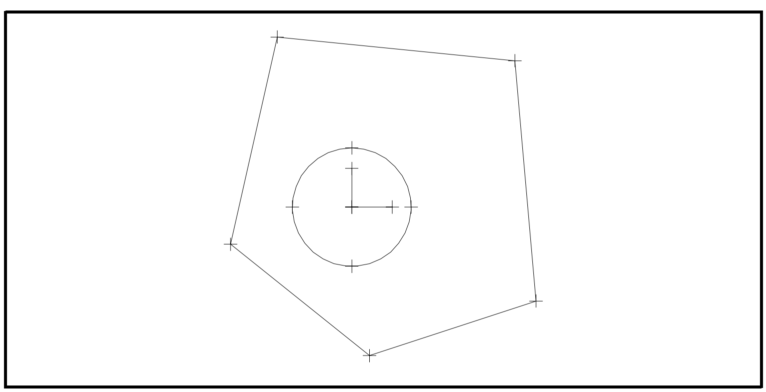
1. Start MSC/N4W with a new model.
2. Select **Geometry | Point**, then move the cursor to a location on the screen and click the left mouse button.
3. Click **OK** to create the point. continue, creating four more points in roughly the same orientation shown below:



4. Select **Cancel** after you have created the five points.

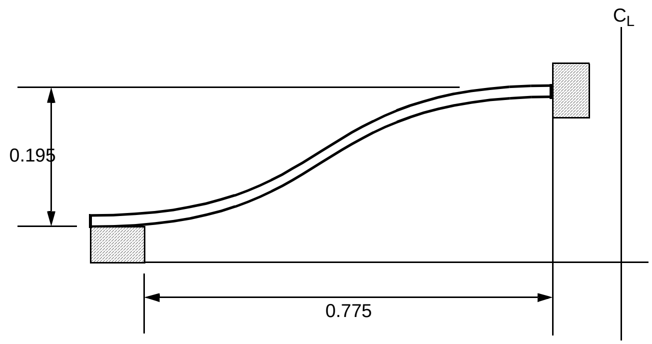
Now create lines and circles using the points

5. Select **Geometry | Curve - Line | Points**.
6. Select two points at a time to create the boundary of the part, and press **OK** after each pair has been selected. Press **Cancel** to continue.
7. The hole is created with **Geometry | Curve - Circle | Center**. Select a location at the center of the screen and press **OK**. Enter a **Radius** of ***0.5*** and press **OK** or hit the Return key.
8. Press **Esc** to clear the dialog box.
9. The final geometry should look like the following:



3.2 Creating Splines

The bellows of the flexible coupling shown below will be modeled in the following example to demonstrate the use of splines and surfaces in MSC/N4W.



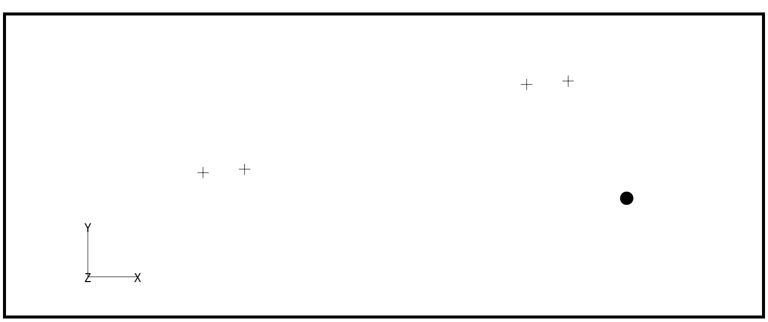
To begin, follow these steps:

1. Start MSC/N4W with a new model.

2. Use the **Geometry | Point** command to make four points that will define the spline at the centerline of the bellows. The required coordinates are as follows:

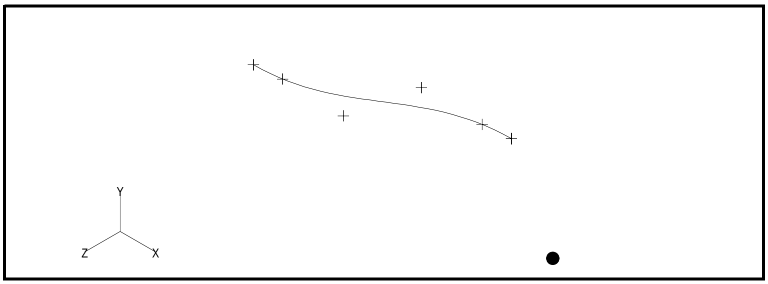
	X	Y	Z
1	-0.900	0.055	0.000
2	-0.812	0.062	0.000
3	-0.213	0.243	0.000
4	-0.125	0.250	0.000

Press **Cancel** to end the Create Point command. Then choose **View | Autoscale**. You should see the following view:



3. Change the default snap to points by pressing **Tools | Workplane | Snap Options** and set Snap To to **Point**, or select “**+**” icon on the toolbar. To create the spline at the centerline of the flexible bellows, select **Geometry | Curve - Spline | Points**. Select the points in order left to right by picking the left point, then **OK**, then the next point and **OK**, and so on. When done with the four points select **Cancel**.

