

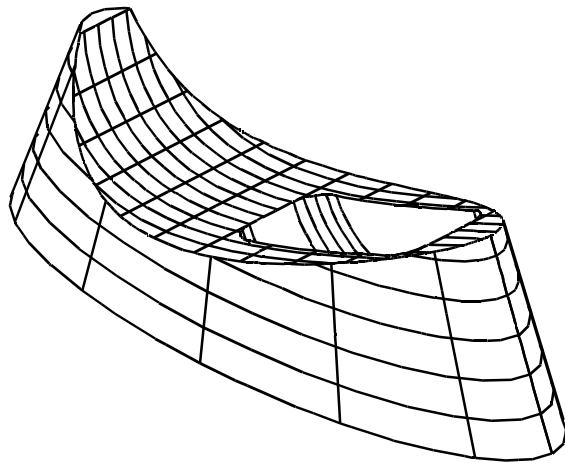
# Constructing a Tape Dispenser

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## Preface

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In this, your final project you will build a tape dispenser. Your part should look like Figure 1 below.



*Figure 1:*

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## Building the Initial Cross Sections

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Since constructing this part requires some precise point placement, we thought that color coordinating the points would ease the process. Therefore, some steps will use both the name of the point (P1, S1, etc.) and the color allocated to it. You will be making the majority of your part using the Attribute LTBLUE.

Geometry assigned the Attribute LTBLUE may appear white on some monitors. You may shade the part (in any view) using the SHADE command on 3D Page 3, in order to see the color more clearly. Be sure to unshade the part however, as you may be unable to locate specific construction points while the part shaded.

To build the initial cross sections of the dispenser, you are going to construct a spline through a set of given points, and then construct an ellipse through a second set of points.

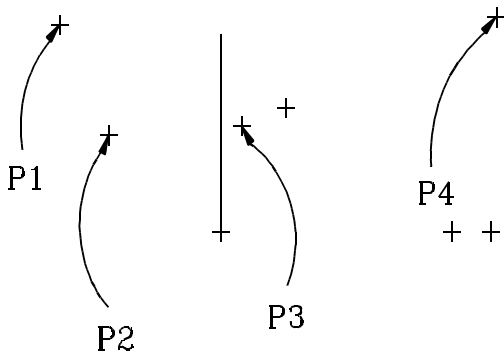


Figure 2:

1. Open the drawing DSPNSR.DWG. Click on **Edit > Surface and Boolean Solids**.
2. Click on **Draw > Spline > Create**.
3. Set the -Input Style ? modifier to “thru points”.
4. In the front view, click on points P1, P2, P3, and P4 (the violet points), as shown in Figure 2.
5. Click on **Done** to accept the points for the spline.

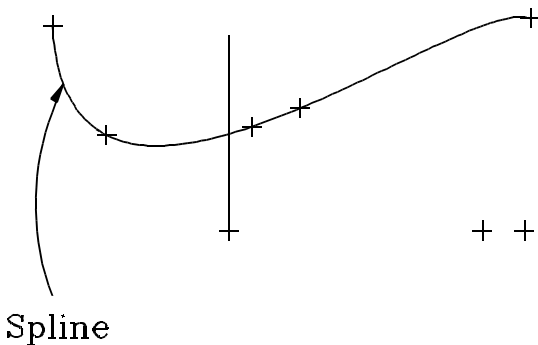


Figure 3:

Figure 3 shows the spline created, passing through the points specified.

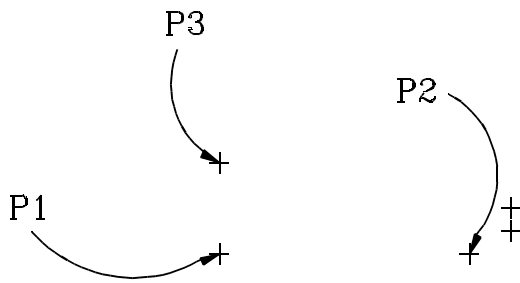


Figure 4:

6. Click on **Draw > Ellipse > Center, 2 Points**.
7. Set the -Attribute Name ? modifier to 'ltblue'.
8. In the top view, click on points P1, P2, and P3 (the blue points), as shown in Figure 4.

Point P1 is the ellipse center, P2 is a point at 0 degrees, and P3 is the 90 degree point of the ellipse.

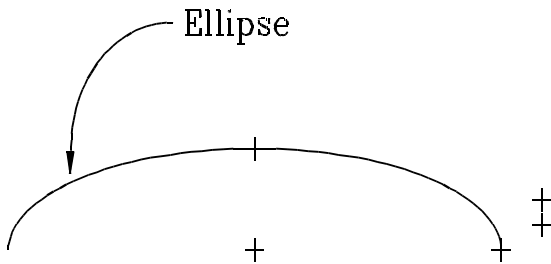


Figure 5:

Figure 5 shows the ellipse created in the top view, passing through the points specified.

