4

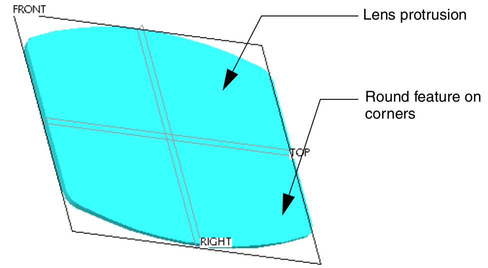
Modeling the Cell Phone

In previous chapters you've become familiar with the interface controls and some of the basic concepts needed to get started in Creo Elements/Pro. In this chapter you'll start the hands-on process of building the eight parts that make up the cell phone model. Before you start the exercises, you should be familiar with the selection tools, the zoom and pan controls, and the basics of using Sketcher. All of these items are covered in the previous chapters.

The instructions for each part begin with a table that lists the techniques used for that part. The first time a technique is introduced, you are given detailed instructions. If the technique is used again in another part, you are only given any additional instructions you'll need to use it in that instance. If you need to review how to use a technique, use the table to find the previous section that covers it in detail.

When you have created all the parts, you will proceed to add them to an assembly and output some detailed mechanical drawings.

Part 1: Lens



In this procedure, you will create a simple solid extrusion that represents the lens of the cell phone. It is centered on horizontal and vertical axes formed by the intersection of two of the datum planes. You'll learn how to quickly mirror drafted lines in Sketcher, so that mirrored halves are constructed and constrained to always be proportional. You'll also learn how to add round features to edges. Finally, you'll learn how to save a sketcher for reuse in another part.

Technique or Feature	Where Introduced
Insert Extrusion	New
Enter Dashboard and Sketcher	New
Mirror Feature	New
Sketcher Section Creation	New
Rounded Features	New

Sketch the Lens Protrusion

To begin, click **File** > **New**. The **New** dialog box opens. Click **Part** in the **Type** area of the dialog box and enter the word `lens` as the part name. Remember to create a working directory if you haven't already done so.

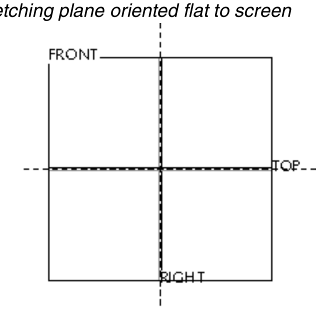
1. Click the Sketch tool on the Feature toolbar. The **Sketch** dialog box opens.
2. Click the Front datum plane in the graphics window or in the Model Tree to select it as the sketching plane. An arrow indicates the view direction. Leave the orientation and view direction set at the default. The Right datum is automatically referenced in the dialog box.
3. Click **Sketch** in the **Sketch** dialog box to start Sketcher. The background changes color, and perpendicular lines bisect the sketching plane in the x-and y-direction. The Sketcher toolbar appears to the right of the graphics window. Use these Sketcher tools to sketch the lens and other parts.

   Note

   At any point, use the datum display buttons on the main toolbar to turn off datum display and clear up the work area. You should see only the horizontal and vertical reference lines.

4. Click the **Sketch Orientation** icon on the main toolbar to orient the sketching plane flat to the screen. The sketching window should now look like the figure below:

Sketching plane oriented flat to screen

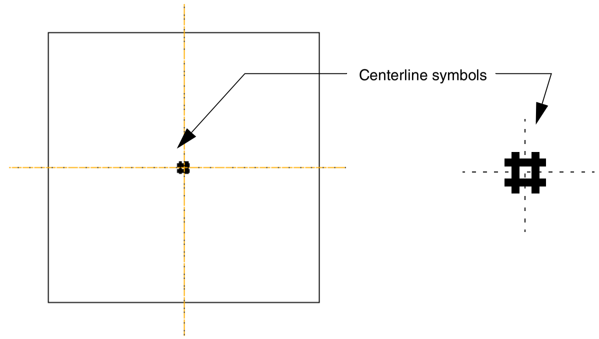


Add Centerlines

Before you draw, you'll add centerlines to the vertical and horizontal axes formed by the Top and Right datum planes. You can use these centerlines to mirror shapes and dimensions and to quickly center geometry by snapping lines to them.

1. On the Sketcher toolbar, click the Centerline tool from the Line flyout menu .
2. Let the pointer snap to one of the axis lines and click the left mouse button. A yellow centerline appears, attached to the pointer. Move the pointer to rotate the centerline so it coincides with the axis line, and click to place it. Place a centerline on both the vertical and horizontal datum plane references. Middle-click to exit the tool.

Sketching plane with centerlines added

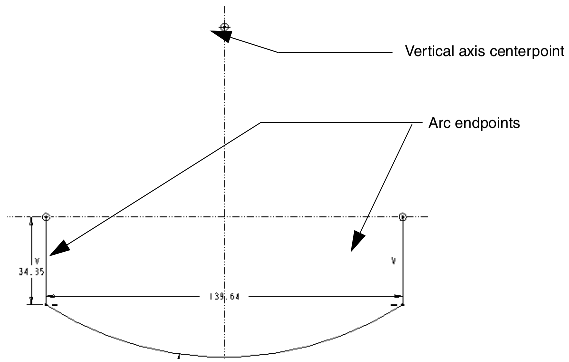


3. Click the Center and Ends tool to sketch the lower curve. First click in the top half of the vertical centerline, and then click anywhere in the lower-left quadrant, below the horizontal centerline, and move the pointer to the right. (You don't need precise dimensions. You can enter exact dimensions when the outline is finished.)

   Remember, you can click , **Edit** > **Undo** or `Ctrl+Z` multiple times to undo any mistakes.

4. As you move the pointer to the right, you will see facing constraint symbols at the start point and at the pointer, indicating that they are horizontal. Click to place the endpoint of the arc.
5. Middle-click to exit the tool. You'll see the arc, with the "weak" dimensions in gray. They will become "strong" dimensions when you enter the true values for them.

Lower half of the lens section with weak dimensions



6. Now click the Line tool in the Line flyout menu. Sketch two vertical lines, one from each arc endpoint to the horizontal centerline. Middle-click to stop sketching the first line, then click the left mouse button again to start sketching the next line. Notice the "*V*" constraint symbols, showing you the line is vertical. Middle-click to exit the Line tool. This is the lower half of the lens.

Mirror Section Geometry

Now you'll use the Mirror tool to create the identical upper section of the lens.

1. Click the Selection icon .
2. Drag a selection rectangle around the lower section. All of its lines should be highlighted as selected.
3. Click or **Edit** > **Mirror** from the main toolbar.
4. Click the horizontal centerline to place the mirror image of the bottom lens section. The complete lens section is now centered on the horizontal and vertical axes. The mirrored lines are associative–if you resize one, its mirror will resize accordingly.

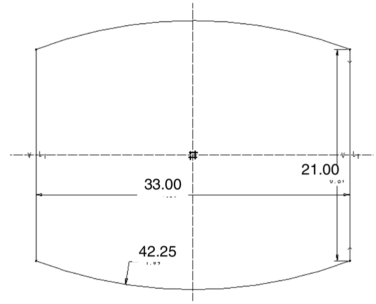
Modify Lens Dimensions

Now you will enter the true dimensions for the lens.

1. Click the Selection icon and drag a rectangle around the entire section *including all the dimensions* (you could also use **Edit** > **Select** > **All**).
2. Click the Modify Dimensions tool . The **Modify Dimensions** dialog box opens, showing value fields for all three dimensions.
3. First, clear the **Regenerate** check box. If you leave the check box checked, each dimension will regenerate as you edit it. In this case, it is faster to change all the dimensions in the section, and then regenerate the section.
4. Click one of the dimension values in the dialog box, and notice that its corresponding dimension is highlighted in the section. Enter the dimensions for the lens section: `42.25` for the arc radius, `21.00` for the height, and `33.00` for the width. Remember to press Enter after entering each value.

   Now, select the **Regenerate** check box and click . The section regenerates and the view is zoomed to the new scale.

Finished lens section with final dimensions



Save the Section

Now you'll save the lens section to a file. This is not usually necessary; the section for each feature is stored in the part. You are saving this section for future use. When you create the lens opening cut in the front cover (the final part in the series), you'll import this section to establish the outline for the cut, rather than drawing and dimensioning another section.

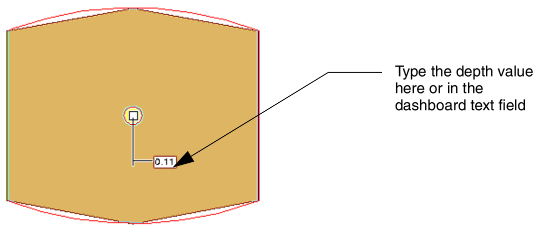
1. Click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Enter the name `lens` in the **New Name** text box.
3. Click **OK**. (The extension `.sec` is appended automatically.)

Exit Sketcher and Enter 3D Mode

You are finished defining the section. Now you'll turn the sketch into a solid feature.

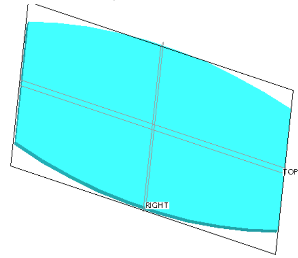
1. Click the Check icon on the Sketcher toolbar to accept the sketch. Click **Insert > Extrude**. The Extrude dashboard opens. Click the lens sketch in the graphics window to activate it. A small drag handle appears on the part axis and the depth dimension is shown. Hold down the middle mouse button to rotate the model slightly to see the handle more clearly.
2. Drag the handle and type a depth of `1.25` into the dimension box directly, or type the value in the dashboard.

Completed section, with depth handle



3. Click the Check icon in the dashboard to accept the feature. When the feature is accepted, you can see it added to the Model Tree.

The finished lens protrusion

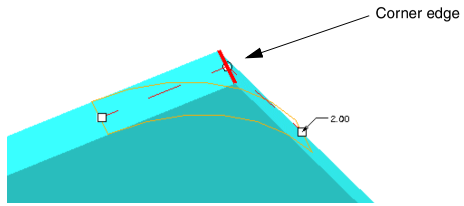


Round the Lens Corners

Now you will add a round to each of the outer edges. If you select and round one corner at a time, each round is added to the Model Tree as a separate feature. However, if you hold down the Ctrl key, select all four edges, and then add the round, the rounds are added as one feature, and all share the same degree value. This is the preferable method if you want all the corner rounds to be the same radius. You can then update them all in one edit, rather than selecting and editing each one in sequence.

1. Select the first edge, then rotate the lens to select the second one. Hold down the Ctrl key when selecting the second and additional edges, releasing it only to rotate and zoom in on the lens.
2. When all four corner edges are selected, click or **Insert > Round**. The Round dashboard opens.

Rounded corner in edit mode



3. Drag the handle to resize it or click the dimension to enter `2.00`, the correct value for the round. Middle-click or click the Check icon on the dashboard to complete the feature. Once again, the feature is added in the Model Tree.

Note
Always save your model after adding each new feature.

Add Color to the Part

When you add color, each part is more easily recognizable as a distinct object, especially in larger assemblies. If no color palette file is available, you can create and save one. Always assign colors that will make it easy to differentiate between parts of the assembly.

Click **View > Display Settings > System Colors** to assign a color to the part from the color palette.

Save and Close the Part

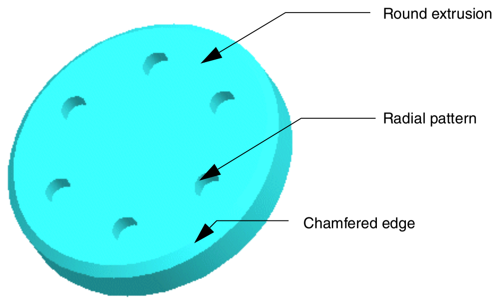
Save the part, then click **File > Close Window**. Now click **File > Erase > Not Displayed** to close `lens.prt` and erase it from active memory.

Summary

Now you've created and saved a simple part file, and you've learned some of the basic Sketcher techniques. You might try to do all of these operations again from memory in a test file, just to practice the sequence of events.

In the next part you'll learn how to place a hole feature and create a circular pattern of holes based on the original hole. You'll repeat some of the same techniques. If you get stuck, refer to the table before each part for the location of the detailed instructions.

Part 2: Earpiece



To create this part, you'll use some of the same extrusion techniques you used in the lens part. The only difference in this part is that the extrusion is round. You'll learn how to insert a hole into a solid, then learn how to use the hole to create a *pattern* of identical features from it.

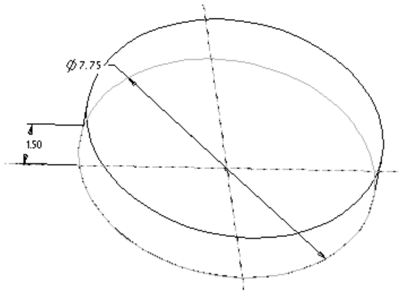
There are several types of patterns and they are extremely useful for repetitive features. This particular pattern, the radial pattern, is commonly used for bolt hole circles. All the patterned features are associated with a "parent" feature, also called the *pattern leader*. When the parent is edited, all the "children" or associated features, update accordingly.

Technique or Feature	Where Introduced
Insert Protrusion	Part 1: Lens
Chamfers	New
Holes	New
Hole Patterns	New

Create the Earpiece Protrusion

To begin, create a new part called `earpiece` using the Extrude tool. Use the following guidelines and techniques from the previous section to create the chamfered protrusion to the following specifications. Then follow the instructions to add a hole and repeat it in a radial pattern.

3D circle with strong dimensions



Guidelines:

- Use the Front datum as the sketching plane, as you did for the lens.
- Use the Circle tool in the Sketcher toolbar to draw a circle and let the pointer snap to the intersection of the horizontal and vertical reference lines.
- In Sketcher, type a diameter dimension of `7.75`. In the Extrude dashboard, type a thickness of `1.50`.
- The chamfer dimension is `0.25`. (The chamfer is a separate feature from the round extrusion, created with the Chamfer tool. Don't try to create it from within the Extrude dashboard.)

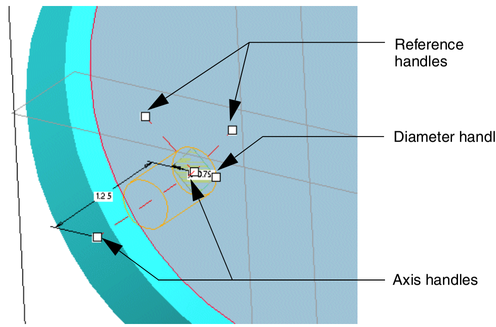
Create the First Hole

Use the Hole tool to specify the dimensions and the location of the pattern leader hole.

