

# Plastic Parts and Features

## Topics in this section

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- Split the Solid
- Create the Grill
- Create a Rule Fillet
- Create a Rest
- Create a Lip
- Create a Body Using Split
- Create a Segmented Lip
- Create a Snap Fit
- Add a Rule Fillet to a Feature
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- Moving Bodies
- Use the Combine Command
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- Export the Design as Individual Parts
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## Explore Multi-bodies and Plastic Features



**Category**

**Time Required**

**Tutorial File Used**

**Experienced Users**

60 minutes

PFTutorial.ipt

Control\_Button\_Solid.ipt

PFTutorial\_Revolve\_Combine.ipt  
PFTutorial\_Complete.ipt (finished version)

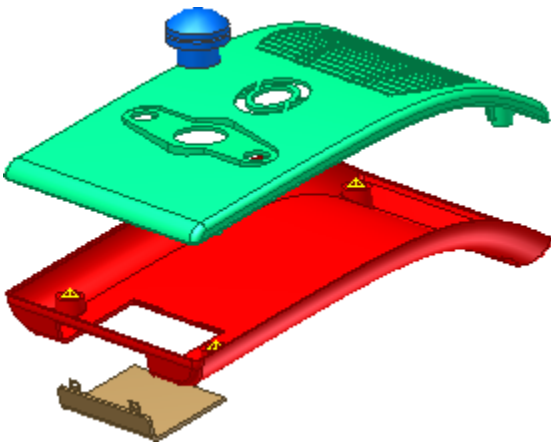
Create a plastic case for a hand-held music device using the basics of multi-body and plastic feature functionality in Inventor. Automate the creation and editing of common plastic part features.

A multi-body part is a top-down workflow. You create and position multiple solid bodies within a single part document. This technique is especially useful in the design of plastic parts.

A top-down workflow eliminates the need for complex file relationships and projected edges between parts. All editing takes place in a single file. At any time, you can generate unique part files for each body. The generated files are derived parts that are associative to the master design in the original part file.

If you do not want to learn how to use the plastic features commands, use the file PFTutorial\_Revolve\_Combine.ipt as a start point. Begin the tutorial at the section titled Insert a toolbody using Derive.

Before you begin, open and review the supplied plastic part. To simplify the process of body and feature creation, the tutorial part sketches, work planes, and surfaces are named to help you identify them.



You can complete the tutorial in segments if you save your work before you exit.

## Objectives

- Create new bodies using **Split**.
- Create grills, rule fillets, rests, and lip features.
- Create snap fits and bosses.
- Create a body in place using feature commands.
- Insert a toolbody using **Derive**.
- Use **Move Bodies** to position a toolbody.
- Use **Combine** to perform a Boolean operation.

## Prerequisites

- Intermediate level of understanding part modeling.

- See the Help topic “Getting Started” for further information.
- Activate the **Tutorials.ipj** project file.

### Navigation Tips

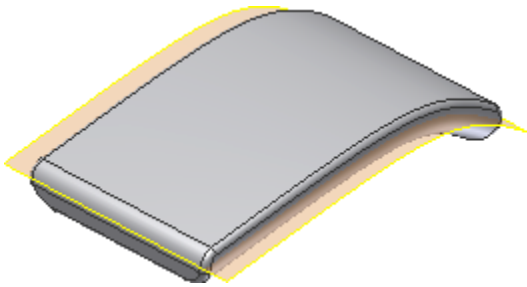
- Use **Show** in the upper-left corner displays the table of contents for this tutorial with navigation links to each page.
- Use **Forward** in the upper-right corner advances to the next page.

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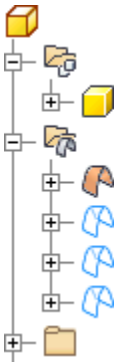
## Split the Solid

To begin:

1. Open **Tutorial Files > Plastic Parts and Features > PFTutorial.ipt**, and orient the view to match the following image.

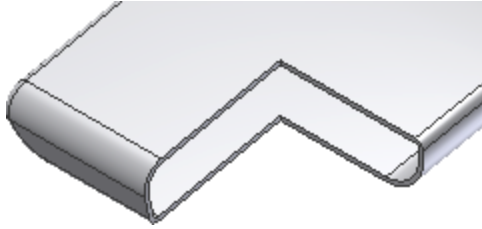


2. In the browser, expand the folders for the **solid** and **surface** bodies. Note that there is only one solid body present and four surface bodies. One of the surface bodies is visible, and all the others are not visible.

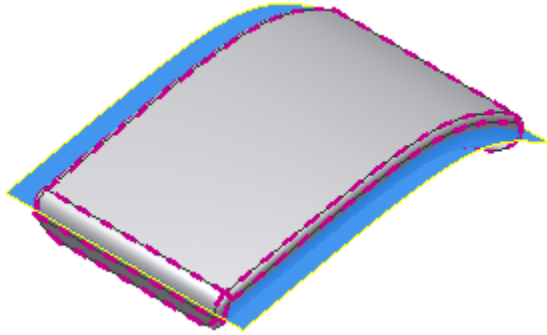


3. On the ribbon, click **Model tab > Modify panel > Shell**. Do not remove any faces. For **Thickness**, enter 1.5 mm. Click **OK** to create.