4. Press **F8** to execute the MSC/N4W **View | Rotate** command or select it from the MSC/N4W menu. Select **Isometric**. MSC/N4W automatically rotates the model to a default isometric view. Select **OK** to save the view.

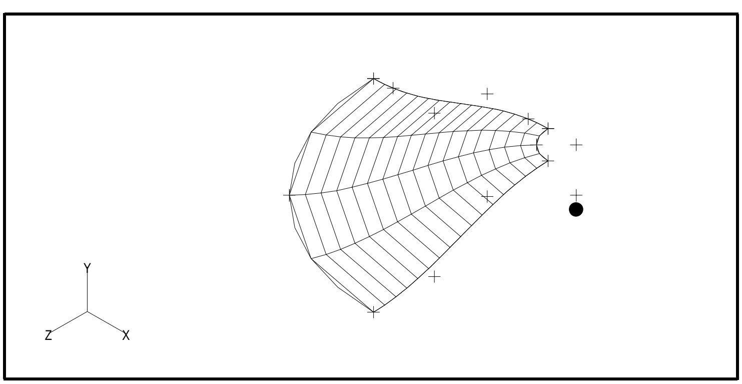


3.2.1 Creating a Revolved Surface

5. To create the revolved surface, select **Geometry | Surface | Revolve**. Select the spline and press **OK**.

6. MSC/N4W now prompts for a vector to revolve the spline about. Select **Methods** then **Global Axis**. In the Global Axis dialog box, select **Y Axis** and set the **Base** point to X=***0.00***, Y=***0.00***, Z=***0.00***, then press **Preview** to display the rotation vector on screen. Press **OK** if the vector appears correct.

7. MSC/N4W now prompts for the angle of rotation. Accept the default of 90º by pressing **OK**. Press **Esc** or **Clear** to clear the dialog box and **Ctrl+A** to autoscale the viewport. The surface should look like the following:



3.3 Creating a Ruled Surface

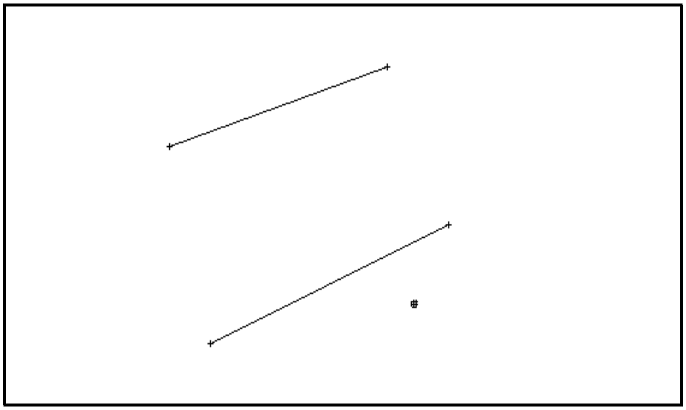
The following example will create a ruled surface between two edge curves. The surface is defined by the locus of straight lines connecting the curves.

1. Start MSC/N4W with a new model.
2. Use the **Geometry | Point** command to make four points that will define the spline at the centerline of the bellows. The required coordinates are as follows:

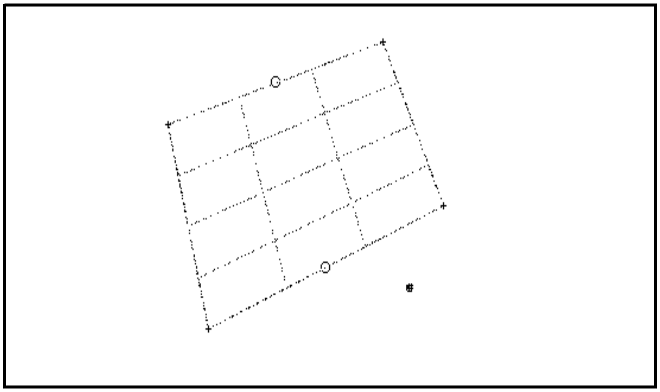
	X	Y	Z
1	-0.900	0.500	0.000
2	-0.100	0.750	0.000
3	-0.750	-0.250	0.000
4	0.125	0.250	0.000

Press **Cancel** to end the Geometry | Point command. Then choose **View Autoscale**. You should see all of the points:

3. Use the **Geometry | Curve - Line | Points** command to connect the points. First connect Point **1** and Point **2**, then connect Point **3** and Point **4**. Your model should appear as follows:



4. To generate a ruled surface, select **Geometry | Surface | Ruled**. Select line **1** and then line **2**. Press **OK** to create the surface.
5. Press **Cancel** to clear the dialog box. You should see the following:



3.4 Finite Element Modeling

Finite element analysis is a simulation tool that enables engineers to simulate the behavior of a structure. Finite element analysis is an important part of the overall design process, serving to verify or validate a design prior to its manufacture.