There are various ways to position a hole on a solid. In this example, you'll use a radial hole, which is defined by 1) a surface to lie on, 2) an axis to be offset from, and 3) a plane to use as a zero degree reference for rotating about the axis from which it is offset. You'll use the extrusion surface, the extrusion center axis and the Top datum plane as these references.

1. Click **Insert** > **Hole**. The Hole dashboard opens.
2. Click the front surface of the earpiece. The hole is placed in preview outline. Several handles extrude from the outline. Two of them define the ends of the axis line. One defines the diameter. The remaining two are the reference handles.

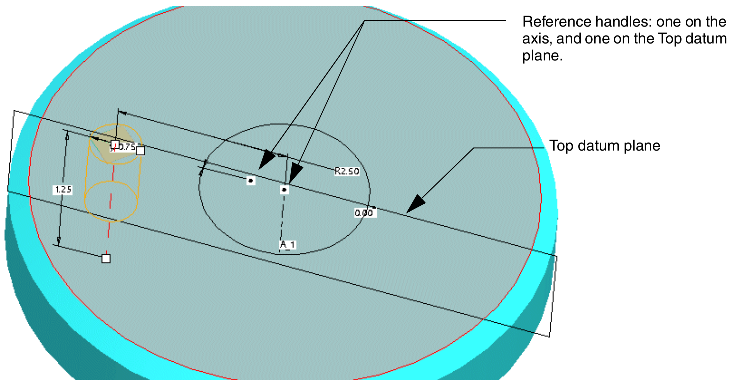
The hole in preview mode, before referencing



3. In the dashboard, you can enter the diameter, `0.75`, and the depth, `1.25`.
4. Click the Placement panel on the dashboard, and set the hole type to **Radial**. Leave the panel open.
5. To place the first radial reference, drag one of the reference handles to the extrusion's axis. (Be sure axes are displayed.) The handle should snap to the axis and show as a white square with a black dot if it is referencing the axis properly. The axis will also appear in the Placement panel as a secondary reference.

6. To place the second reference, drag the second reference handle to the Top datum plane. The datum should highlight, and the handle should snap to it and show as a dot in the square. The datum plane should appear in the Placement panel as a secondary reference.
7. With the two handles placed, type `2.50` for the axial distance value in the Placement panel. This places the hole `2.50` from the axis. Type `0` for the datum plane's angular value. This centers the hole on the datum plane.
8. Click to close the Placement panel, then click the Check icon on the dashboard to accept the feature.

The radial hole referenced



Create the Radial Pattern

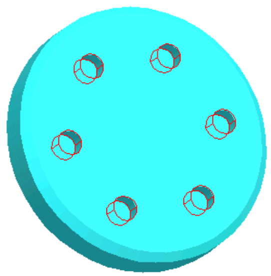
Now you'll create a radial pattern based on the first hole. It is easier to understand patterning if you think of it as repeating dimensions rather than repeating features, although it is the feature that is repeated. In the setup process for patterning you are asked to identify dimensions that indicate the direction in which you want to repeat the pattern and to specify how many instances, including the original, that you want.

1. Select the hole in the Model Tree. From the right mouse button shortcut menu, click **Pattern**. The Pattern dashboard opens. The dimensions for the hole feature are activated.

2. You need a total of six items around the hole's radial dimension, which is now set to `0`. You express the pattern as follows: "Increase the selected dimension by 60 degrees. Do it 6 times."

   On your model, double-click the `0` dimension, and type `60`. Press Enter. If you open the Dimensions panel, you will see the dimension in the **Direction 1** list, with 60 as the increment value. Close the Dimension panel.
3. Now specify how many times to perform this increment. In the text box for the first direction in the dashboard, type `6` and press Enter.
4. Click the Check icon on the dashboard to accept the feature. The pattern is added to the Model Tree in place of the original hole, which is now part of the pattern.
5. Save and close `earpiece.prt`.

Finished pattern

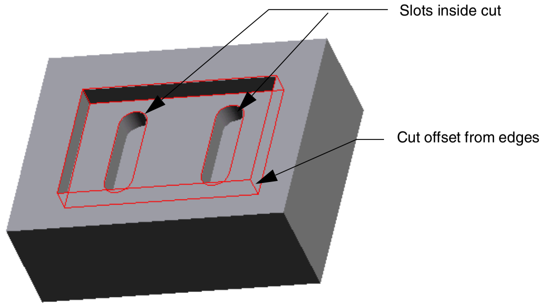


The pattern is parametric and associative, in that if you change the diameter, or any other dimension of the leader feature, the patterned features will update to the new value. If you add a feature to the leader feature, for example a round on the edge of the hole, you can pass the new feature on to the patterned holes.

Summary

You've now created the second part and have learned how to repeat selected dimensions to create a pattern as a feature. In the next exercise you'll learn some more advanced methods for applying parametric constraints in Sketcher and how to use an extrusion to specify an area of material to remove.

Part 3: Microphone



The microphone is a rectangular box with a cut and two slots extruded into it. The two slots are parametrically associated to share dimensions and position from the center. In this part, you'll learn how to use an extrusion to remove material from a solid. You'll also learn how to use geometric constraints in Sketcher not only to measure geometry accurately, but to build in associativity between features in the process.

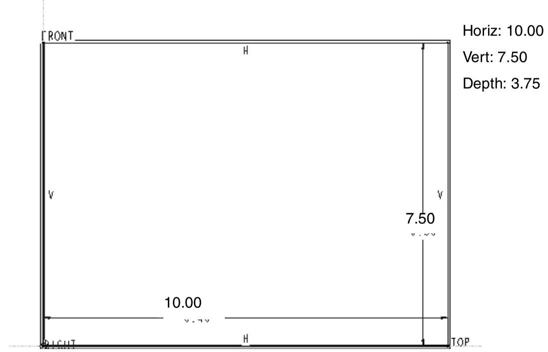
Technique or Feature	Where Introduced
Insert Extrusion	Part 1: Lens
Remove Material	New
Select by Loop	New
Place a Construction Line	New
Use Sketcher Point Constraint	New

Create the Rectangular Box

To begin, create a new part and name it `microphone`. This time, we'll use an internal sketch for the extrusion. Click **Insert > Extrude**, and then open the dashboard Placement panel and click **Define** to start Sketcher.

Using the Front datum plane as the sketching plane, create a rectangular section in the upper-right quadrant of the work area as defined by the intersecting datum panes. Dimension the section and enter the depth as shown in the next figure.

Sketcher drawing with strong dimensions

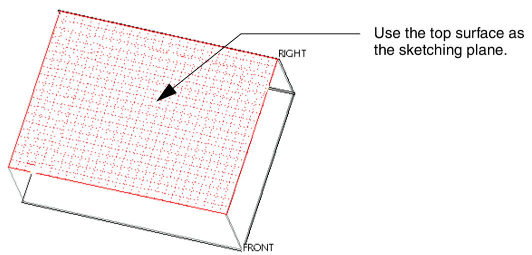


Create the First Cut

A quick way to specify an area from which to remove material is to use the outer edge of the first section to define an offset for a new section inside it.

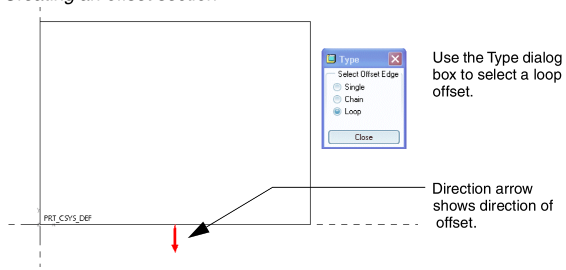
1. Click **Insert** > **Extrude** from the main menu. Select the top surface of the first extrusion as the sketching plane.

3D extrusion



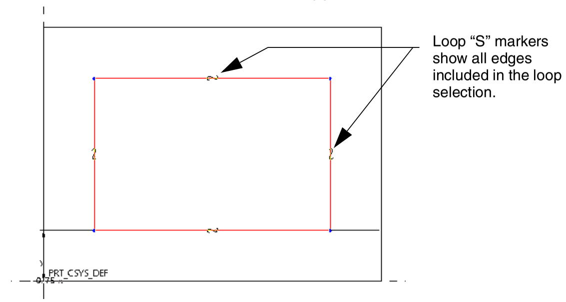
2. Click the Offset Entity from an Edge tool on the Sketcher toolbar. The **Type** dialog box opens, showing different types of offset references.
3. Click **Loop** in the **Type** dialog box. Be sure not to close the dialog box.
4. Select one of the edges of the section to loop all the edges as the reference entity. A directional arrow points outward from the selected edge. You are prompted for the offset value.

Creating an offset section



5. Because you want to define an offset inside the loop, type an offset value of `-1.50`.

Cut section after the offset value is applied



6. Close the **Type** dialog box and click the Check icon on the Sketcher toolbar to accept the section and return to the dashboard.
7. In the dashboard, set the properties for a cut:
   a. Click the Remove Material icon.
   b. Check the extrusion direction, and click the direction arrow if the extrusion extends out of the solid.
   c. Set the depth to `0.75`.
8. Click the Check icon on the dashboard to accept the feature.

You have made the first cut in the extrusion. The feature is added to the Model Tree. You’ll now make two mirrored slots in the bottom of this cut.

Create the Slots

The design intent for the slots is to have them identical, centered horizontally and vertically, and offset equally from the microphone center. They must also be associated so that an edit to one is reflected in the other. There are many ways to sketch these shapes—this procedure is just one of them. Basically, you will use construction lines to mark the center of the rectangle. You will then create the slot on the left side using the same mirror technique you used to create the lens. When the first slot is created, you will mirror it to the opposite side.

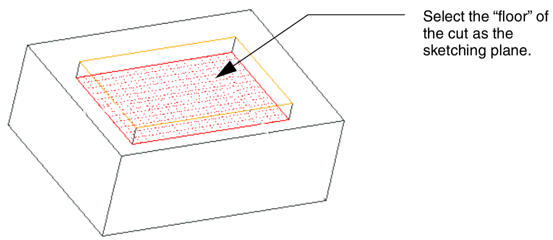
You must first locate the center of the section you created. You could simply calculate half the width and half the height, and place the point at the intersection, but the point would be determined by exact measurements and would not update if the height or the width dimensions changed. You want to locate the center point in such a way that it is parametrically associated with the boundaries of the rectangle section. Here is how to use Sketcher constraints to do this.

Define the Horizontal and Vertical Centers

1. Click **Insert** > **Extrude**. Enter Sketcher from the dashboard Placement panel using the bottom surface of the first cut as the sketching plane.

2. Click **Sketch > References** to open the **References** dialog box. Click on each of the outer edges of the first extrusion to add them as references. Highlighted reference lines appear on all sides of the rectangle. Close the **References** dialog box.

Setting up the slot section



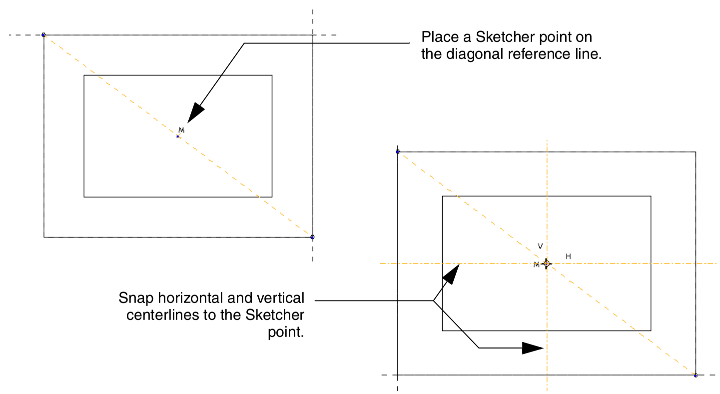
3. Now use the Line tool on the Sketcher toolbar to place a line from corner to corner diagonally across the outer rectangle. The line ends will snap to the corners. Middle-click to exit the tool when the line is finished.

4. Because the line will not be part of the section, select it and choose **Construction** from the right mouse button shortcut menu. This toggles the line into a construction line, used for reference only.

5. Click the Create Point tool on the Sketcher toolbar to place a Sketcher point on the construction line. As you move the point to the center of the line, an M symbol appears when it reaches the center. This is also the center of the rectangle. With the M symbols showing, click to place the point. The center is now permanently located because the point is constrained to the center of the line.

6. Click the Centerline tool , and snap horizontal and vertical centerlines on the center point. The first cut should now be bisected in both directions. You have now found the center of the rectangle and used it to define the horizontal and vertical center. Because the Sketcher reference point is constrained to be in the middle of the diagonal line, it will update its position if either the width or the height of the first protrusion changes.

Using a Sketcher point to locate the center of the extrusion

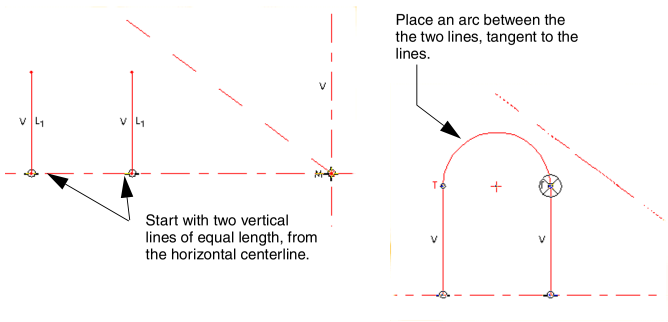


Sketch the Slot Section

Create the top half of the new cut above the horizontal centerline, and then mirror it to create the bottom half, as follows:

1. Zoom in to the left half of the cut. Draw two vertical, parallel lines up from the horizontal axis. Notice the constraint symbols for vertical, *V*, and equal length, *L1*.