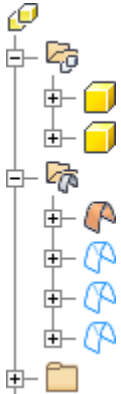
The following cutaway image shows the results of the shell operation. The cut is not part of the model.



4. Click Model tab > Modify panel > Split. In the dialog box, choose the **Split Solid** option. Select the surface named **PartingSurface** as the **Split Tool**. Click **OK** to finish.



Notice that the part icon in the browser changed to represent a multi-body part. The **Solid Bodies** folder contains two bodies. You can right-click each body and select **Properties** from the context menu to set visibility or color.



**Note** We recommend that you assign a unique color to each body in a multi-body part to keep them visually separate. Select each body in the **Solid Bodies** folder in the browser, and then right-click and select **Properties** from the context menu. You can also select a body and then choose a color override in the Quick Access toolbar dropdown to set the color.

Before proceeding, expand the **Surface Bodies** folder. and turn off the visible surface.

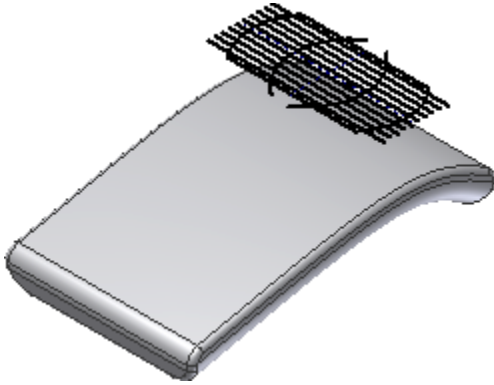
Save the file as **PFTutorial\_1**.

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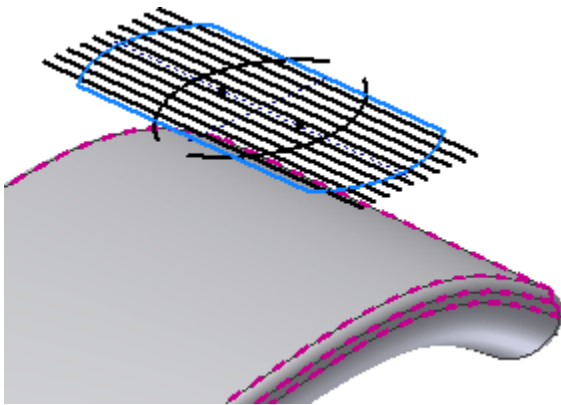
## Create the Grill

In the next section, we will create the plastic grill feature.

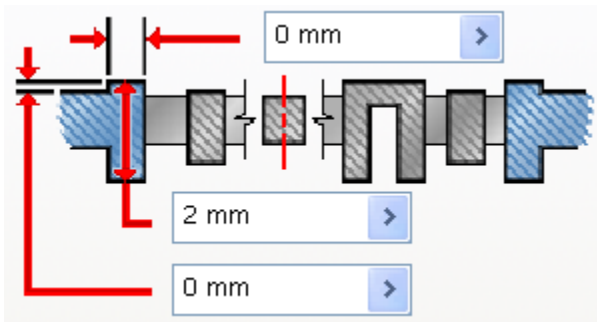
1. In the browser, turn on the **Visibility** of **GrillSketch**.



2. On the ribbon, click Model tab ➤ Plastic Part panel ➤ Grill.
3. Activate the **Boundary** tab and then select the outer closed profile in the graphics window as shown in the following image. Select the target solid if the application did not select the body closest to the sketch.



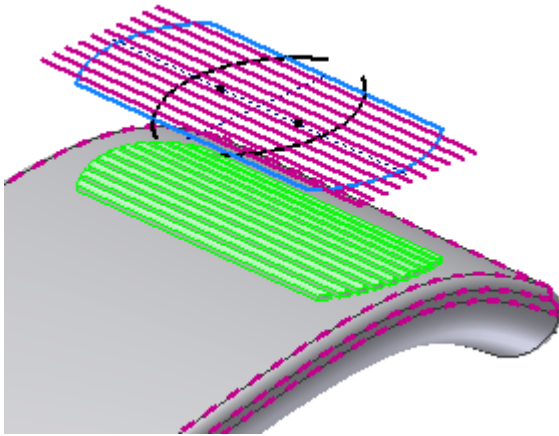
- **Thickness** = 0 mm
- **Height** = 2 mm
- **Outside Height** = 0 mm



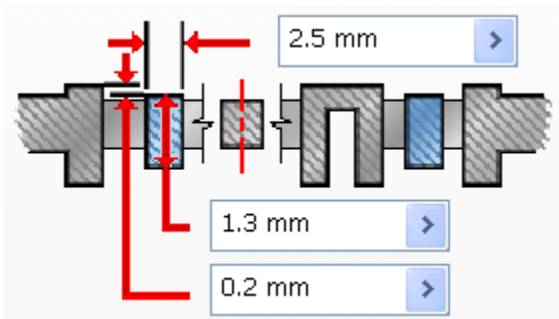
We will skip the **Island** tab. This sketch does not contain a closed boundary for an island. The following image shows an example of an island.