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## Creating the Outer Wall

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To create the outer wall of the dispenser, you are going to scale and extrude the ellipse upward, at an angle of 15 degrees from the center of the ellipse.

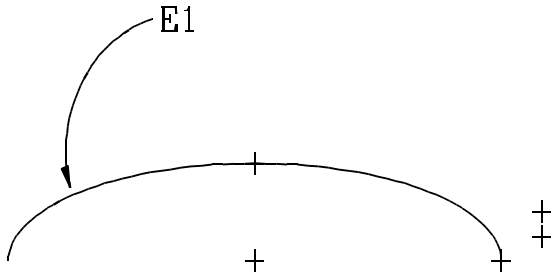


Figure 6:

9. Click on **Select > Select element**.
10. Click on E1 to select the ellipse.
11. Click on **Draw > Surface > Scale by Angle**.
12. Set the -Angle ? modifier to 15.

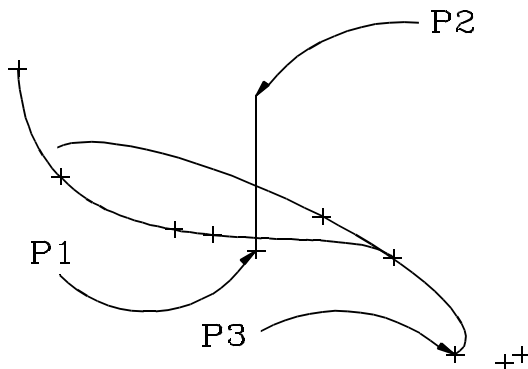


Figure 7:

13. In the isometric view, click on points P1 (the base of the blue line), P2 (the top of the blue line), and P3 (the blue point at the start of the ellipse), as shown in Figure 7.

Points P1 and P2 define the neutral axis of the ellipse. The distance between P1 and P2 determines the amount by which to extrude the ellipse. Point P3 is the point that scales at the specified angle (in this case, 15 degrees) to the neutral axis. The angle of all other points is proportioned to their distance to the neutral axis and point P3.

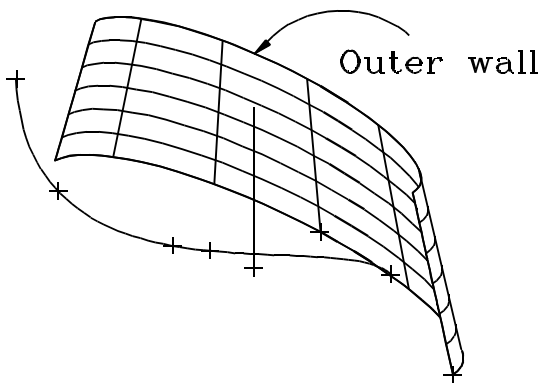


Figure 8 shows the generation of the outer wall.

Figure 8:

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## Making the Basic Shape of the Tape Cavity

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In this section, you will be building the basic shape of the tape cavity. You will place a circle onto the outer wall that you created earlier, and then sweep that circle to create a surface.

Looking down on the top of the dispenser, you will place a circle that represents one wall of the cavity at the depth indicated by the dashed line in Figure 9. The circle will be placed in the front view. To ensure the proper depth, you will use **Set Depth** to temporarily freeze the depth in the front view.

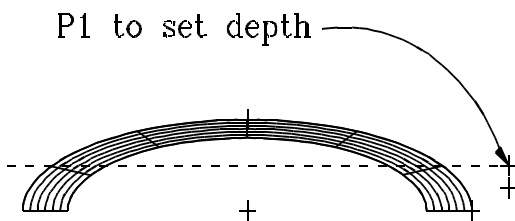


Figure 9:

14. Click on the **Set Depth** icon on the Quick Snap tool bar or along the right edge of the display.
15. Click on the **Set Depth** button in the function box.
16. In the top view, click on point P1 (the topmost red point), as shown in Figure 9 to freeze the depth to this point. The -Depth Snaps? sets itself to 'view'.

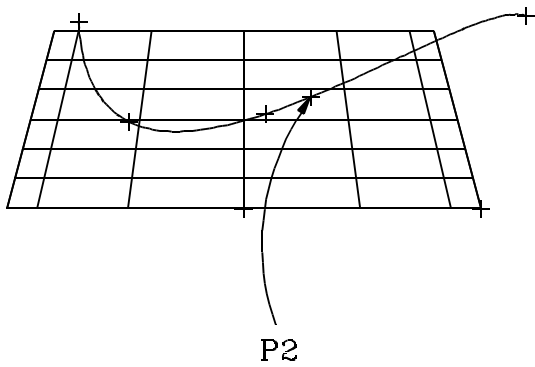


Figure 10:

17. Click on **Draw > Circle > Center Point**.
18. Set the -Circle Plane ? modifier to “view”.
19. In the front view, click on point P2 (the green point), as shown on Figure 10.
20. Type Ctrl+ 14.5, then press **Enter** to finish the circle.
21. Click on the **Set Depth** icon and set -Depth Snaps? from “view” back to “free”.