Creating the slot section



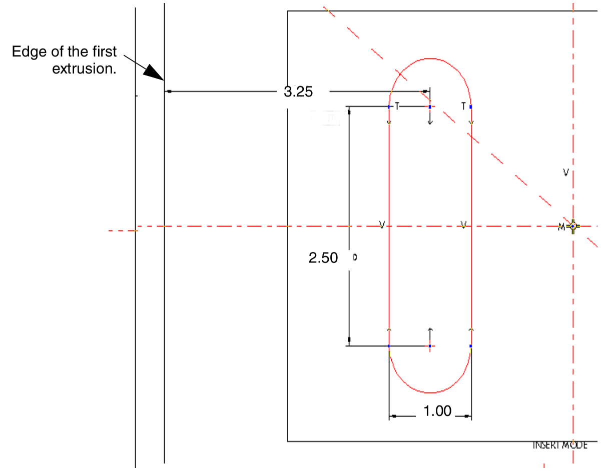
2. Click the Arc tool on the Sketcher toolbar and join the top points with an arc. Place the arc when the two *T* symbols appear, indicating tangency to the lines. (See the previous figure.)

3. Click the Selection arrow on the Sketcher toolbar and drag a selection rectangle around all the lines and the arc.

4. Click the Mirror tool on the Sketcher toolbar. Click the horizontal centerline. The lines are mirrored to form the first cut.

5. Use the Add or Modify Dimension tools to add the final dimensions to the section as shown in the next figure. The arc centers are `3.25` from the outer edge of the first extrusion, and `2.50` apart. The arc is `1.00` in diameter.

Final dimensions for the first slot



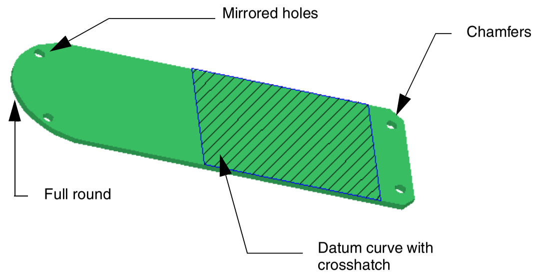
6. Hold down Ctrl to multiple-select the section. Click the Mirror tool again, and then click the vertical axis. The slot is copied across the vertical axis.

7. Click to finish the section. You are returned to the dashboard. As you did with the first cut, click the Remove Material icon and set the extrusion direction into the solid. Set the depth to `0.75`. Click the Check icon on the dashboard to accept the feature. Save the completed part.

Summary

To demonstrate the associativity of the slots, select the internal sketch of the slot cut feature in the Model Tree and choose **Edit** from the shortcut menu. The dimensions are shown for the original slot. If you enter a different dimension for the height, 3.00 for example, you'll see both slots update to the new height.

Part 4: PC Board



The printed circuit board is another simple part with a full round, two chamfers, and four holes. You will learn how to create two holes in Sketcher as one feature, and then mirror the feature in 3D mode. You will create a datum curve as a reference for the placement of the keypad.

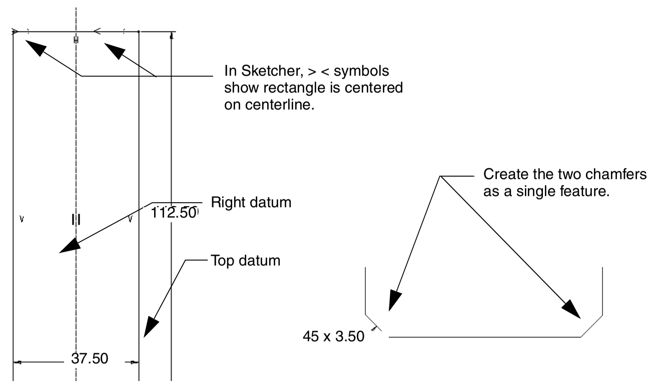
Technique or Feature	Where Introduced
Insert Extrusion	Part 1: Lens
Full Round Feature	New
Chamfer Feature	Part 1: Lens
Holes	Part 2: Earpiece
Copy and Mirror in 3D	New
Create a Sketcher Datum Curve	New

Create the PC Board Protrusion

Create a part called `pc_board`. Click **Insert > Extrude** and sketch the board as described below:

- Use the Front datum plane as the sketching plane. In Sketcher, place a centerline down the vertical datum so that you can use Sketcher constraints to center the rectangle.
- The Top datum plane is the horizontal reference. Draw a rectangle so that the bottom is on the Top datum plane.
- When you exit Sketcher, set the depth to `1.50`.

Dimensions for PC board sketch



Create the Chamfers and Round

Now, add the round and chamfers as features. Remember, you could create the round and chamfers in Sketcher, but it is better to create as many aspects of a part as possible as discrete, adjustable features.

Add the Edge Chamfers

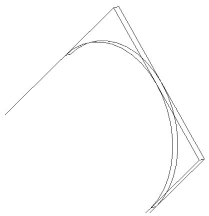
1. Hold down Ctrl to multiple select the two lower thin edges. Click **Insert > Chamfer > Edge Chamfer**. The Chamfer dashboard opens.
2. In the Chamfer list, click **45 X D**. This gives you a 45-degree angle chamfer with an assigned dimension.
3. Enter `3.50` for the dimension in the **D** text box.
4. Click the Check icon ✔ on the dashboard to accept the feature and exit the dashboard.

Add the Full Round

To round the corners you will use a full round rather than two 90-degree rounds. A full round will update with any changes to the width of the parent protrusion.

1. Click **Insert > Round**. The Round dashboard opens.
2. To select the edge opposite the chamfers for the round, rotate the model and select the edge.
3. Rotate the model, hold down Ctrl, and select the second edge. Both edges should be highlighted.
4. Click **Sets** in the Round dashboard. Both selections appear as a single set in the Placement panel.
5. Click the **Full round** button to add the full round to the PC board. Click the Check icon ✔ on the dashboard to accept the feature.

Rounding the top



Place the Holes

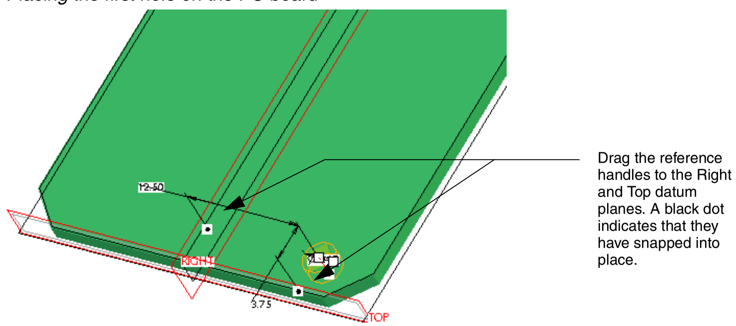
1. Use **Insert** > **Hole** to create the hole features as shown on the PC board model for this part. When the Hole dashboard opens, enter the settings from the table below. For the primary reference, click the surface near where you want to place the first hole. This surface appears in the Placement panel's primary collector.

Values for the first hole in the PC board

Hole Type	Straight	
Diameter	3.25	
Depth	Through All	
Primary reference	Surface (XXX)	Type: Linear
Secondary reference 1	Right datum	Offset 12.50
Secondary reference 2	Top datum	Offset 3.75

2. Drag the reference handles to the Right and Top datum planes as shown in the next figure. Use the Placement panel to enter the exact distance values, or double-click a dimension to enter it directly on the model.
3. Set the depth to **Through All** from the depth settings list on the dashboard.
4. Accept the feature.

Placing the first hole on the PC board



Create the Second Hole

Click **Insert > Hole** again to place the second hole. Use the same datum planes, Top and Right, and type a distance of `4.15` from the Top datum, and `0.4` as the distance from the Right datum. Now enter the following references in the Placement panel collectors.

Values for the second hole in the PC Board

Hole Type	Straight	
Diameter	3.25	
Depth	Through All	
Primary reference	Surface (XXX)	Type: Linear
Secondary reference 1	Right datum	Offset 10.00
Secondary reference 2	Top datum	Offset 103.75

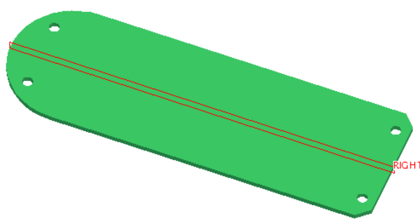
Copy and Mirror the Holes

Now that you have two holes on one side, you can quickly copy and mirror them onto the other side of the PC board.

1. From the Model Tree, select the two hole features you just created. Click Mirror on the main toolbar. The Mirror dashboard appears.
2. Select the Right datum plane as the mirror plane and accept the feature. The new copied group is added to the Model Tree.

   Note
   The mirrored copy is dependent on the first hole by default.

Mirrored holes

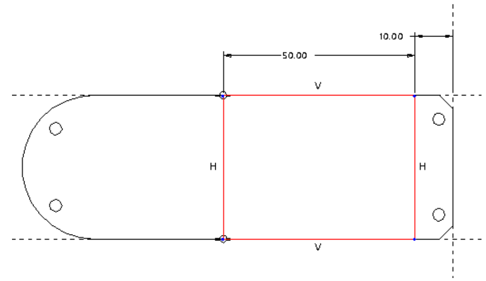


Create a Datum Curve for the Keypad Reference

A datum curve is a type of datum that is created by defining a series of points connected by an arc. A sketched datum curve is similar, except it is actually drawn onto the model using the section drawing tools in Sketcher. It does not define geometry as a section does, but it can be added for any number of reasons: as a reference for assembly, other features, or as a notational device. Here you'll use a flat datum curve to show the PC board designer the intended position of the keypad.

1. Click **Insert > Model Datum > Sketch**. The **Sketch** dialog box opens.
2. Select the front surface of the PC board as the sketching plane, and then click **Sketch** in the dialog box. Sketcher opens.
3. Click **Sketch > References** to open the **References** dialog box. Select the side edges as references, and then close the dialog box.
4. Draw a rectangle across the middle of the PC board section and dimension it using the dimensions from the next figure (height = `50.00`, `10.00` from the Top datum).

Dimensions for the datum curve feature

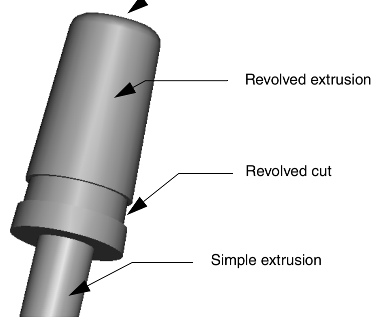


5. Click the Check icon to complete the section. The datum curve is added as a simple outline. To crosshatch it, right-click the datum curve in the Model Tree and choose **Edit Definition** from the shortcut menu. Click **Sketch > Sketch Setup** to open the **Sketch** dialog box. Click the **Properties** tab and select the **X-hatch** checkbox to turn on crosshatching. Accept the section. The datum curve is now crosshatched.
6. Save and close `pc_board.prt`.

Summary

So far, you've learned to create simple extrusions to define solids and cuts, and to create patterns. You have created a datum curve as a reference for the placement of the keypad. As you create the antenna, you'll use a variation of the extrusion technique, a revolved protrusion, and learn some new Sketcher techniques.

Part 5: Antenna



A sketched 2D profile that is revolved around an axis, called a revolved extrusion, is used to create the antenna. The tip of the antenna is created first and a revolved cut is then added to it. The antenna shaft is created with another round extrusion from the base of the tip.

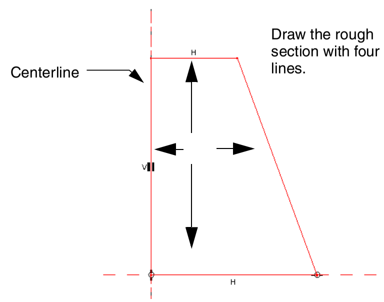
Technique or Feature	Where Introduced
Revolved Extrusion	New
Round Extrusion	Part 1: Lens
Revolved Cut	New
Insert Extrusion	Part 1: Lens

Sketch the Revolved Protrusion

Create a new part file and call it `antenna`.

1. Select **Insert** > **Revolve**. Enter Sketcher through the dashboard Placement panel, and use the Front datum as the sketching plane.
2. In Sketcher, place a centerline along the vertical axis.
3. Use the Line tool to sketch the polygon as shown in the next figure.

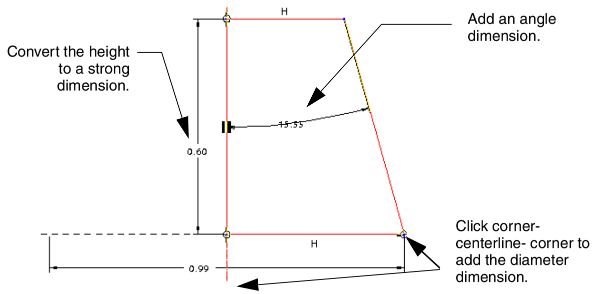
Rough sketch of the antenna tip



Creo Elements/Pro automatically generates weak dimensions for the section. These dimensions define the revolved section for the extrusion. However, you may devise another method that better meets your needs.

4. Use the Add Dimension tool on the Sketcher toolbar to insert a dimension for the angle of the outer wall. Click each side of the angle and then middle-click between the sides to place the dimension. The weak dimension value for the top of the shape disappears, because that dimension is now determined by the angle. Middle-click to exit the tool.

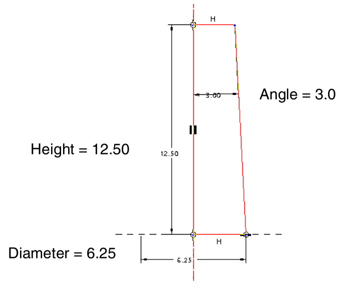
Adding strong dimensions



5. Select, and then right-click the weak height dimension and choose **Strong** from the shortcut menu. The weak dimension is converted to a strong dimension, which remains when other dimensions or constraints are added.
6. Select the Add Dimension tool to add the diameter dimension. Click the outer corner of the section, the centerline *below the section*, and the outer corner of the section again. (See the previous figure.) Middle-click to place the dimension. This action doubles the radius of the revolve and enters it as the diameter of the base. Middle-click again to exit the tool.

   Every parametric relationship that determines the shape of the revolved extrusion has now been defined: height, base diameter, and taper. This process is known as solid definition.

Final dimensions for the antenna tip extrusion



7. Now add the real values for all the dimensions. Click **Edit** > **Select > All** to select everything in the sketch.

8. Click the Modify Dimensions tool . The **Modify Dimensions** dialog box opens. Type the values for all the dimensions. Remember, you can select a dimension in Sketcher to highlight it in the dialog box.

9. Now click on the Sketcher toolbar to finish the sketch. You are brought back to the dashboard, with the revolution completed to 360 degrees. Drag the degree handle to see how the revolved solid works. It can be set to any angle up to 360. Set the revolve to 360 and accept the feature.