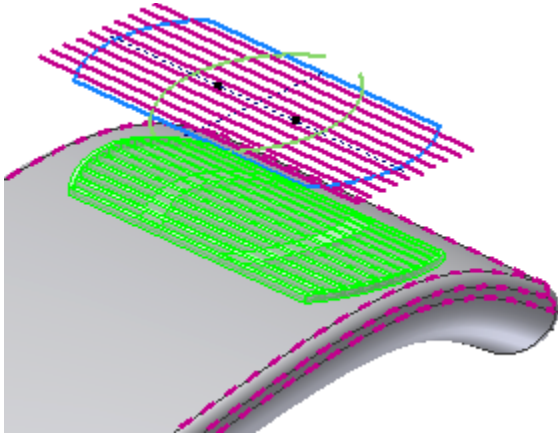
4. Activate the **Rib** tab and then select the lines.



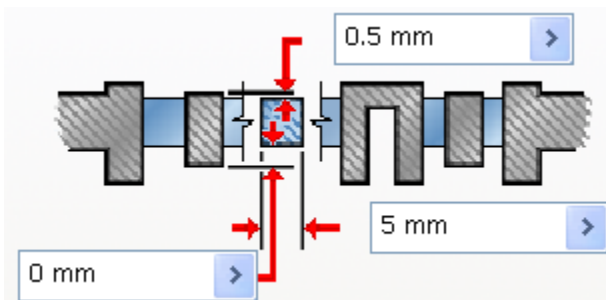
- **Thickness** = 2.5 mm
- **Height** = 1.3 mm
- **Top Offset** = 0.2 mm



5. Activate the **Spar** tab and then select the arcs.



- **Top Offset** = 0.5 mm
- **Thickness** = 5 mm
- **Bottom Offset** = 0 mm

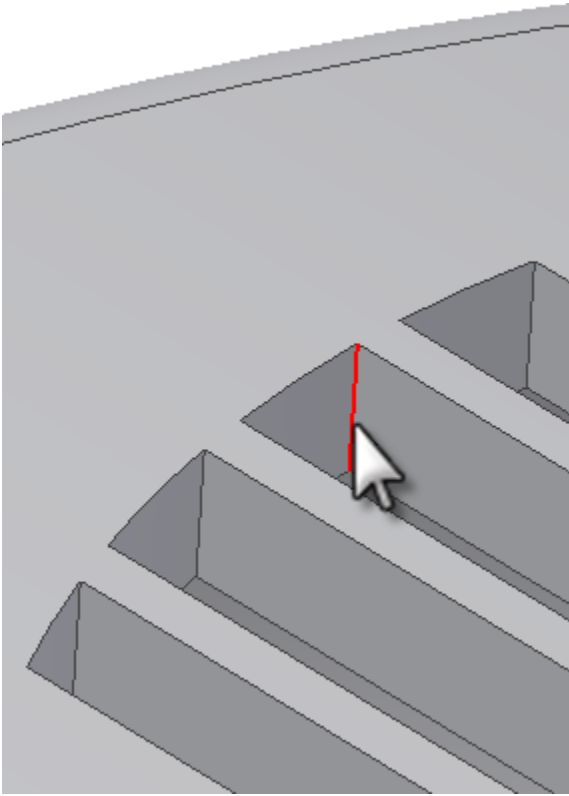


6. Click **OK** to create the grill.

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## Create a Rule Fillet

We now want to fillet all the vertical edges of the grill like the one indicated in the following image.

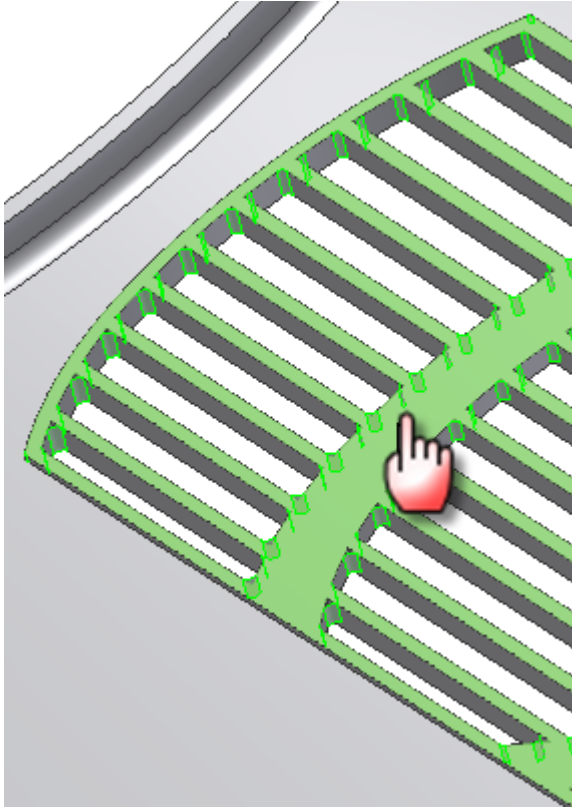


The Rule Fillet feature is designed to fillet an entire feature based on pre-determined rules. It means we can fillet the entire grill without having to pick individual edges.