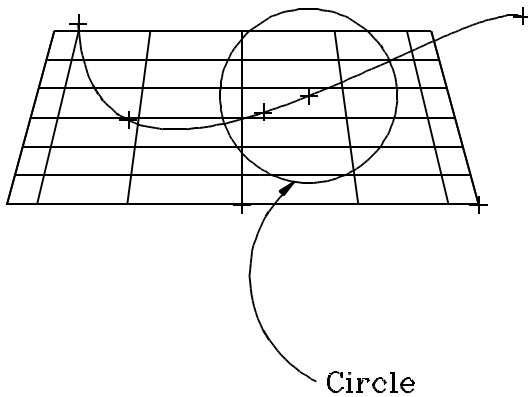


Figure 11:

Figure 11 shows the size and placement of your circle.

Using **Set Depth** enabled you to perform “2D like” construction in an orthogonal view, while freezing the depth.

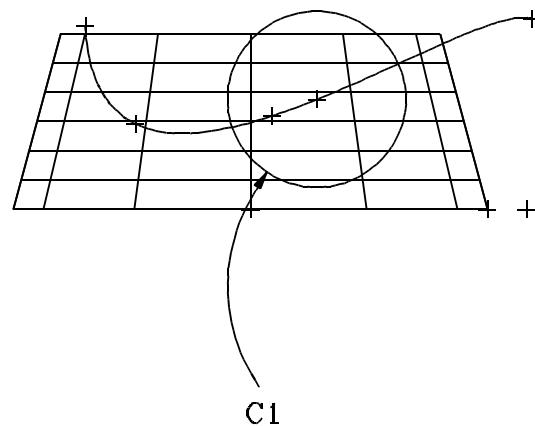
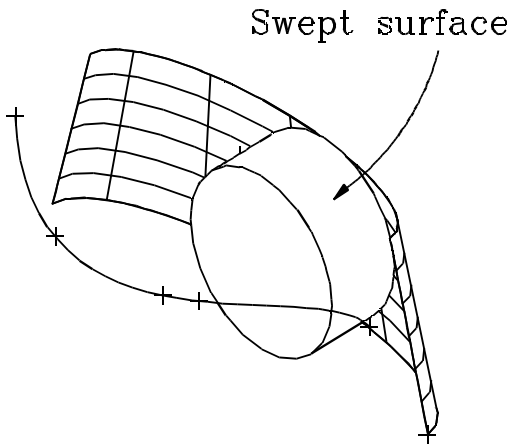


Figure 12:

22. Click on **Select > Select Element**.
23. Click on C1 to select the circle.
24. Click on **Modify > Sweep > Linear, Surfaces**.
25. Click anywhere in the drawing to choose a pickup point.
26. Type 4rz, then press **Enter**, to place your put down point.

You can replace steps 25 and 26 by using the grid in the top view, picking any intersection as your pick up

point, and then picking a point a distance of 4 in the Z axis from the pick up point.



Your model should look like Figure 13.

Figure 13:

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## Filleting the Tape Cavity

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To fillet the tape cavity, you are going to place a fillet between the back plane of the cavity and the cylinder. Then, you are going to cut and remove the excess geometry.

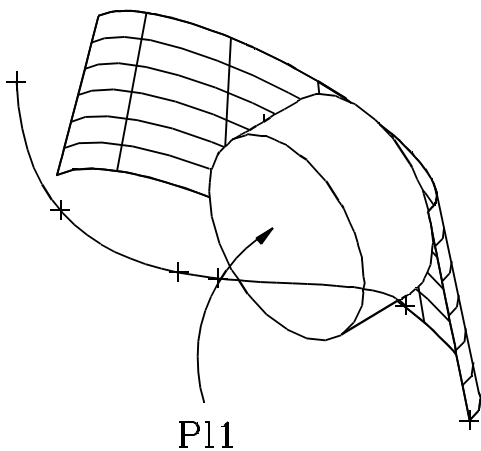
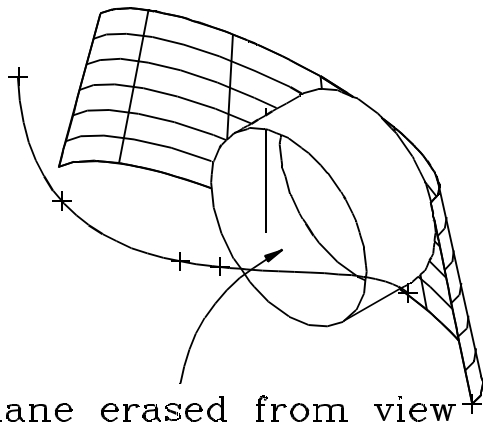


Figure 14:

27. Click on **View > Zoom**.
28. Click on the top left and the bottom right corners of the isometric view to zoom.
29. Click on **Select > Select Element**.
30. Click on P11 to select the front plane. Make sure that you click on the surface of the plane and not on boundary geometry.