Add a Round to the Top

Select the top edge of the protrusion and use the right mouse button shortcut menu to add a round of `0.50`.

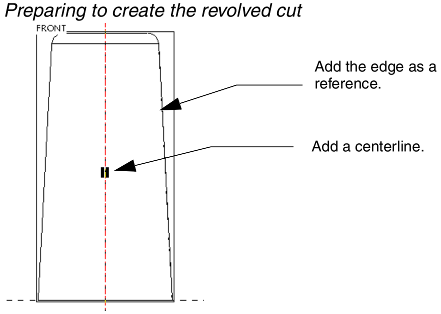
Add the Revolved Cut

This section covers more techniques for using constraints in Sketcher. You'll add a revolved cut to the revolved surface. This cut could have been included in the revolved protrusion's profile, but best practice is to keep as many features as possible as separate entities.

1. To start, use **Insert** > **Revolve** and enter Sketcher. Use the Front datum as the sketching plane. Click **Sketch**, and then add the angled outline of the antenna tip as a reference (click **Sketch > References** to open the **References** dialog box). Click **OK** and close the dialog box.

2. Add a centerline along the axis of the protrusion.

Preparing to create the revolved cut

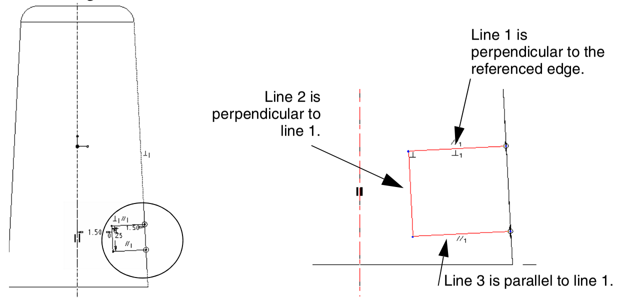


3. Sketch the first line (line 1 in the next figure) into the solid from the revolved surface. Notice that the constraint defaults to *H*, or horizontal to the top. Since the line should be perpendicular to the revolved surface, quickly right-click to disable the default constraint while the *H* is visible. (Don't hold down the button long enough to bring up the shortcut menu—just click the button.) You'll see a line drawn through the default constraint symbol, indicating the default snapping constraint is turned off.

   You'll now be able to draw the line to an angle perpendicular to the revolved surface. (You'll see the perpendicular symbol, an inverted *T*, when the line is in position.)

4. Draw the remaining lines, disabling the default constraints as needed. Lines 1 and 3 in the next figure are perpendicular to the revolved edge. Line 2 is parallel to it.

Sketching the cut section



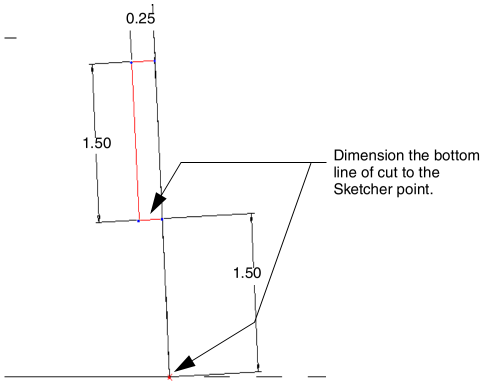
5. Use the Sketcher Point tool to place a Sketcher point at the intersection of the tip edge and the base. When the rough section is complete, a dimension is needed to locate it in reference to the tip base. Because the tip base and the bottom of the cut are not parallel, a constant dimension cannot be entered between them.

6. Use the Add Dimensions tool to add the dimension between the Sketcher point and the bottom line of the cut. For the bottom line, select the line itself, not its intersection with the tip edge. When the dimension is entered properly, the witness lines for the dimension are parallel to the bottom of the cut, not the tip base. Replace the weak dimensions with the strong ones shown in the next figure, giving them the values indicated.

   Tip

   You can toggle existing weak dimensions into strong dimensions; just select the dimension and press Ctrl+t.

Finished dimensions for the revolved cut



7. When the sketch is finished, accept the section and return to the dashboard. Enter `360` for the revolve angle. Make sure that Remove Material has been selected. Accept the feature and return to 3D mode.

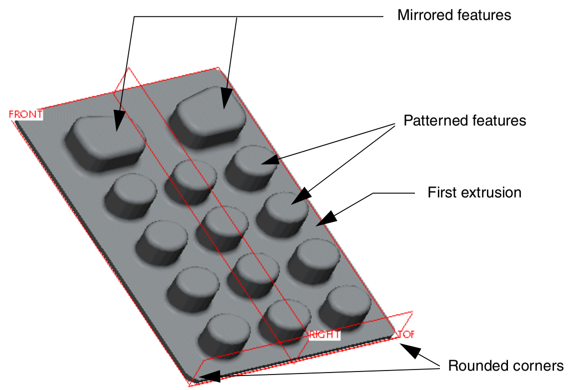
Make the Shaft Extrusion

The shaft is a single cylindrical extrusion centered on the revolve axis. To add it, click **Insert** > **Extrude**. Enter Sketcher from the dashboard and select the bottom of the tip extrusion as the sketching plane. Use the Circle tool in Sketcher to create a circular section, letting the center snap to the center of the tip feature. The circle diameter is `3.125`. Enter a length of `75` in the dashboard. Make sure the extrusion points away from the tip base. If it does not, click the Direction button to change it. Accept the feature and save and close the completed part.

Summary

So far, the five cell phone parts contain fewer than a dozen features. You should now have a good understanding of Sketcher operations, especially how to define the shape of solids by building in geometric constraints. The associativity among dimensions in the antenna tip illustrated this process. Remember, best practice is to create features as separate entities. Each feature then appears in the Model Tree so changes can be easily made to it.

Part 6: Keypad



In the earpiece part, you defined a radial pattern, one that created new instances of a feature at intervals around a radius. The keypad also uses a pattern, but this one defines the number of instances in the x and y directions.

Like the PC board, the keypad uses a copy and mirror operation for the upper two large buttons. This part also introduces relations, or formulas used in place of dimension values. You'll learn how to refer to dimensions by their ID numbers rather than by their values, and how to reference dimensions by their ID numbers within a formula. The relation technique is useful in Sketcher, as well as in Part and Assembly modes, to ensure that features and parts are properly associated.

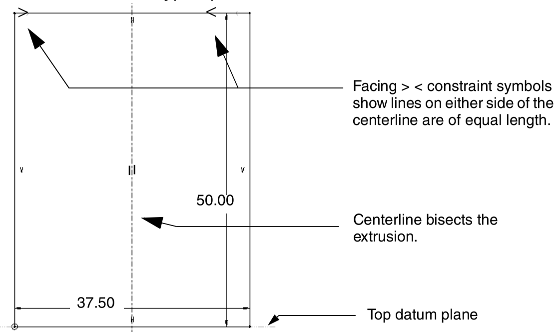
Technique or Feature	Where Introduced
Insert Extrusion	Part 1: Lens
Round Corners	Part 1: Lens
Pattern Feature in Two Directions	New
Use Relation as a Dimension Value	New

Sketch the Keypad Protrusion

To start, create a new part called `keypad`. Use the following guidelines to create the first extrusion with the dimensions shown in the next figure:

1. Start Sketcher from the Extrude dashboard, and use the Front datum plane as the sketching plane. Add a vertical centerline as a point of reference for the keypad rectangle constraints. The centerline will bisect the pad vertically.
2. Start in the upper-left corner and drag a rectangle across the centerline, down to the right. Stop at the Top datum. Remember, the facing arrows tell you that the lines on either side centerline are the same length.

Dimensions for the keypad protrusion



3. Select the weak dimensions and make the height `50` and the width `37.50`, as shown in the previous figure.
4. Click the Check icon on the Sketcher toolbar to complete the sketch. Enter `0.75` as the depth value, and accept the feature.

Round the Corners

1. To round the corners, rotate the model while zooming in on the corners, and use Ctrl + click to multiple-select all of the 0.75 edges. You can set the filter to **Geometry** to select the edges directly, or use the Smart filter to cycle through objects under the pointer. When you have selected the first edge, you don't have to cycle through to select the next edge. The Smart filter puts similar objects at the start of the selection cycle based on your previous selection. Just hold down the Ctrl key and click the remaining edges to add them to the selection.

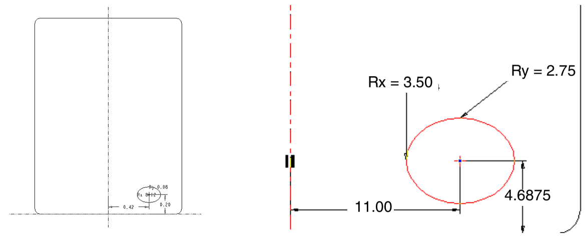
2. When all of the edges are selected, right-click and choose **Round Edges** from the shortcut menu. In the dashboard, set the round value to `1.50`. When you multiple-select and round this way, all the corners are associated and an edit to one of them affects all the others.

Add the First Button Feature

Now you'll add the first button feature. This will serve as the leader for the button pattern.

1. Click **Insert > Extrude** to create an elliptical extrusion on the first feature, as shown in the next figure. Use the front surface of the first extrusion as the sketching plane. In Sketcher, choose the Ellipse tool on the Circle flyout menu to sketch the basic ellipse. Type the distance and radius values as shown in the next figure: `11.00` from the centerline and `4.6875` from the Top datum plane. Type the values for the button radii as shown in the next figure.

Location and dimensions for the first button



2. When you accept the section and return to the dashboard, give the button a depth value of `5.50`. Accept the feature.

Pattern the Button

Now you'll pattern the button feature. Patterning is easier to understand if you think of it as repeating dimensions rather than repeating features, although it is the feature that gets repeated in a pattern. You define the dimensions to be repeated in the Pattern dashboard, indicate the direction in which they will be repeated, and enter the number of copies, including the original, to be made.

1. Select the button extrusion in the Model Tree. Right-click and select **Pattern** from the shortcut menu. The Pattern dashboard opens, and you are prompted *"Select dimensions to vary in the first direction."* Notice that the collector for the first direction is active.

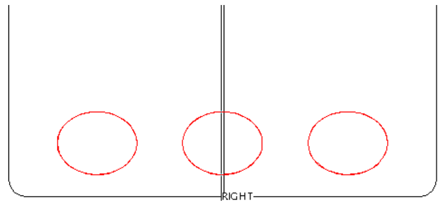
2. Use the x direction as the first direction and create two more buttons to the left of the original; one centered on the first extrusion, and one `11.00` to the left of the center:

   a. Select the `11.00` dimension. Enter `-11.00` in the dimension box and press Enter. A positive value would have sent the pattern to the right, away from the referenced dimension.

   b. In the text box to the left of the dimension collector, enter `3` for the number of instances, including the original, to create in the pattern.

   c. Click the Check icon ✔ on the dashboard to accept the pattern and check whether all values are entered correctly. The first part of the pattern should be added as shown in the next figure.

The first part of the button pattern: the X direction

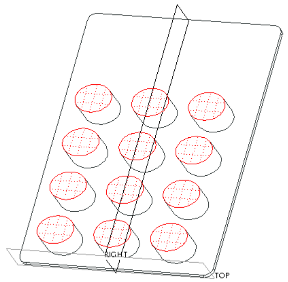


3. Now you'll go back and enter the Y direction. (You usually enter both the X and Y directions at the same time. This step is simply to check the pattern setup.) Right-click the pattern in the Model Tree and choose **Edit Definition** from the shortcut menu. The Pattern dashboard opens to the existing pattern.

4. Open the Dimensions panel. Click the second direction collector. It should become yellow and show **Select items**.

5. Select the vertical dimension. Enter `8.75` for the increment and `4` as the number of instances. Accept the feature.

The finished pattern

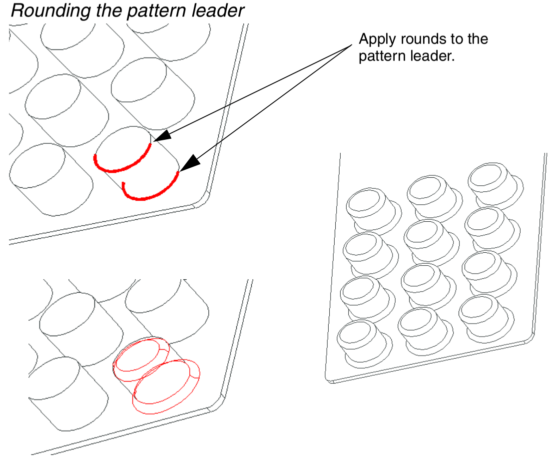


Apply Rounds to the Pattern

1. Now use Ctrl + click to select the top and bottom edges of the first button, the pattern leader, and apply a round of `0.75` to them. When the rounds are complete, they are added as one feature to the Model Tree.

2. Right-click the new round in the Model Tree, and select **Pattern** from the shortcut menu. Accept the feature. The rounds are applied to the rest of the pattern.

Rounding the pattern leader



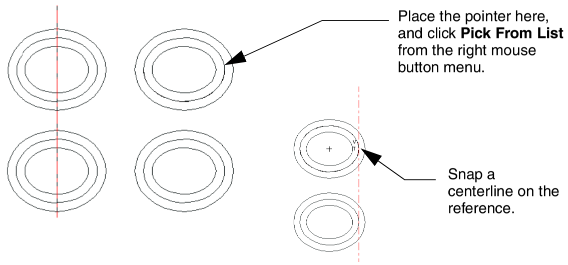
Extrude the Large Button

1. Click **Insert** > **Extrude** and use the front surface as the sketching plane.

2. Open the **References** dialog box. Select the edge of the upper-right button as a reference, as shown in the next figure. You are actually referencing the vertical surface of the button, not the edges of the rounds. To select the correct entity, place the pointer over the button edge and select **Pick From List** from the shortcut menu. A list box opens, listing all entities in the vicinity under the pointer.

3. From the list box, select any one of the surfaces listed. The edge around the entire surface will highlight, and the center of the ellipse is shown. When the surface is selected, it is added to the list in the **References** dialog box, and its color changes to orange to indicate that it is a reference. Close the dialog box.