In the following sequence, we will define the rules which allows the entire grill to be filleted in a few picks.

1. In the browser, turn off the **Visibility** of the bottom solid without the grill. Rotate the part with the grill feature to the inside face of the solid.  
**Tip** The context menu contains three commands to control body display: **Visibility**, **Show All**, and **Hide Others**.
2. On the ribbon, click Model tab ➤ Plastic Part panel ➤ Rule Fillet.
  - Use the Source drop-down to set the source to **Face**.
  - Select the inside curved face of the grill as shown in the following image.
  - Set the **Radius** to 0.5 mm.
  - Set the **Rule** to **Incident Edges**.
  - Select the **Y Axis** in the **Origin** folder to specify the direction.
  - Set the **Tolerance** to 1 deg.





3. Click **OK**. You created 128 fillets using rules and a couple of picks.

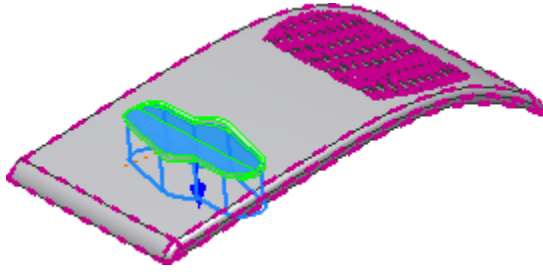
**Note** The Rule Fillet can be used on any feature in a part file. It is not exclusively for use with plastic parts.

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## Create a Rest

We now want to build a flat area for some control buttons.

1. Orient the view to the outside of the part as shown in the following image.
2. Turn on the visibility of **RestSketch** in the browser.
3. On the ribbon, click Model tab ➤ Plastic Part panel ➤ Rest.
4. On the **Shape** tab:
  - Select the **RestSketch** as the profile. If it is the only visible sketch it is automatically selected.
  - If required, select the top solid as the **Solid**.
  - For **Thickness**, enter 1.5 mm and specify **Inside**.
  - In the drop-down, select **Through All**.



5. On the **More** tab:
  - For **Landing Options**, select **Distance** and enter 0 mm.
  - For **Landing Taper**, enter 0 deg.
  - For **Clearance Taper**, enter 0 deg.
6. Click **OK** to create the Rest feature.

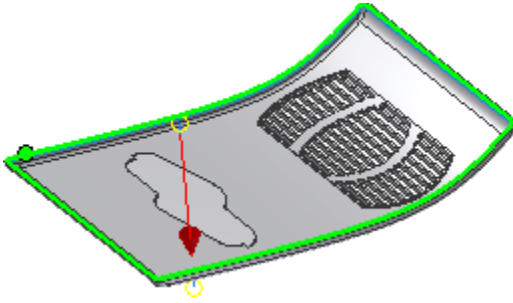


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## Create a Lip

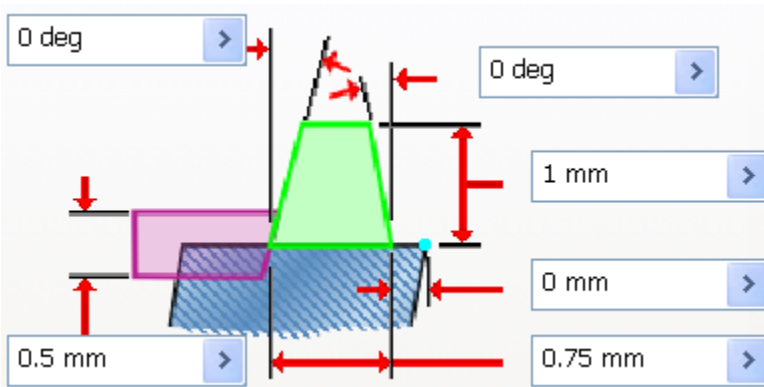
We will now build a lip-groove combination to facilitate placing the mating parts in a physical assembly. Orient the view to the inside of the part like the following image.

1. On the ribbon, click Model tab ➤ Plastic Part panel ➤ Lip.
2. Make sure the **Lip** button is selected in the dialog box.
3. On the **Shape** tab:
  - Select the inside edge as the **Path Edges**.
  - Leave the **Path Extents** unchecked.
  - Click the **Pull Direction** check box.
  - Select the **Y Axis** in the **Origin** folder as the pulling direction.