The current view does not permit you to see the back plane of the cavity. However, in order to create a fillet between the back of the cavity and the cylinder, you need to be able to select the back surface. CADMAX provides a facility that allows you to temporarily peel away or remove a surface so that you can work inside of the object. This is called “view editing”.



Plane erased from view

Figure 15:

31. Click on **3D View > Erase from View**.
32. Click anywhere within the isometric view to remove the selected surface. Now the front plane has been removed, permitting you to see the back of the cavity, as shown in Figure 15.
33. Click on **Clear**.

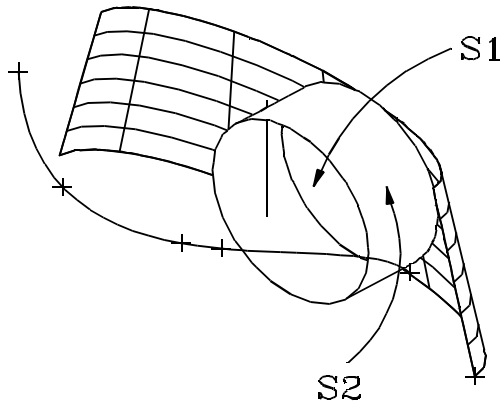


Figure 16:

34. Click on **Modify > Corner > Fillet 2 Surfaces**.
35. Set the -Radius? modifier to 1.
36. Click on S1 (the back plane) and S2 (the cylinder), as shown in Figure 16. When you click on the surfaces, make sure that there is no geometry inside of the cursor.

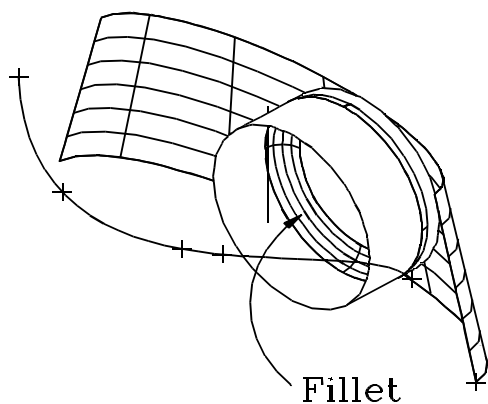


Figure 17:

Figure 17 shows the resulting fillet and its location.



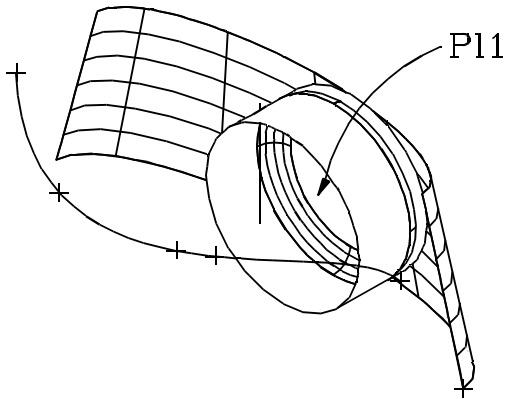


Figure 18:

In the next series of steps, you will cut the back plane of the cavity where it intersects with the fillet, and then remove the portion of the plane that is outside of the fillet edge.

37. Click on **Select > Select Element**.
38. Click on P11 to select the back plane. Make sure that you click fully on the back plane, and that no geometry is inside the cursor.

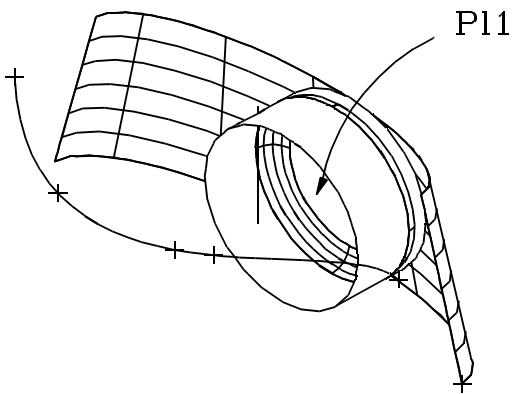


Figure 19:

39. Click on **Modify > Edit Surface > Cut**. This cuts the selected surface into two surfaces where it intersects with the fillet. You will know that the cut is successful because you will see two "P's", indicating that the single surface is now two surfaces.
40. Click on **Select > Unselect Element**.
41. Click on PL1 to unselect the inner back plane, leaving selected the surface outside of the fillet.
42. Click on **Modify > Erase Element**.
43. Click on **Select > Clear**.

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## Filleting the Other Side of the Tape Cavity

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To fillet the other side of the tape cavity, you are going to follow the same procedure as in the previous section. In this case, however, you have first to replace the front plane that you removed from view earlier. You will place a fillet between the front plane and the cylinder, and then cut and remove the excess geometry.