Adding a reference for the first large button

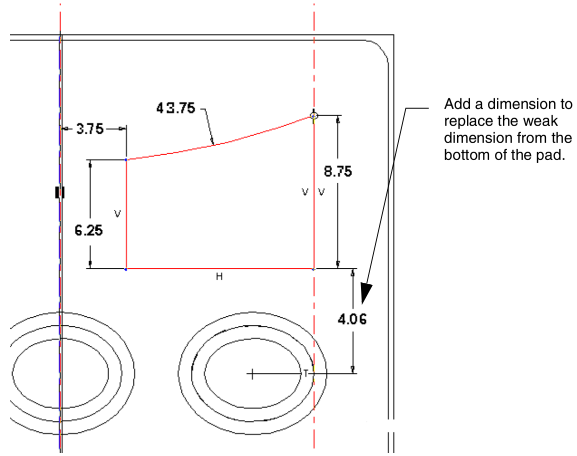


4. Click the Centerline tool, and let the centerline snap to the referenced edge, as shown above. You'll see a *T* at the intersection indicating tangency between the edge and the centerline. Place another centerline along the vertical axis of the first extrusion.

Sketch the Large Button Section

Now you can add the section for the large button. The section is drawn using three straight lines and an arc closing the top. Set the strong dimensions as shown below.

Strong dimensions for the large button



1. Use the Add Dimensions tool to place a dimension between the bottom line of the large button and the center of the top right button of the pattern. For now, accept the weak value. This is the dimension you will "drive" with a relational formula.

2. Accept the section. Check whether the section is extruding in the proper direction and correct it if necessary.

3. For the depth, click **To Selected** from the depth settings list on the dashboard. You are prompted to select an existing surface to use as a reference for the button height.

4. Select the top of one of the small buttons. The height of the large buttons will now be dependent on the height of the smaller ones. Accept the feature.

Establish Distance Relations for the Buttons

The design intent is to keep the large buttons vertically spaced to the lower pattern in the same proportion as the lower pattern buttons are vertically spaced to each other. You will use a relational formula to keep the proportion exact.

In this procedure you'll also learn how to reference dimensions by their ID numbers (for example, d56) rather than their values. To toggle the display between dimension values and ID numbers, use **Info** > **Switch Dimensions**. (Your dimension ID numbers may be different from the ones shown here.)

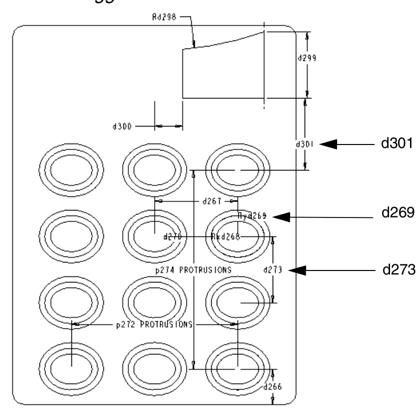
The dimension ID has a prefix that identifies its dimension type. A dimension in Sketcher will have an *s* in front of the *d*, noting it as a Sketcher dimension, for example, sd44. The dimensions for the elliptical buttons use the letters *Rx* and *Ry* for radius x-direction and radius y-direction. You don't need the extra prefix letters when entering the dimension IDs in relations. The dimension Ryd8 would just be entered as `d8`.

The relation includes the dimension between the vertical centers of two adjacent small buttons (d273) and the button y-direction radius (d269) as shown in the next figure. It stipulates that the distance between the bottom edge of the large button and the center of the small button below it (d301) is the same as the distance between the edge of a small button and the center of the next one in the vertical direction.

That distance is the distance between two button centers, minus the radius of one button. So the expression you'll enter is

> *Distance between large button bottom edge and center of next small button=*
> *small button vertical center to center,*
> *minus small button y-radius,*
> *or:*
>
> d301=d273-d269. (You'll use your equivalent dimension IDs.)

Dimensions toggled to show dimension IDs



1. To enter the relation, click **Tools** > **Relations**. The **Relations** dialog box opens.
2. Click the large button extrusion and then the pattern. The dimensions should show in symbol form. If they are not in symbol form, click the Toggle Symbol button in the toolbar to switch them.
3. Using the above figure as a guide, and substituting the symbol names as they appear on your model, type the following into the **Relations** dialog box: `d301=d273-d269`.
4. Click **OK** in the dialog box. The formula now determines the value of d301.

   To see the effect of the relation:

   a. Select the button pattern in the Model Tree and choose **Edit** from the shortcut menu. The dimensions for the pattern are activated.

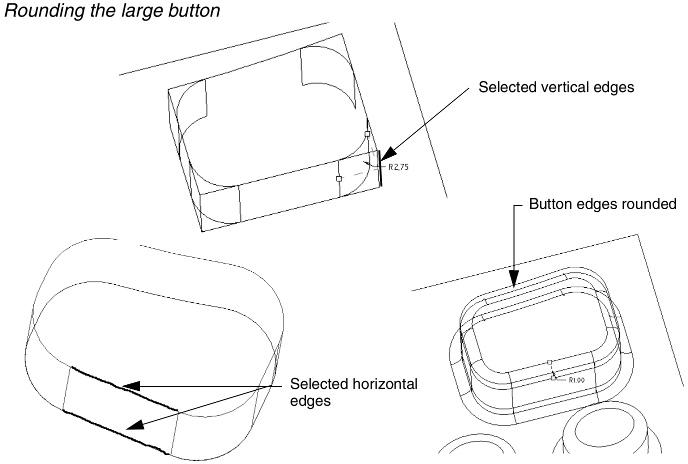
   b. Select the dimension for the distance between the small button centers and change it to a larger number.

   c. Click **Edit > Regenerate**. You should see the large button move in proportion to the small buttons. Repeat the process to return the part to its original dimensions.

Round the Large Button Edges

Multiple-select the four vertical edges (corners) of the large button extrusion. From the shortcut menu, add a round of `2.75`. Open the **Sets** panel in the dashboard and click **New set**. Multiple-select part of one edge from the top and part of one edge from the bottom of the button, and then apply a round of `1.00`, which is automatically applied to the entire edge.

Rounding the large button



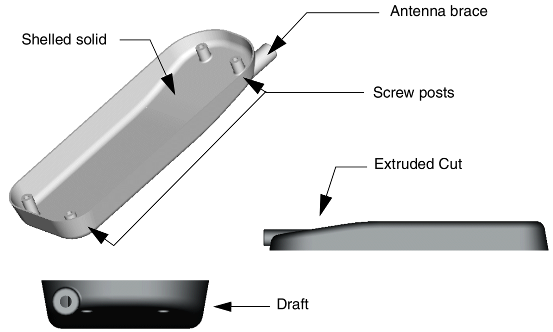
Mirror the Large Button Section

1. In the Model Tree, select the large button feature, including the rounds.
2. Select Edit > Mirror from the main menu or the Mirror icon on the toolbar.
3. Select the Right datum plane for the mirror. Accept the feature. A mirrored copy of the selected features is added opposite the existing button. Save and close the part.

Summary

A relatively complex part using combined patterns and 3D mirroring is now complete. You have also learned how to reference dimensions by ID numbers in relational formulas.

Part 7: Back Cover



The back cover part revisits several basic techniques you've used already and introduces some new ones: adding a draft angle and shelling a solid. Shelling hollows out a solid to a given wall thickness. You'll learn how to create a new datum plane offset from the solid, and then use this technique to create the antenna brace and the screw posts on the floor of the shell.

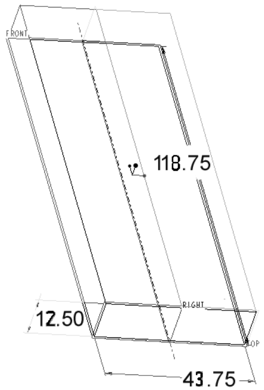
Technique or Feature	Where Introduced
Insert Extrusion	Part 1: Lens
Round Feature	Part 1: Lens; Part 5: Antenna
Extruded Cut: Two Sides	New
Draft Feature	New
Shell Feature	New
Make Datum on the Fly	New
Hole Features	Part 2: Earpiece
Copy and Mirror in 3D	Part 6: Keypad
Round Feature Using Edge Chain	New

Create the Basic Extrusion

In a new part called `back_cover`, create the basic extrusion according to the dimensions in the next figure. Use the Front datum plane as the sketching plane, and add a centerline down the vertical axis, as with the keypad.

Sketch a rectangle that is centered on the centerline and rests on the Top datum plane. Dimensions are `118.75` (height) and `43.75` (width). In the dashboard, type a value of `12.50` for the depth.

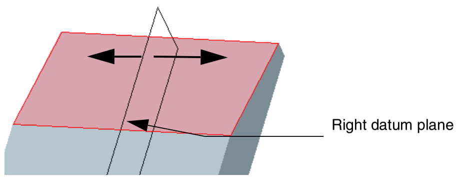
Dimensions for the back cover extrusion



Create the First Cut

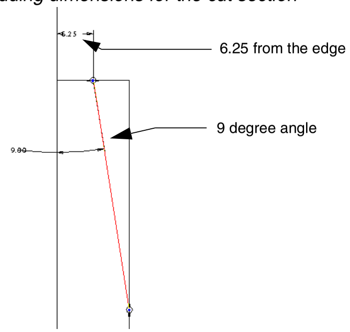
Now you'll remove an angled slice of the solid to taper the back side of the phone and extrude the cut in two directions from a central point.

A cut feature can extrude from both sides of the sketching plane



1. Click **Insert > Extrude**. Use the Right datum plane as the sketching plane. Make sure the sketch orients in the top right quadrant. If not, click **Flip** in the **Sketch** dialog box.
2. Add the top and right edges as references. Sketch the cut with a single line as shown in the next figure.

Adding dimensions for the cut section

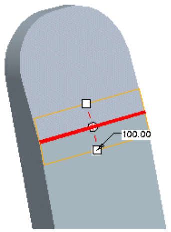


3. Enter the exact dimension values: `6.25` for the distance from the edge and `9` for the degree of angle.
4. Accept the sketch. Now set up the rest of the cut properties in the dashboard. As long as you can see the extrusion direction arrow, you don't have to move the model in any way:
   a. Click the Remove Material icon to make the extrusion a cut.
   b. Click the Direction icon if necessary so the direction arrow points away from the solid.
   c. In the Options panel, set the depth for both side one and side two to **To Next**.
5. Click the dashboard Preview icon and rotate the model to see the cut. Click the Check icon in the dashboard to accept the feature.

Round the Corners

1. Select the two bottom corner edges. Right-click, select **Round Edges** from the shortcut menu, and apply a round of `12.50`. In the Sets panel, click **New set**. Select the two top corner edges and apply a round of `18.75`.
2. Apply a round to the juncture between the extrusion and the cut. Enter `180` as the value. Accept the feature and save the part.

Rounding the cut edge

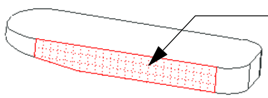


Add the Draft

The Draft feature tapers the back cover by 10 degrees on all sides, from the front to the back. To define a draft, you select the surface to which to apply the draft, and then specify a hinge plane, a draft direction, and a draft angle.

1. Select one segment of the surface to be drafted. Orient the model as shown in the next figure, and then use the Smart filter to select the surface segment. By default, Creo Elements/Pro assumes that you want to draft this surface and the ones adjoining it.

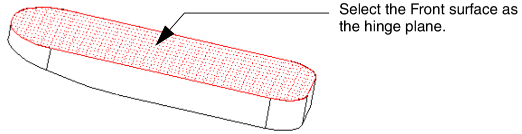
Applying the draft feature



2. Click **Insert > Draft**. The Draft dashboard opens. Open the References panel. The **Draft Surfaces** collector is active.

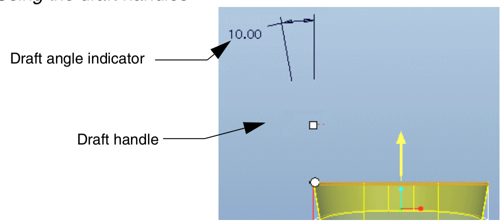
3. Click the front surface of the cover as the hinge reference. The edge of the surface is highlighted, and the draft angle handles appear. The draft direction arrow should be pointing away from the solid.

Using the front surface as the draft neutral plane

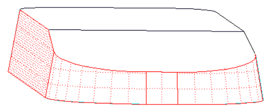


4. Drag the handle to 10 degrees, or enter `10` for the depth box.
5. Click Preview to see the finished feature, and then click the Check icon ✔ to accept the feature and return to the work area.

Using the draft handles



The completed draft

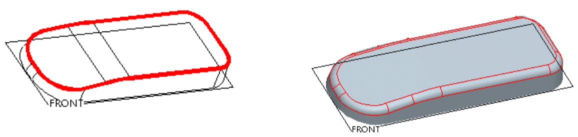


Round the Back Edges

Because there are 10 sections of the back edge to select, it is faster to use an *edge chain* to define the edge to round. Because the back edge forms a continuous line, you can use the Shift key to select a chain of edge sections by selecting two of them.

1. With the Smart filter on, click an edge section until it highlights in bold. (Or set the filter to Geometry and select the edges directly.)
2. Hold down the Shift key and click another section of the same edge. All the adjoining edge sections are automatically selected in an edge chain, as shown in the next figure.

Rounded Edges: Edge Chain Selection



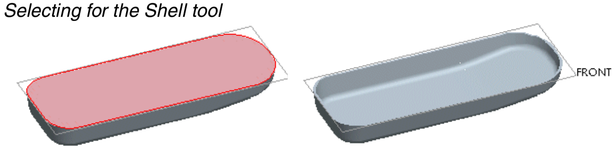
3. Select **Round Edges** from the right mouse button menu and enter `3.75` for the radius value. The rounded sections are entered as one feature in the Model Tree.
4. Accept the feature, exit the dashboard, and save your model.

Shell the Extrusion

Now you are ready to use the Shell function to hollow out the solid. You only need to select the surface that you want to remove and specify a thickness for the walls of the shell.

1. Click **Insert > Shell**. In the Shell dashboard, enter a thickness of `0.75`.
2. Rotate the model so that you can select the front surface. Accept the feature and save the part.

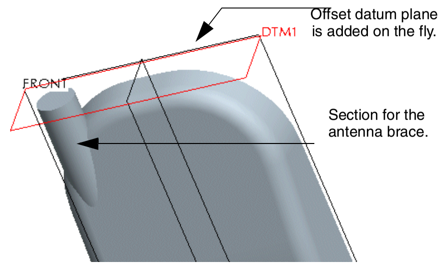
Selecting for the Shell tool



Add the Antenna Brace

The antenna brace is extruded into the solid from a datum plane offset above the part. When a feature is created using a datum made on the fly, the datum is automatically grouped with the feature in the Model Tree. The datum plane is hidden when the feature is finished and is only displayed when the feature is edited. Use the **To Selected** depth setting to make the extrusion conform to the cover surface.