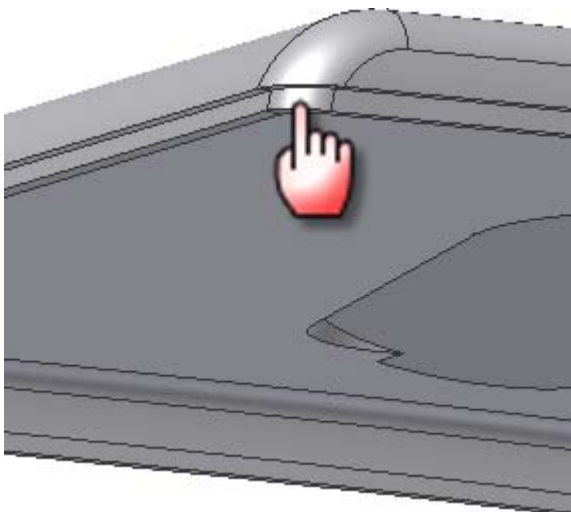


4. On the **Lip** tab:

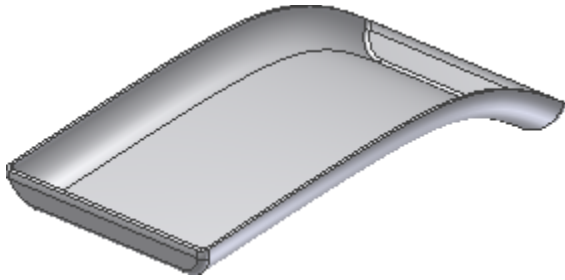
- For **Outside Angle**, enter 0 deg.
- For **Inside Angle**, enter 0 deg.
- For **Height**, enter 1 mm.
- For **Shoulder Width**, enter 0 mm.
- For **Width**, enter 0.75 mm.
- For **Clearance**, enter 0.5 mm.



5. Click **OK** to create the lip.

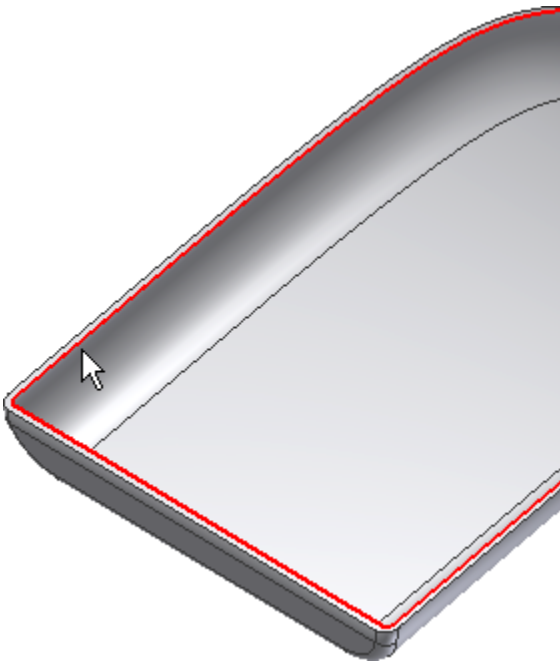


6. In the **Solid Bodies** folder in the browser, use the context menu to turn off the **Visibility** of the top body. Turn on the visibility for the bottom body. Orient the view to the inside of the part like the following image.

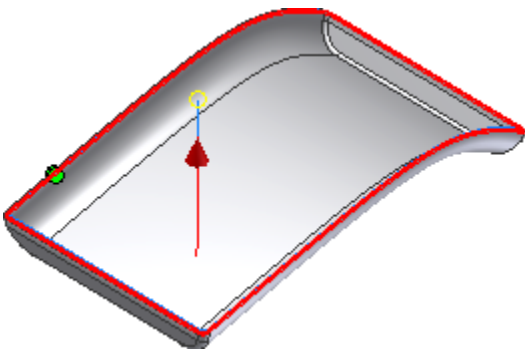


Next, we will use the **Lip** command to create the mating groove.

7. Click the **Lip** command.
8. On the **Shape** tab:
  - Make sure the **Groove** button is selected.
  - Select the inside edge as the **Path Edges**.

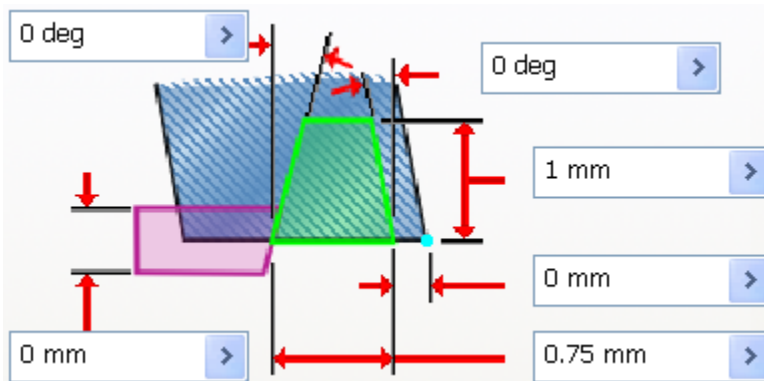


- Leave the **Path Extents** unchecked.
- Click the **Pull Direction** check box.
- Select the **Y Axis** in the **Origin** folder.