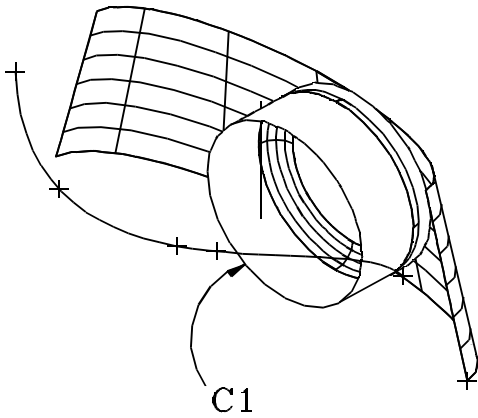


Figure 20:

44. Click on **Select > Select > Attached Surfaces**.
45. Click on C1 to select the front circle and all of the surfaces attached to the front circle.

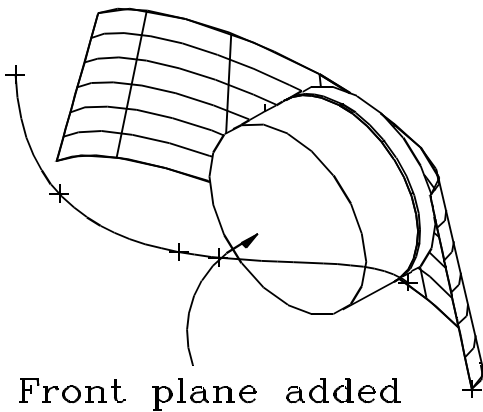


Figure 21:

46. Click on **3DView > Add to View**.
47. Click on the isometric view to restore the front plane that you removed from view earlier.

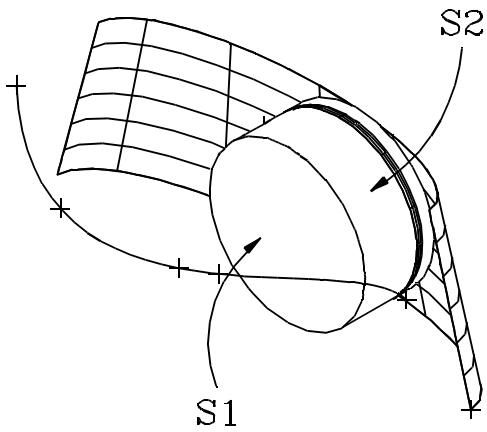


Figure 22:

48. Click on **Modify > Corner > Fillet 2 Surfaces**.

49. Set the -Radius ? modifier to 1.

50. Click on S1 (the front plane) and S2 (the cylinder), as shown in Figure 22.

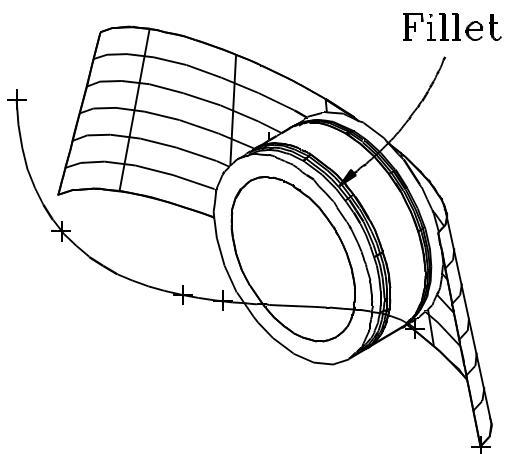


Figure 23:

Figure 23 shows the resulting fillet.

In the final sequence to create the cavity, the front plane must be cut where it intersects the fillet, and the cylinder must be cut in two places where it intersects the two fillets. Then the excess surfaces must be removed.

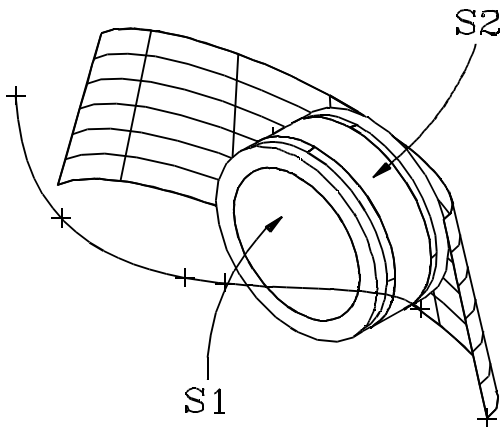


Figure 24:

51. Click on **Select > Select > Boundaries of Selected Surfaces.**
52. Click on **Modify > Edit Surfaces > Cut** to cut the selected surfaces where they intersect with the fillets.
53. Click on **Select > Unselect Element.**
54. Click on S1 and S2 to unselect the inner cylinder and the inner plane respectively, as shown in Figure 24.