Creating the antenna brace protrusion



1. Click **Insert > Extrude** and enter Sketcher from the Placement panel. Instead of selecting an existing plane for the sketch, click **Insert > Model Datum > Plane**. The **Datum Plane** properties dialog box opens.
2. Make sure the References collector of this dialog box is active, then click the Top datum plane, either in the Model Tree or in the model itself. A new plane is added, offset from the Top datum.

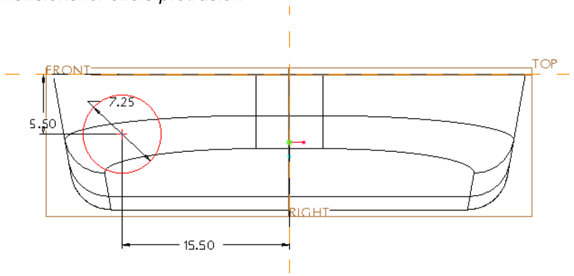
   Note

   Offset is the default datum type when you select a datum plane as a reference. You can use the list box to the right of the selection to change the reference type. To add extra references to the collector, hold down the Ctrl key when you select the references.

3. In the **Offset Translation** box, enter 121.25. This will place the datum plane `2.50` above the edge of the cover. Click **OK** to close the **DATUM PLANE** dialog box, then click the Resume button on the dashboard. Now click **Sketch** and select the new datum plane.
4. The cover is now oriented in a top-up position, making it easier to visualize the location of the section while drawing. If necessary, click **Sketch > Sketch Setup** to reopen the **Sketch** dialog box and make sure the **Orientation** is **Left**. Click **Sketch**.

5. Create a circle on the sketching plane, with the dimensions as shown in the next figure. The diameter is `7.25`, the center is `5.50` from the Front datum, and `15.50` from the Right datum.

Dimensions for circle protrusion

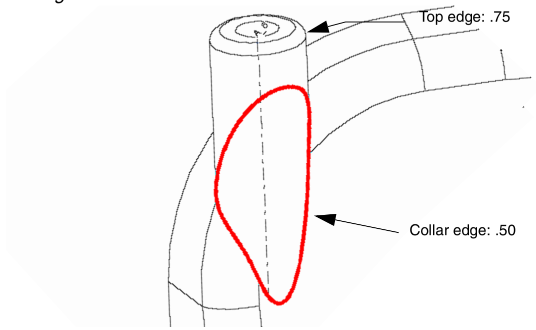


6. Click the Check icon on the Sketcher toolbar to accept the section. Check the extrusion direction and click the direction arrow to correct if necessary.
7. Click **To Selected** as the depth type and select the surface the extrusion will encounter. This stops the protrusion at the next surface it encounters, making it conform to an uneven surface. Click the dashboard Preview icon to see the result. Accept the feature.

Add Hole and Rounds to Brace Feature

1. To add a hole centered on the brace axis, select the brace axis and click **Insert > Hole**. A preview hole is placed in line with the axis. (Be sure axes are displayed. To simplify selecting the axis, use **Pick From List** or set the filter to **Datums**.) Set the diameter to `3.25`.
2. Open the Placement panel. It should show the hole type as coaxial, with the brace axis as the primary reference. Click in the secondary reference collector to activate it, and select the circular surface of the brace as the secondary reference. This positions the hole at the upper surface of the brace.
3. Select **To Next** as the depth type, so that the hole will stop at the next surface. Accept the hole feature and save the part.

Adding rounds to the brace

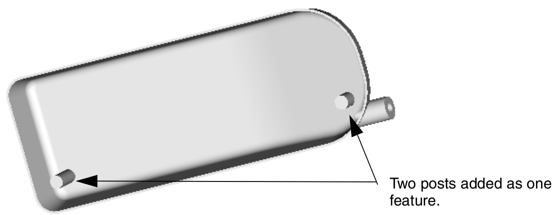


4. Now add the round. Select and then right-click the top edge of the brace and choose **Round Edges** from the shortcut menu. Set a round of `0.75`.
5. Open the Sets panel and click **New set**. Select a tangent chain the same way you did earlier in the part, for the draft edge. With the Smart filter on, click over an edge section until the section itself is selected. Then hold down the Shift key and select any other section in the chain. The entire edge is automatically selected. Add a round of `0.50`.

Add Screw Post Extrusions

In this exercise, you'll finish the cover by creating an extrusion feature consisting of two posts used to screw the halves of the cell phone cover together. You'll put a standard-spec type of hole into the posts, and add rounds to the posts' intersections with the shell. When these features are finished, you'll copy and mirror them to the opposite side of the shell.

Screw posts

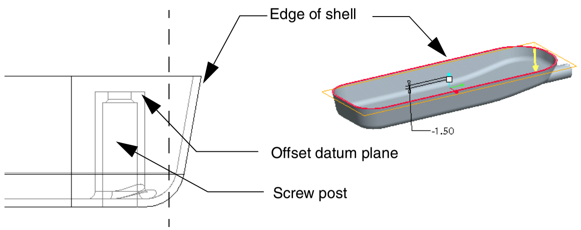


Because the posts sit below the surface of the shell profile, you will make a datum plane offset below the rim of the shell to use as a sketching plane. The posts will extrude from the new plane down to the floor of the shell, the same way the antenna brace joined to the shell.

1. Select **Insert > Extrude**. With the plane collector active in the **Sketch** dialog box, click **Insert > Model Datum > Plane** from the main menu. Set up an offset datum plane the same way you did for the antenna brace, but offset the datum from the lip surface of the cover by `-1.50`. A negative value places the datum below the rim. Click **OK**.

2. Click **Sketch** in the **Sketch** dialog box. If the part does not appear in the correct orientation, reopen the **Sketch** dialog box and click the Flip and Orientation buttons to orient it correctly, then click **Sketch.**

Placing the offset datum plane



3. Draw a circle for the bottom post section first. Scale is not important–draw a large circle so you can see what you're doing, and dimension it down later. Click to complete the first circle, and then move the pointer toward the upper part of the cover, and draw the second circle.

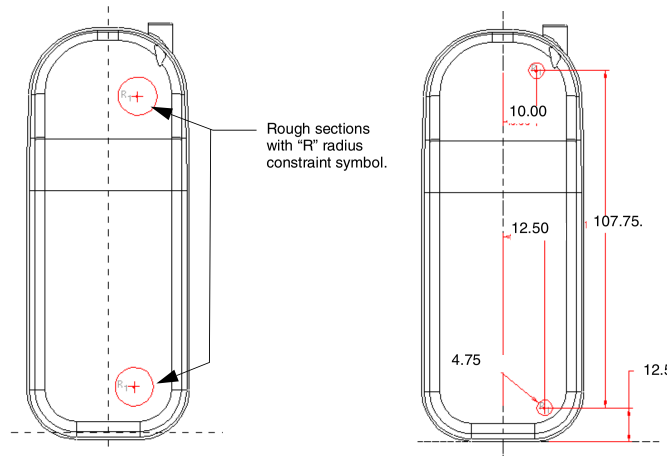
4. As you draw the second circle away from the center, an *"R"* constraint symbol appears at the new circle when its radius matches that of the first one. Complete the circle when the *R* is visible. The second circle is now constrained to share the same radius value.

   Note

   You can also apply this constraint to the two circles after the fact, using **Sketch > Constrain**.

5. Add the strong dimensions, and then enter a diameter value of `4.75`. Accept the sketch. Make sure the extrusion direction is pointing inward, and set depth type to **To Next**. Accept the feature.

Dimensions for the post feature



Add Holes to the Screw Posts

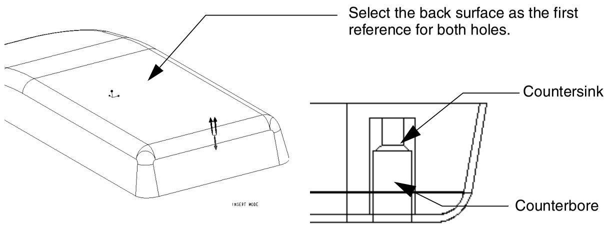
Now add countersunk holes to the posts. You'll add one hole, then copy it to the other post. When you finally mirror the posts and holes to the other side of the shell, they will all be associative.

The first hole is through the back of the shell. It is counter bored, so the countersink occurs near the top of the post, as shown in the next figure. Be sure axes are displayed–they are used as references for the holes.

1. Select **Insert > Hole**. Select the axis with which to place the lower-post hole. A preview hole is placed concentric to the axis.

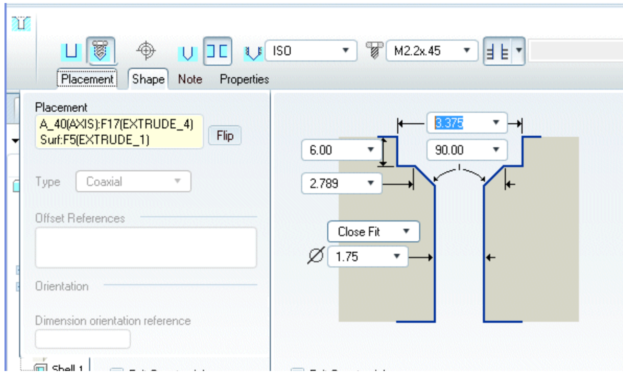
2. Click the Placement panel. The hole should show as coaxial, with the hole's axis as the primary reference. If it is not, click to activate the primary reference collector and select the axis. Now select the large flat surface on the back of the shell as the secondary reference. The hole is now properly referenced.

Placing the hole on the back of the shell



3. Select Standard Hole , an ISO screw size of M2.2x45, Countersink and Counterbore , and a drill depth of Through All. Click the Shape panel and enter the information as shown. Accept the feature.

Countersunk hole setup values



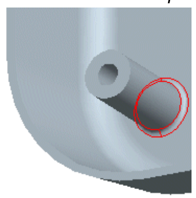
Copy the Hole Feature

You will want the four holes that you will eventually create to be associative, that is, that they all update when a change is made to the parent. The best way to do this is to copy the parent feature to another location, in this case, the second post. After the hole is copied to the second post, you will mirror the posts, holes, and the rounds you will make at the bases to the other side of the Right datum plane. You will then have four drilled posts, all associative.

When you copy a feature, you proceed though a series of prompts in the Menu Manager that let you change dimensions or references for the copied feature. In this case, you'll just be copying the hole along the same plane, to a different axis:

1. Click **Edit > Feature Operations**. Click **Copy** in the Menu Manager.
2. Highlight **New Refs** in the **COPY FEATURE** submenu, (you will be supplying a new reference) **Select** (you will select a feature to copy from the current part), and **Dependent** (the copy will take and retain dimensions from the parent), and then **Done**. You are prompted to select the feature to copy.
3. Select the hole from the Model Tree, and then **OK** in the **Select** box and **Done** in the Menu Manager. The Menu Manager opens to the **Group Elements** menu, with the **Var Dims** line highlighted. The dimensions of the hole are numbered and highlighted. If you wanted to vary the copy, you would use this menu. Because you want the copy to be exact, click **Done**. The **WHICH REF** menu opens. The surface reference for the parent hole is highlighted in the part.
4. Because you are going to use this reference for the copy, click **Same**. The second reference for the parent hole, the axis, is now highlighted.
5. Leaving **Alternate** selected, select the axis for the second post. Click **Done**. The hole is copied, and added to the Model Tree as a copied group. Click **Done**.
6. Multiple-select the base joint of each post and apply a round of `0.75`. Accept the feature.

Round at the base of a post



Mirror the Screw Posts

Finally, copy and mirror the screw posts to the other side of the shell using the same process as copying the holes in the PC board.

1. In the Model Tree, select the:

    a. Protrusion with the two posts
    b. Original hole
    c. Group for the copied hole
    d. Round feature at the post's bases (The rounds should be one feature.)

2. Click the Mirror icon on the toolbar  and select the Right datum plane as the mirror plane.

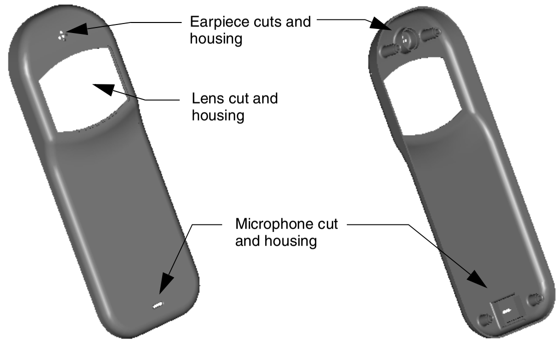
3. The copied features are mirrored and added as a group to the Model Tree. Save and close `back_cover.prt`.

Holes and screw posts mirrored

Summary

You have created a part using a shell, a draft, rounds, and holes as features, and learned how to use some fairly simple geometry to create a detailed and complex form. The final part is a more elaborate exercise that uses many of the same techniques.

Part 8: The Front Cover



The front cover is similar to the back cover in that it is a shelled protrusion with screw posts extruding from the interior surface. It includes cuts to accept the lens and mounting shelves for the earpiece and the microphone parts.

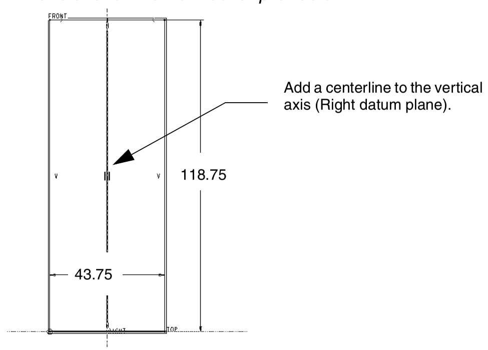
The front cover also requires cuts to accommodate the buttons of the keypad. In this case, the easiest way to create the holes in the front cover is to do it at the assembly level. When the keypad and the front cover are part of an assembly, you can create all the required cuts by removing material where interference exists between the two parts. The cutouts are passed back to the part and remain stored with it.