9. On the **Groove** tab:
  - For **Outside Angle**, enter 0 deg.

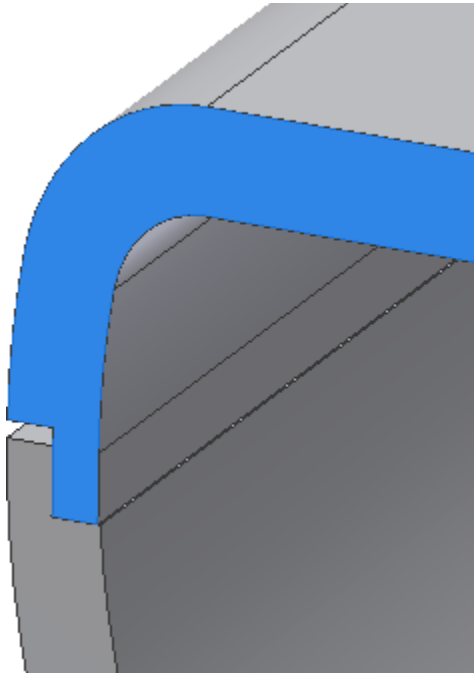
- For **Inside Angle**, enter 0 deg.
- For **Height**, enter 1 mm.
- For **Shoulder Width**, enter 0 mm.
- For **Width**, enter 0.75 mm.
- For **Clearance**, enter 0 mm.



10. Click **OK** to create the groove.



The mating lip and groove features are shown in the following cutaway view.

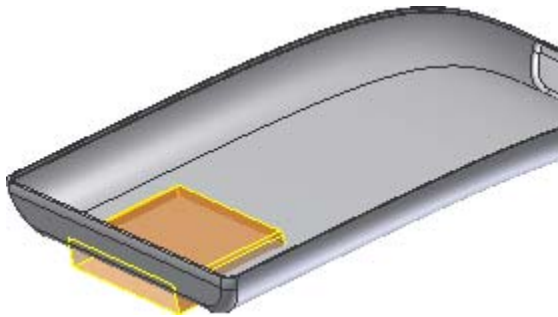


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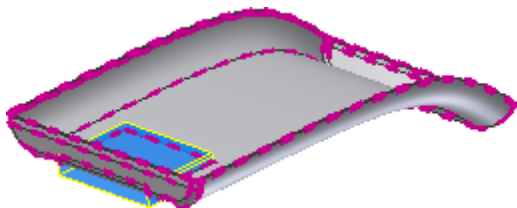
## Create a Body Using Split

We will now build a sliding cover for the battery compartment. If it is visible, turn off the visibility for the top body.

1. In the **Surface Bodies** folder, turn on the **Visibility** of **Srf2**. Note that enabling this surface also turned on the visibility of the stitched **BatterySplitSurface** in the browser. You can also enable the surface visibility by selecting **BatterySplitSurface** in the browser.



2. On the ribbon, click Model tab > Modify panel > Split.
3. Choose the **Split Solid** option in the dialog box.
4. Choose **Srf2** as the **Split Tool**, and the **bottom body** as the **Solid** to split.



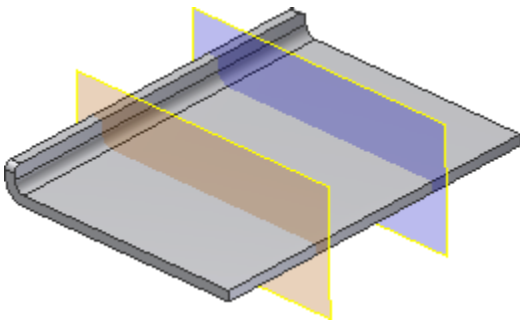
5. Click **OK** to finish.
6. A new body is created. Turn off the **Visibility** of the **bottom body** and **Srf2 (BatterySplitSurface)** before proceeding.

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## Create a Segmented Lip

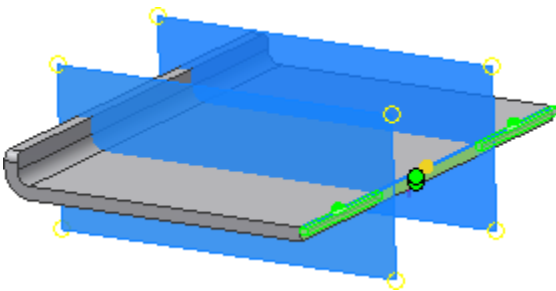
Orient the view to match the following image. We will use the **Lip** command to create a lip-groove combination on one side of the battery cover limited by two planes.