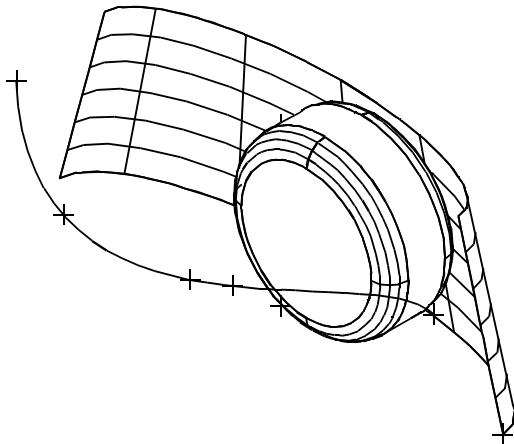


Figure 25:

55. Click on **Modify > Erase Element.**
56. Click on **Select > Clear.**
57. Click on **View > Show Largest Fast.**

Figure 25 shows the tape cavity after removing the excess geometry.

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## Completing the Basic Shape

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Now, you are going to complete the basic shape of your tape dispenser. First, you will create the second, outer wall by making a mirror copy of the first wall. Then, you will place a surface over the top of the dispenser, effectively joining the two walls.

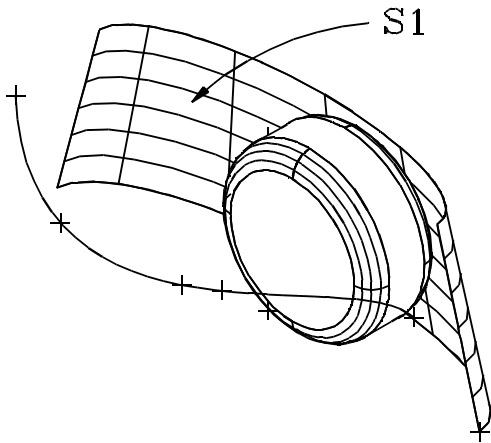


Figure 26:

58. Click on **Select > Select Element**.
59. Click on S1 to select the outer wall.

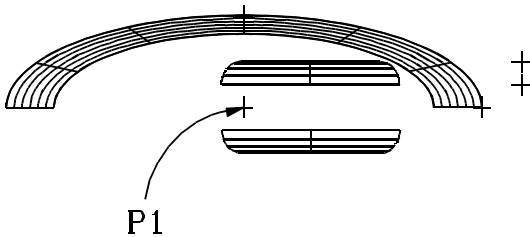
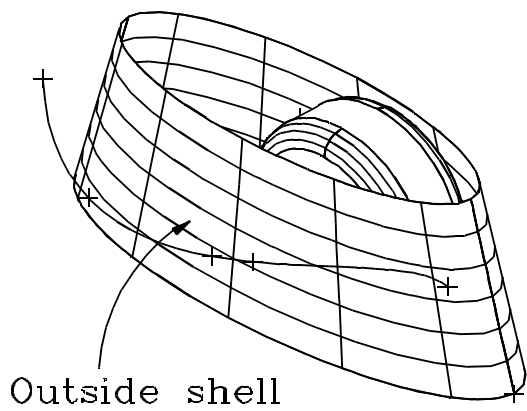


Figure 27:

60. Click on **Modify > Mirror**.
61. Set the -Move Which ? modifier to “copy”.
62. Click on point P1 in the top view, as shown in Figure 27, to place the mirrored copy of the outer wall.
63. Click on **Select > Clear**.



Outside shell

Figure 28:

Figure 28 shows the placement of the outer wall.

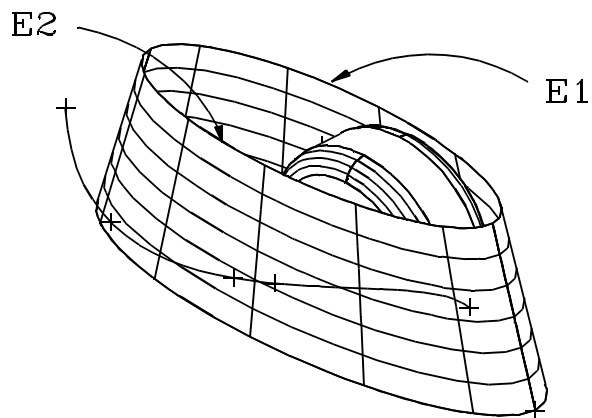


Figure 29:

64. Click on **Select > Select Element**.

65. Click on E1 and E2 to select the ellipses at the tops of the outer walls.

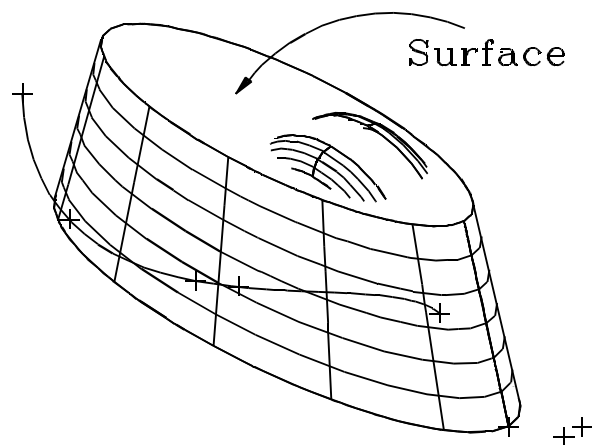


Figure 30:

66. Click on **Draw > Surface > Simple**, then click on **Finish** in the Make Surfaces function box.

Figure 30 shows the resulting surface.