Because finite element analysis is a simulation tool, the actual design is idealized, with the quality of the idealization dependent on the skill and experience of the analyst. As more experience is gained, in both the modeling method and the design, confidence in the analysis results will also increase.

In finite element analysis, the design is discretized or subdivided into a series of elements that are connected by nodes. Material properties and element properties are specified to represent the physical properties of the model. Boundary conditions and applied loads are then defined to represent the operating environment for which the design is to be subjected.

As the analyst, the choices of element types and number of nodes are up to you. It is important to realize that your results will vary somewhat depending on these choices. It is also important to realize that the accuracy of your results—how well your model simulates reality—depends greatly on the proper choice of material properties, boundary conditions, and applied loads.

Let’s look at each kind of finite element input in detail.

Nodes

Nodes fundamentally define the geometry of the finite element model and are points to which the elements connect. Deformations can only be obtained at the node points, so the proper choice of the number of nodes is important. More nodes means more elements, and usually a more accurate model (from a discretization standpoint). However, it should be remembered that the computer time increases substantially with a finer model.

Nodes define admissible degrees of freedom. Each node has six degrees of freedom: X, Y, and Z translation, and X, Y, and Z rotation. Therefore, a general 3D model has the total number of degrees of freedom of six times the number of nodes.

Nodes are generated when the model is meshed. (The corresponding elements are generated at the same time.)

Coordinate Systems

MSC/N4W nodes can be defined in several coordinate systems—rectangular, cylindrical, and spherical. These coordinate systems aid in the specification of the overall finite element geometry.

Local coordinate systems can also be defined. These local coordinate systems are defined with respect to existing coordinate systems, and they can be offset and rotated.

Different coordinate systems can be used for nodal input and output. For example, the nodal location could be defined in a cylindrical coordinate system and the output computed in a rectangular coordinate system. Constraints are applied in the output coordinate system.

Element Types and Properties

Proper simulation of a real structure depends on the proper choice of element types and properties. There are several types of elements in MSC/N4W, broadly classified as follows:

3D elements that have a volume, which connect a set of non-planar nodes. 3D elements are used when the stress state varies in all three dimensions. Examples could include an engine block and “chunky” parts. Specification of the connecting nodes uniquely specifies each element.

2D elements that have an area, which connect a set of planar (or near-planar) nodes. 2D elements can be used when the stress state varies in two dimensions and is constant in the third dimension. 2D elements can also be used as plane strain elements. Common elements are membrane elements, which have stiffness only in the plane of the element, and plate elements, which have stiffness transverse to the plane of the element. Even though each element is 2D, an assemblage of 2D elements can be used to represent 3D geometry. Examples could include a car body and “thin” parts. Specification of the connecting nodes does <u>not</u> uniquely specify each element; the thickness of each 2D element has to be specified explicitly.

1D elements that have a length, which connect two nodes. Common 1D or line elements include rod elements, which carry only axial force, and beam elements, which transmit bending moments. Even though each element is 1D, an assemblage of 1D elements can be used to represent 3D geometry. Examples could include a bridge truss and a space frame. Specification of the connecting nodes does <u>not</u> uniquely specify each element; the cross-sectional area for rods, bars and beams and the bending inertias for bars and beams need to be specified.

Scalar elements are associated with either one or two nodes. Scalar elements include springs, dampers and concentrated masses.

Each element has an element coordinate system used for defining input and for interpreting output. The element coordinate systems are described in the Reference Manual.

Material Properties

The material—or materials—of the model has properties, such as Young’s modulus and mass density. These material properties are dependent on the type of material and can be defined explicitly or obtained from a material library within The following material types are allowed:

- ❍ Isotropic
- ❍ Orthotropic
- ❍ Anisotropic

Applied Loads

The applied loads define the operating environment to which the model is subjected. Loads include nodal forces and moments, pressures, body loads (such as gravity), and enforced displacement.

Boundary Conditions

Constraints define the boundary conditions of the model or how it is restrained. The boundary conditions are specified for each degree of freedom of each grounded node. Each degree of freedom can be specified independently, offering complete generality in describing boundary conditions. The boundary conditions are specified in the output coordinate system of the node.

Summary

The steps in creating a model are:

- ❍ Create or import geometry
- ❍ Specify material and element properties
- ❍ Mesh the geometry into nodes and elements
- ❍ Apply the loads and boundary conditions

Once the model is created it is ready for analysis

The following chapters contain step-through examples covering basic and advanced analyses. Further examples of basic modeling can be found in the Linear Static Analysis example, in the Basic Analysis chapter.