1. Create two work planes parallel to the planar side faces of the battery cover offset **-15 mm** to the inside.



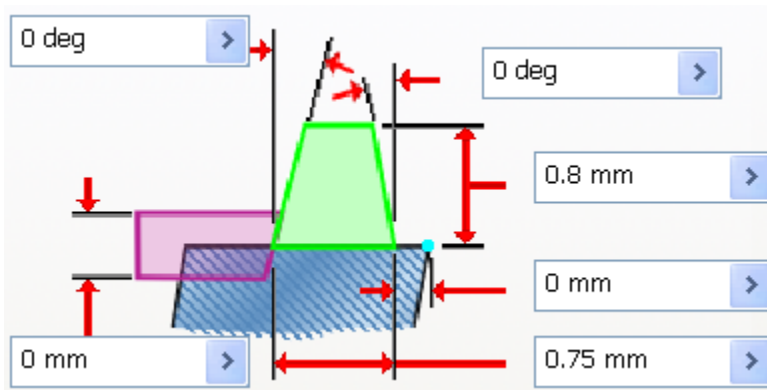
2. Click the **Lip** command.
3. On the **Shape** tab, make sure the **Lip** button is selected.
  - Select the internal edges on the far side of the curve as the **Path Edges**.
  - Select the **Guide Face** option, and pick the planar face next to the selected edge.
  - Leave the **Pull Direction** unchecked.
  - Check the **Path Extents** box, and select the two planes.

The preview shows the portions of the Lip that are selected (first and last). If the preview matches the following image, the selection is correct. If it does not match the preview, click the green and yellow dots to change the selection.



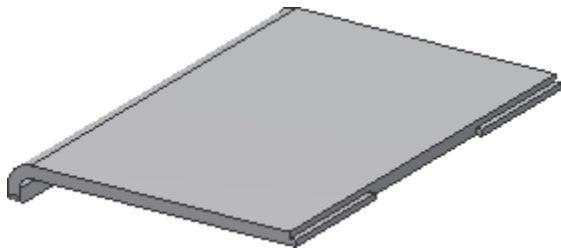
4. On the **Lip** tab:
  - For **Outside Angle**, enter 0 deg.
  - For **Inside Angle**, enter 0 deg.

- For **Height**, enter 0.8 mm.
- For **Shoulder Width**, enter 0 mm.
- For **Width**, enter 0.75 mm.
- For **Clearance**, enter 0 mm.



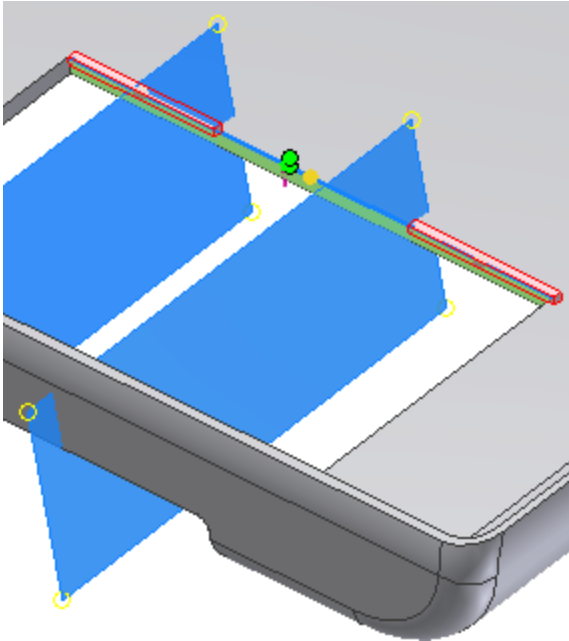
5. Click **OK** to create the lip.

The work planes are not shown in the following image. Leave the visibility of the work planes on to create the mating lip feature on the bottom body.



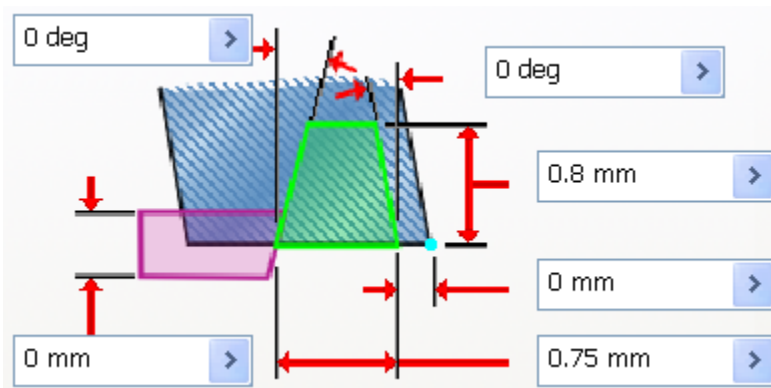
6. Turn off the **Visibility** of the battery cover, and turn on the **Visibility** of the bottom body.  
We will now create the mating groove for the battery cover on the bottom body.
7. Click the **Lip** command.
  - On the **Shape** tab, click **Groove**.
  - Select the inside edge as the **Path Edges**.
  - Select the **Guide Face**, and pick the planar face next to the edge.
  - Leave the **Pull Direction** box blank.
  - Click the **Path Extents** check box and select the two limiting planes. The default selection of the two outer groove segments is correct.