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## Creating the Dispenser Base

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Now, you're ready to use the spline that you created in step 1. The spline is a profile of the top edge of the dispenser. In the next series of steps, you will extrude the spline through the base, and then remove all of the excess geometry above the extruded spline to form the tape dispenser.

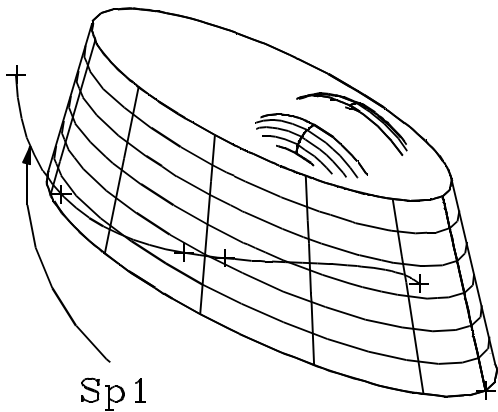


Figure 31:

67. Click on **Select > Select Element**.
68. Click on Sp1 to select the violet spline.
69. Click on **Draw > Surface > Extrusion**.

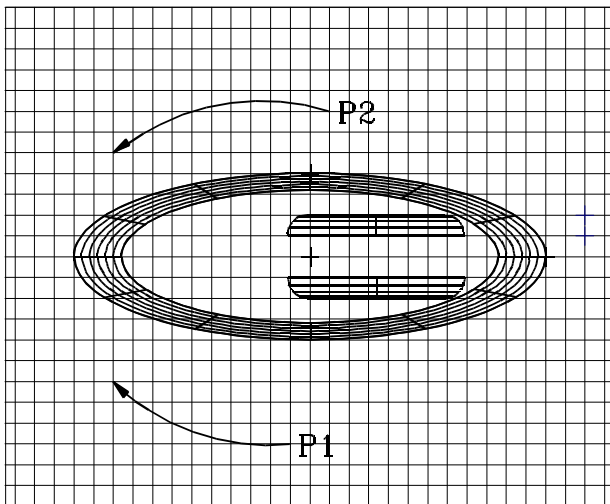


Figure 32:

70. Working in the top view, click on a pick up point on the grid in line with the spline, as shown in Figure 32.
71. Click on a put down point on the opposite side of the base, in a straight direction from the pick up point, as shown in Figure 32.

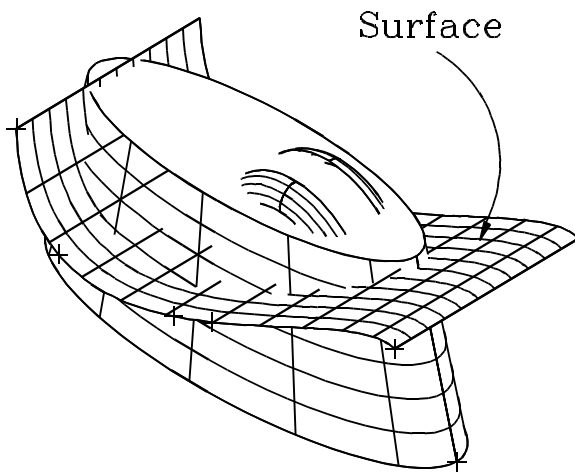
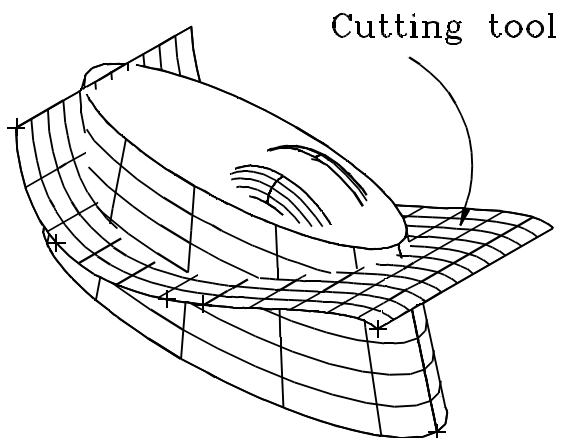


Figure 33 shows the resulting surface of the extruded spline through the basic shape of the dispenser.

Figure 33:

Your dispenser is almost finished! Now, you have to remove the excess geometry from the basic shape. Using the extruded spline surface that you created in the previous section, you will intersect and trim the basic shape, then erase the unnecessary geometry.