Technique or Feature	Where Introduced
Insert Protrusion, Rounds	Part 1: Lens
Extruded Cut: Two Sides	Part 7: Back Cover
Draft Feature	Part 7: Back Cover
Shell Feature	Part 7: Back Cover
Make Datum on the Fly	Part 7: Back Cover
Hole Features	Part 2: Earpiece
Copy and Mirror in 3D	Part 6: Keypad; Part 7: Back Cover
Rounds using Edge Chain	Part 7: Back Cover
Import Saved Section	New

Create the Front Cover Protrusion

Create a new part called `front_cover`. Create the first protrusion in the same way that you created the first protrusion for the back cover. Use the Front datum as the sketching plane, and accept the defaults for orientation. In Sketcher, place a centerline on the vertical axis and sketch the rectangular protrusion as shown in the next figure. Make sure that the " > <" symbols show that the rectangle is centered horizontally on the centerline. When you accept the sketch and return to the dashboard, set the depth at `4.875`.

Dimensions for the front cover protrusion

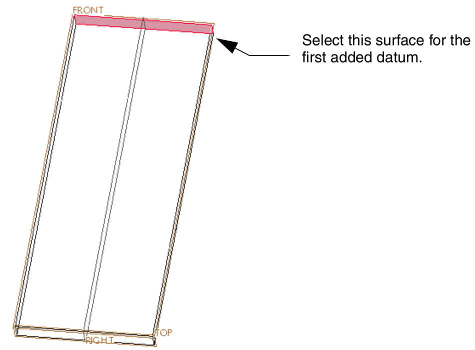


Add Construction Datum Planes

Next, you'll add two construction datum planes. The first one will be through the upper surface of the first protrusion. The second will be offset by 2 units along the protrusion from the first one.

1. To add the first datum plane, click **Insert > Model Datum > Plane**. Select the surface opposite the Top datum as a reference. The new datum appears in preview.
2. Click the direction arrow on the preview datum so that it points toward the solid. Because the datum will be flush with the surface, leave the Translation value set at zero. Click **OK** in the **Datum Plane** dialog box.
3. To add the second datum plane, click **Insert > Model Datum > Plane** again. Click the datum you just added as the reference for the new datum. Enter a translation value of `50`.
4. Click **OK** in the **Datum Plane** dialog box. The new datum is added, offset into the protrusion 50 units from the first.

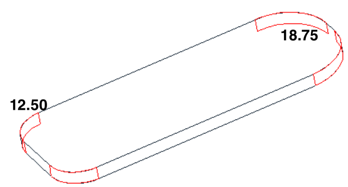
Adding construction datum planes



Round the Front Cover Corners

1. Use the Ctrl key to select the top two corner edges. (It will be easier to select the edges if you turn off datum display.)
2. Right-click and select **Round Edges** from the shortcut menu to add a round of `18.75`. In the **Sets** panel, click **New set**. Select the bottom two corner edges and apply a round of `12.50`. Accept the feature.

Dimensions for rounded corners



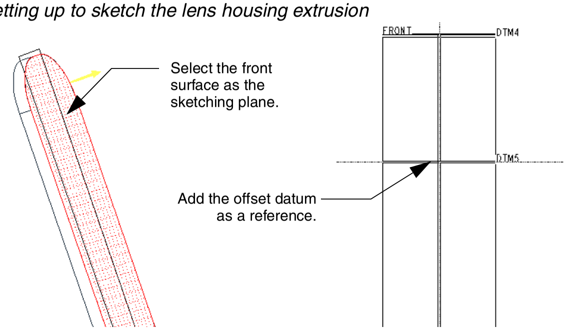
Lift the Lens Housing Extrusion

In this procedure, you'll create a new extrusion from the front surface of the first one, and you'll learn how to use some of the edges that already exist to define the new section.

1. To start, click **Insert > Extrude** and select the front surface of the first extrusion as the sketching plane. This is the surface that is offset from the Front datum. Click **Sketch**.

2. Open the **References** dialog box, add the offset construction datum to the references, and click **Close**. (Make sure that datums are displayed. Once the datum is referenced, you can turn the datum display off to unclutter the drawing area.)

Setting up to sketch the lens housing extrusion



3. Select **Sketch > Edge > Use** to define the extrusion using existing edges. When the **Type** dialog box opens, click **Chain**, but don't close the dialog box.

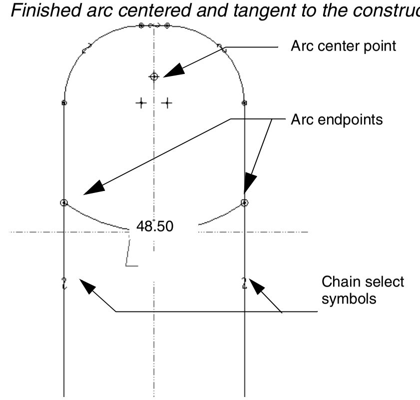
4. At the prompt, click both sides of the existing extrusion. Creo Elements/Pro selects the two lines and one of the arcs between them, the upper or lower. Use the the **Choose** dialog box to make sure the upper arc is selected (click **Next** to change the selection), and click **Accept**. Click **OK** and then click **Close** in the **Type** dialog box. The chain is marked by the *S*-shaped chain symbol.

   Note

   You could have selected the edges individually, but when you establish a chain, the whole shape updates when any changes are made to the underlying geometry.

5. Place a Center and Endpoints arc so that the center point is aligned with the vertical center, and constrain it so the arc is tangent to the construction datum plane reference line.

Finished arc centered and tangent to the construction datum



6. Now trim the unnecessary lines below the arc points. Use the Dynamic Trim tool to erase the lines below the arc you added.

7. Enter `48.50` for the arc radius. Because the section's edges are defined by the underlying edges, and the arc's center is in line with the centerline, the angle value is enough to define the dimensions of this section.

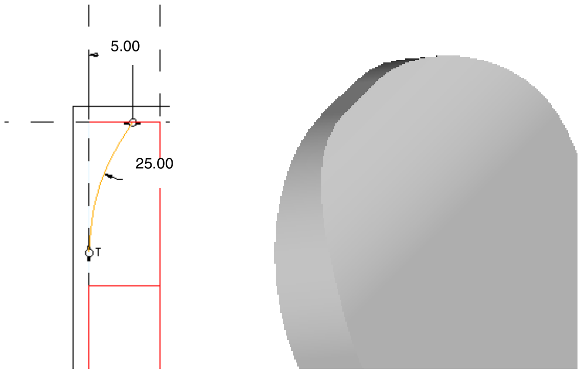
8. Click in the Sketcher toolbar to accept the section. Enter a depth of `3.25`. Check the direction and accept the feature.

Add the Earpiece Cut

Now you'll make a cut to round the top of the cover for the earpiece. The section for this cut is similar to one you made previously in the back cover.

1. Click **Insert > Extrude**. Use the Right datum plane (bisecting the cover) as a sketching plane. Add the front edge and the first datum plane you created as references.
2. Use a simple arc to define the section as shown in the next figure. Make sure the automatic tangent constraint symbol (*T*) appears. If not, use the Constrain tool  to add it.
3. If the 5.00 dimension is not automatically added as a weak dimension, use the Add Dimensions tool  to insert it.

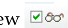
Dimensions for the earpiece cut section



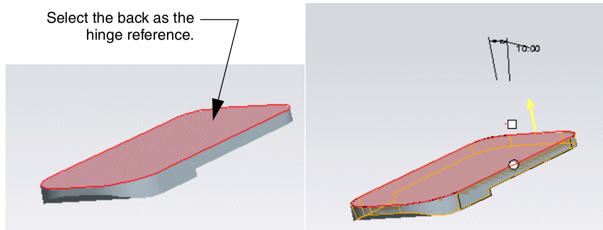
4. Enter a value of 25.00 for the arc. Accept the sketch.
5. Click the Remove Material button to make the extrusion a cut. Make sure the cut direction arrow is pointing away from the solid. In the Options panel, set Side 1 and Side 2 to **Through All**. Preview the cut and then accept it.

Create the Draft Feature

Now you'll apply a draft to the side surfaces in the same way that you did to the back cover part.

1. Select one of the side surfaces of the solid. Click **Insert > Draft**. The Draft dashboard opens.
2. With the **Draft hinges** collector active, click the back surface of the cover. The draft angle indicator and handles should appear on the model.
3. Drag the handle out to 10 degrees, or type 10 in the dashboard value box.
4. Click Preview  to see the finished feature, or accept the feature and return to the work area.

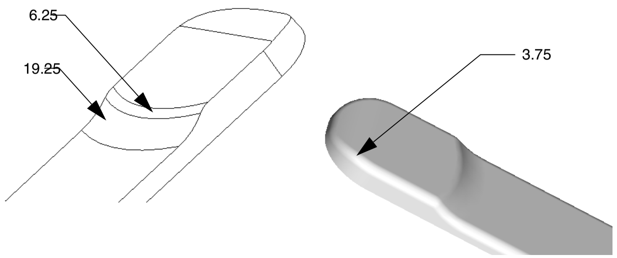
Handles and direction indicator for the draft angle



Apply Rounded Edges

1. Select the edges between the face and the upper extrusion and apply rounds. The lower radius is 19.25. The upper radius is 6.25. When these two rounds are applied, the upper edge of the phone becomes an unbroken line.
2. Select any section of the cover edge, hold down Shift and select another section. The entire edge is chosen. Apply a rounded edge of 3.75 from the shortcut menu.

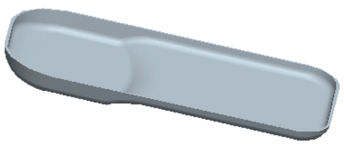
Dimensions for face edge rounds



Shell the Solid

With the rounds complete, you are ready to apply the shell feature. Use the same procedure you applied to the back cover. Enter `0.75` for the shell thickness.

Cover after the shell process



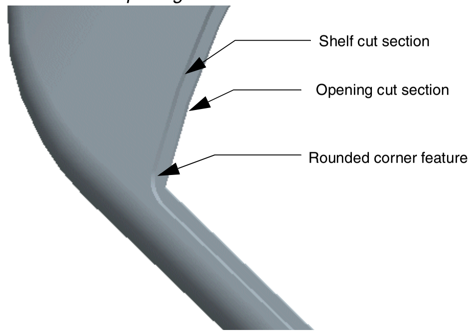
Create Lens and Earpiece Cuts

The following set of procedures completes the interior of the phone. You'll create an opening for the lens, and mounting forms with small holes for the microphone and the earpiece. Finally, you'll create the screw posts to join with the ones in the back cover.

Make the Lens Shelf and Opening Cuts

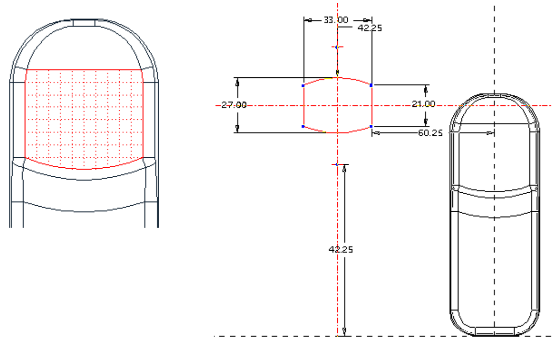
The first cut is the shelf on which the lens mounts. Its dimensions are the same as the lens itself. When the lens was created in the first exercise, the section was saved as `lens.sec`. This saved section is used for the cut.

Detail of the lens opening



1. Click the Saved View icon and orient the cover to the Front view. (This isn't absolutely necessary but it will make following the directions easier.)
2. Select **Insert > Extrude**. Select the surface that will use the lens opening cut as the sketching plane. Click **Sketch**.

The sketching plane for the lens cut, and the imported section



3. Select **Sketch > Data from File > File System**. In the browser, select the `lens.sec` file. Click on the approximate area to place the sketch. The section is imported and set into the graphics window. The **Move and Resize** dialog box opens.

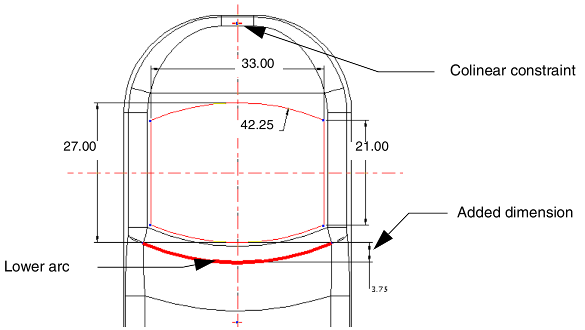
4. Make sure that the scale is set to 1. Click the Check icon in the dialog box to accept the section. The section is placed with weak dimensions defining the distances from the reference lines.

5. Select **Sketch > Dimension > Normal**. Add a dimension of 3.75 between the bottom arc of the lens section and the lower arc of the lifted extrusion.

6. Now align the centers of the section and the cover. Select **Sketch > Constrain**. In the **Constraints** dialog box, select the **Collinear** constraint. Click the centerline of the section, and then the centerline of the cell phone cover. The two centerlines are aligned concentrically. Close the **Constraints** dialog box. Now enter the strong dimensions as shown in the next figure and accept the section.

7. Click the Remove Material icon. Click the direction arrow so it points into the cover. This cut does not go completely through the face; set the depth to **Blind**  and 0.50. Accept the feature.

Imported section in place



Create the Lens Cutout

As demonstrated previously, you can use an existing edge as the basis for the dimensions of another edge. This is the obvious technique for creating the cutout for the lens.

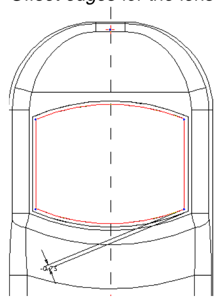
1. Select **Insert > Extrude** . Select the floor of the lens cut as the sketching plane.

2. Click the Offset From an Edge tool  from the Use Edge flyout menu. In the **Type** dialog box, click **Loop**. (Don't close the dialog box.)

3. Select the shelf outline. The loop is defined from all the connected lines of the shelf outline. The direction of offset is shown by a yellow arrow, and you are prompted for an offset value. Because you want the offset inside the shelf outline, type a value of -0.75. Close the **Type** dialog box. The outline for the cutout section is created as an offset. Accept the section.

4. Set the depth to **Through All**. Click the Remove Material button to create the cut. Make sure the direction arrow points into the cover. Accept and save the feature.

Offset edges for the lens cut section



Round the Opening Corners

Select all four inside corners of the shelf cut, and add rounds of `2.00` to them, as shown in the next figure.