72. Click on **Select > Select > Geometry Attribute**.

73. Set the -Attribute Name ? modifier to ltblue.

74. Click on **Modify > Edit Surface > Intersect & Trim**.

75. Click on the extruded spline surface to choose it as your cutting tool. The boundary of the surface will be highlighted in red, indicating that you have selected that surface as the cutting tool.

Figure 34:



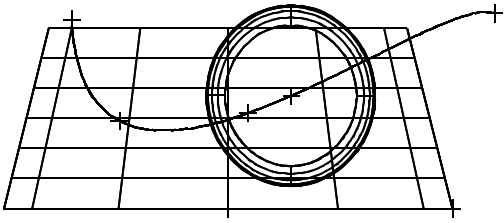


Figure 35:

76. In the front view, click on any point below the extruded spline surface to indicate that you want to keep all of the geometry below the surface.

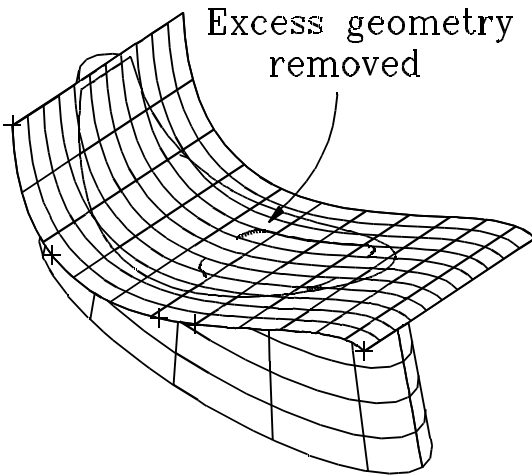


Figure 36:

77. Once CADMAX processes the geometry, click on **Modify > Erase Element** to remove all of the geometry above the extruded surface.
78. Click on **Select > Clear**.

At this point, your part should appear like Figure 36.

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## Finishing: Keeping the Sectioned Proportions

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All you have to do now, is perform one more cut using the violet extruded surface, erase a small amount of geometry, and you have your tape dispenser!

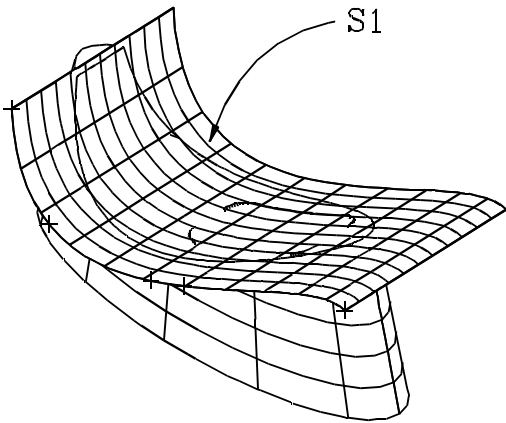


Figure 37:

79. Click on **Select > Select > Single Surface**.
80. Click on S1 to select the extruded surface.
81. Click on **Modify > Edit Surface > Cut** to cut the surface where it intersects with the base and the cavity.

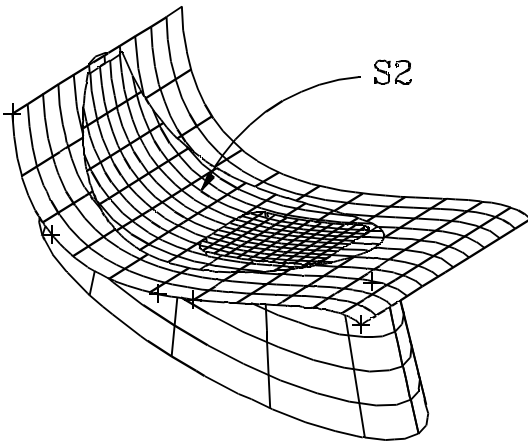
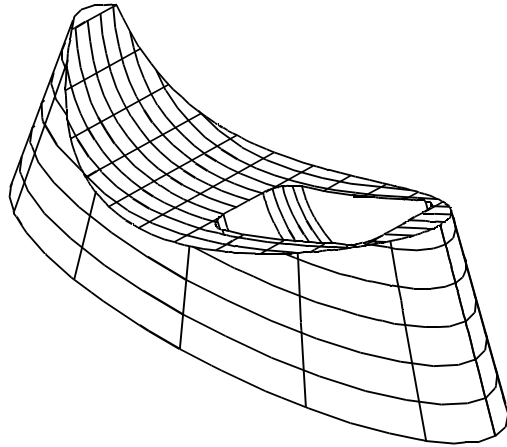


Figure 38:

82. Click on **Select > Unselect > Surface**.
83. Click on S2 to unselect the portion of the surface that you want to keep.
84. Click on **Modify > Erase Element**.
85. Click on **Select > Clear**.

Your tape dispenser is finished! For fun, you can try adding the spool and tape. Your dispenser should look like Figure 39 below.



*Figure 39:*