Round the lens housing corners

Note

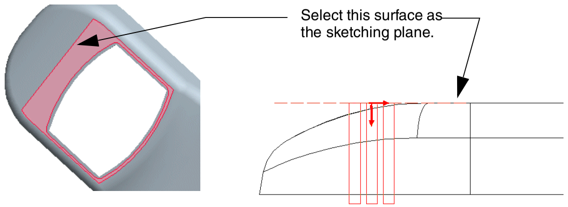
The lens part has the same rounds, added as separate features to the lens extrusion. There are many methods in Creo Elements/Pro, beyond the scope of this tutorial, that ensure that adjoining parts not only match in size and dimensions, but are also associative. They include creating new parts from within the assembly, or using a skeleton part as a reference for all associated parts.

Create the Earpiece Cuts

You will now use a section consisting of five holes (one at the center and the other four positioned evenly in a circle around it) to arrange the earpiece cuts as geometrically aligned holes.

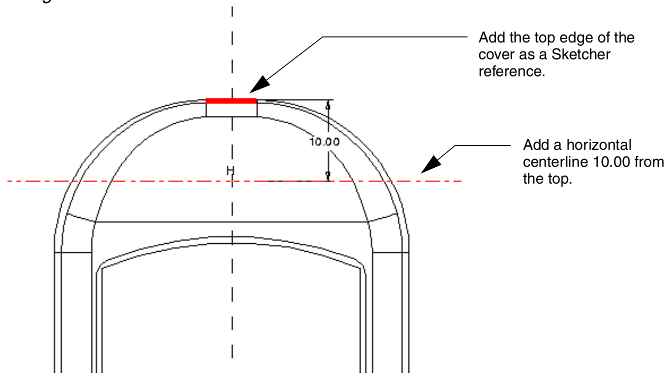
1. Select **Insert > Extrude**, and select the flat area surrounding the lens cutout as the sketching plane. The holes will enter the shell at 90 degrees to the face, and not to the plane of the curved surface they go through. Add the topmost edge of the cover as a reference line.

Setting up the earpiece cuts



2. Next, place a horizontal centerline `10.00` below the referenced line. Use the Add Dimensions tool to dimension the centerline from the top reference.

Dimensioning the horizontal centerline



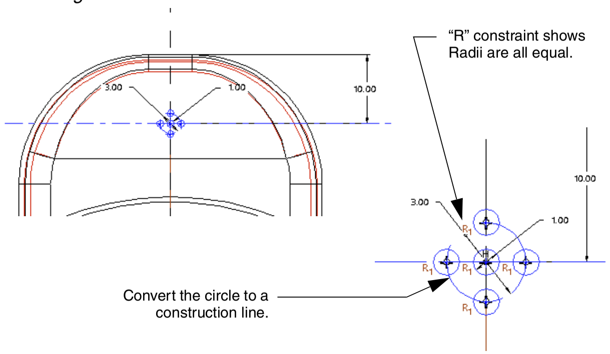
3. Place a circle to use as a construction line for the outer holes. Select the Circle tool, and center a circle where the vertical and horizontal references meet. Set the circle's diameter at 3.00.

4. Select the circle and click **Construction** on the right mouse button shortcut menu to toggle it to a construction line.

5. Now define the cuts. Select the Circle tool again and draw a circle centered on the intersection of the vertical and horizontal axes. Middle-click to exit, and set the diameter of the circle to 1.00.

6. Click the Circle tool again and draw four more circles where the centerlines intersect the construction line circle (see the next figure). When the R constraint symbol appears, the radius of the new circle is the same as the radius of the center circle.

Sketching the earhole section



7. After the cuts are defined, accept the section. Click the Remove Material icon to create the cut. Set the depth to **Through All**. Make sure the direction arrow points into the cover. Accept the feature.

Create the Earpiece Holder and Shelf

The earpiece holder is a thin extrusion. A thin extrusion is hollow, which precludes combining a solid and a cut to define a shape, for example, a cylinder. You give the thin extrusion a wall thickness value when you define the section.

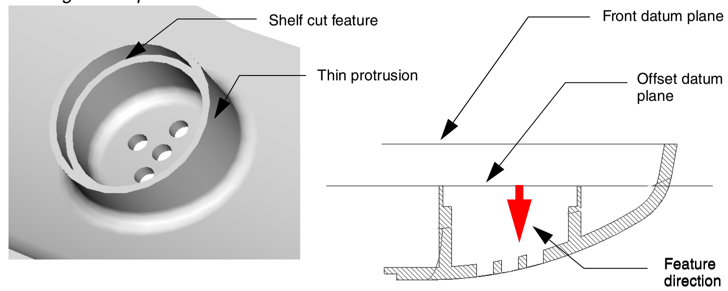
Once again you are placing a feature through a surface that is not level, in this case, the curved upper surface of the cover. The solution is to extrude it down to the floor of the cover from an offset datum plane, as shown in the next figure. The depth setting of **To Next** will make the collar conform to the surface when the two intersect.

You'll create this datum when you are prompted to select the sketching plane. You could create the datum before you start the feature, but when you create it in the dashboard environment it "belongs" to the feature, and is grouped with the feature in the Model Tree.

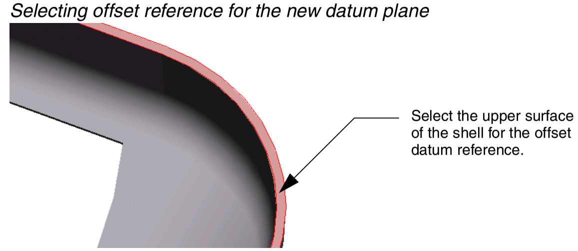
You have already created a datum like this in the back cover, so use the following guidelines to add the new offset datum plane and create the earpiece holder section.

1. Select **Insert > Extrude**. Click **Insert > Model Datum > Plane** to create a datum to be used as the sketching plane. Use the upper surface of the shell for the offset reference and offset the new datum plane by -2.50 from the reference surface.

Creating the earpiece holder



Selecting offset reference for the new datum plane



2. Zoom in on the ear hole pattern feature. Use the Concentric Circle tool to draw a circle. Select the center ear hole. Drag the new circle outward. It is concentric with the center ear hole. Set the circle diameter to 7.00. This will be the inside diameter. Accept the section.

3. Set up the thin extrusion:

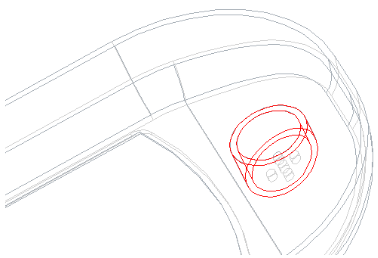
   a. Set the depth to **To Next**.

   b. Make sure the direction arrow points from the datum toward the cover.

   c. Click the Thicken Sketch icon . Enter a value of 0.75 This determines the thickness of the wall.

   d. Click the Direction icon (to the right of the value field) to be sure the sketch dimension is the inside diameter. The icon toggles the thickness to the inside of the section, the outside of the section, or the center of the section. Watch the preview as you toggle: the feature will be at its largest when the section is used as the inside diameter. Accept the feature.

The finished thin extrusion

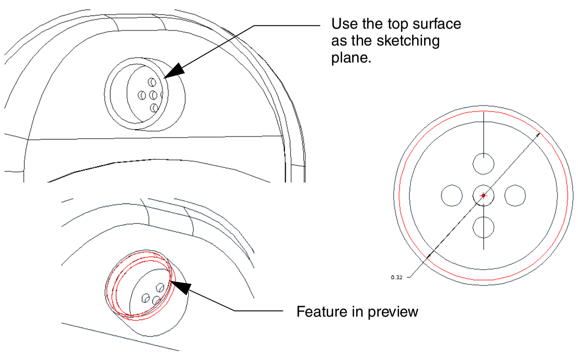


Add the Shelf Cut and Final Round

The cut for the shelf in the earpiece holder is simply an extruded cut feature that uses the top surface of the holder as the sketching plane. The cut section is a concentric circle within the holder section, centered on the same axis as the holder.

1. Use the Concentric Circle tool in the Sketcher toolbar to create the cut with a diameter of `8.00` and a height of `1.25`.
2. Create a round at the seam between the holder and the shell with a radius of `0.75`.

Sketching the shelf cut

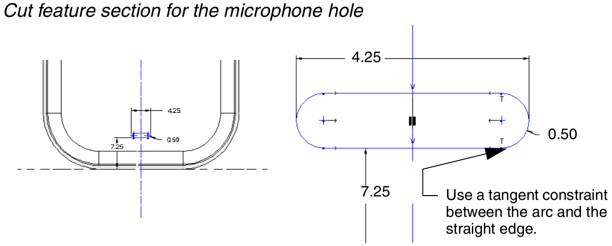


Make the Microphone Cut and Holder

Now, at the lower end of the shell, you'll create features to house the microphone.

1. First, create the cut for the microphone hole. Select the floor surface of the shell as the sketching plane. Use the lowest edge of the cell phone shell as a reference.
2. Place a centerline along the Right datum, down the center of the shell. Draw one side of the slot. Use a tangent constraint between the arc and the straight edge. Mirror the sketch along the centerline, using the dimensions shown.
3. Complete the feature: remove material and check the direction. Set the depth to **Through All**.
4. Accept the feature. Place a round (`0.50`) on the front edge of the slot.

Cut feature section for the microphone hole

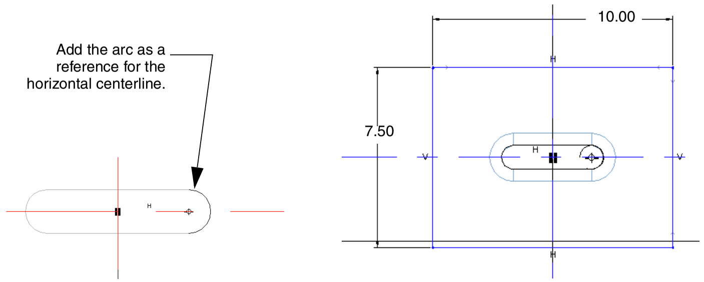


Make the Microphone Housing

This is a rectangular, thin extrusion with a wall thickness of 0.75. It is also extruded down into the shell from an offset datum created on the fly as a sketching plane.

1. Create the datum plane in the same way as the earpiece holder, but offset it by `-2.50` from the upper surface of the shell. After the datum plane is created, it automatically appears in the sketching plane collector.
2. Place a vertical centerline along the Right datum. In the **References** dialog box, add one of the arcs of the cut as a horizontal reference, then snap a horizontal centerline through the center of the arc.
3. Use the Rectangle tool to create the section. Be sure you see the " > <" equidistant constraint symbols indicating that the rectangle is bisected by the centerlines. Dimension as shown.

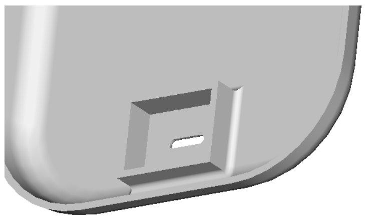
Sketching the microphone housing extrusion



4. Accept the section. Click the Thicken Sketch icon and set the depth to **To Next**. Toggle the extrusion direction until the section is at its largest. Set thickness to `0.75`.

5. Accept the feature. Add a `0.25` round to the base of the protrusion.

Completed microphone housing

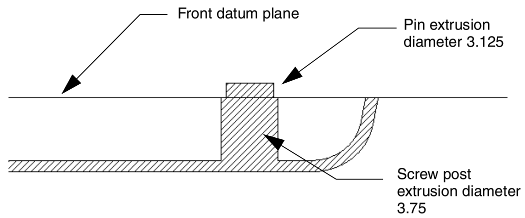


Add the Screw Posts and Holes

Now all that remains is to create the four screw posts. As shown in the next figure, a screw post comprises three features. The first feature is a simple round protrusion extruding down into the shell from a datum plane. Instead of adding a datum plane, use the Front datum plane for this feature.

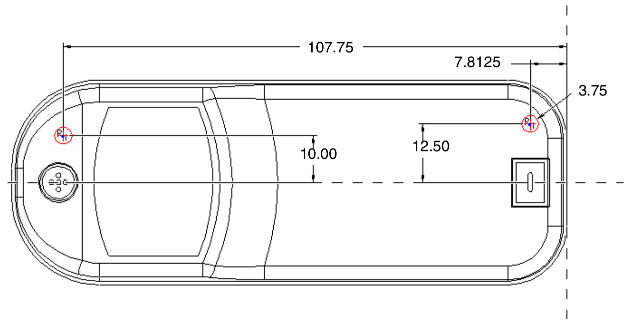
Create two of these posts, upper and lower, as one feature, the same way that you did the posts in the back cover. Again, because the protrusion is flush with the edge of the shell, you can use the Front datum plane as the sketching plane.

The screwpost features: post, pin, and hole



1. Make the upper and lower posts as one feature on one side of the shell, the same way that you did the posts in the back cover. See the next figure for the section dimensions.

Overall dimensions for the posts section

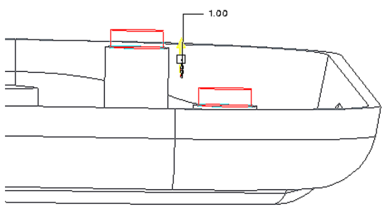


2. Now sketch the pin feature, using the top surface of the post as the sketching plane. For this section, set the direction so the protrusion extrudes up from the top of the post.

   Use the Concentric Circle tool to draw the sections concentric with the existing posts. Remember to look for the "R" constraint symbol, showing you that the second pin you draw is the same radius as the first. Set the diameter at 3.125.

3. Set the direction outward, up from the sketching plane. Set the depth at `1.00`. Accept the feature.

The "pins" extrude above the shell edge



Insert Holes

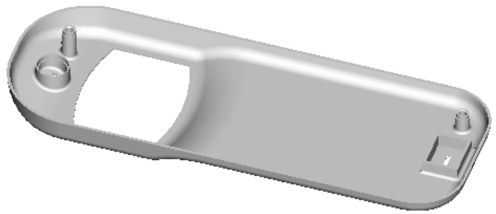
1. Use the Hole tool to insert the coaxial holes in each post. The holes use the pin axis as the first reference and the pin surface as the second reference. They are standard M2.2X.45 holes with ISO threading. Screw depth is `5.25`.

2. Accept the feature and add `0.50` rounds to the juncture between the posts and the shell.

Copy and Mirror the Posts

When the two posts are finished, select all the features they include, and use the Copy and Mirror procedure to add them to the opposite side of the Right datum plane, as you did for the back cover. Save and close the part.

Post features ready to mirror



Summary

All of the parts needed for the assembly exercises are now finished. Read the Introduction to Assembly in the next chapter and complete the exercises to create a new assembly file.