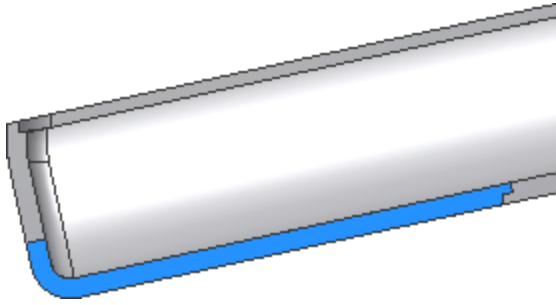
8. On the **Groove** tab:

- For **Outside Angle**, enter 0 deg.
- For **Inside Angle**, enter 0 deg.
- For **Height**, enter 0.8 mm.
- For **Shoulder Width**, enter 0 mm.
- For **Width**, enter 0.75 mm.
- For **Clearance**, enter 0 mm.



9. Click **OK** to create the groove feature. Turn off the **Visibility** of the work planes.

The following image shows the lip and groove features in a cutaway view of the battery cover and the lower body.



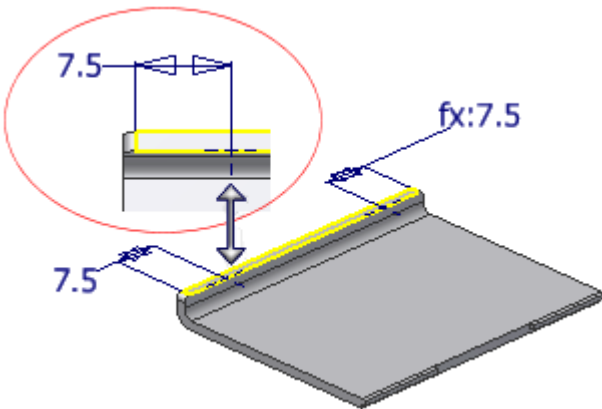
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## Create a Snap Fit

We will now build a retention mechanism on the battery cover with the Snap Fit feature. Turn off the **Visibility** for the lower body and turn on the **Visibility** for the battery cover.

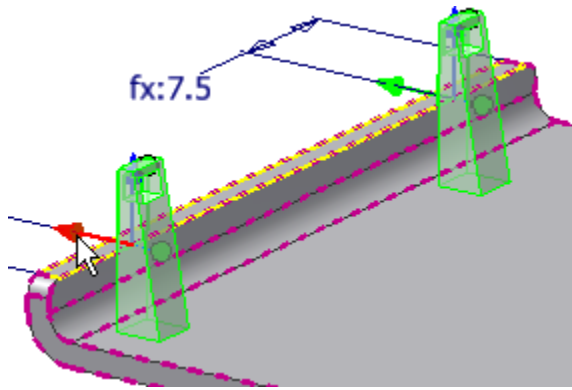
First, we will create the two positioning points for the snap feature.

1. Start a **2D sketch** on the flat wall of the narrow face, as indicated in the following image. Make sure that the internal edge is projected so the sketch points are able to be coincident with the edge.
2. Place two sketch points on the internal projected edge, and use dimensions to locate them **7.5 mm** from the sides. In the following image, the second dimension is linked to the first dimension. You can also place two independent dimensions (no fx on the dimension).



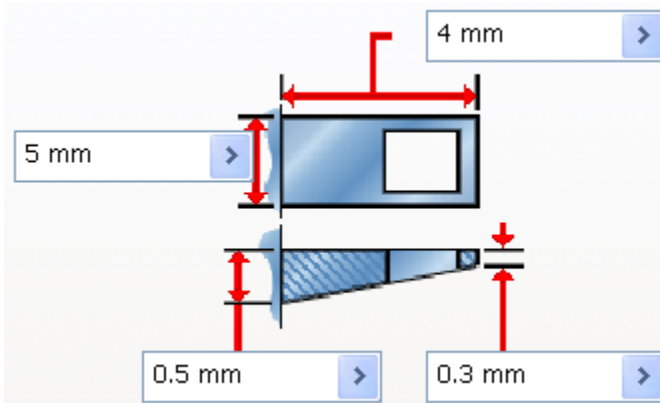
3. **Finish** the sketch.
4. Click the **Snap Fit** command.
5. Select the **Cantilever Snap Fit Loop** style.
6. On the **Shape** tab:
  - Select the battery cover as the **Solid**. If it is the only visible solid, it is automatically selected.
  - Select **From Sketch** as the **Placement** type.
  - Select the two sketch points as the **Centers**. If they are the only sketch points displayed, they are automatically selected.
  - Click the **Flip direction** arrow and the **Catch** direction manipulator arrows in the graphics

area until the clips are oriented as shown in the following image.



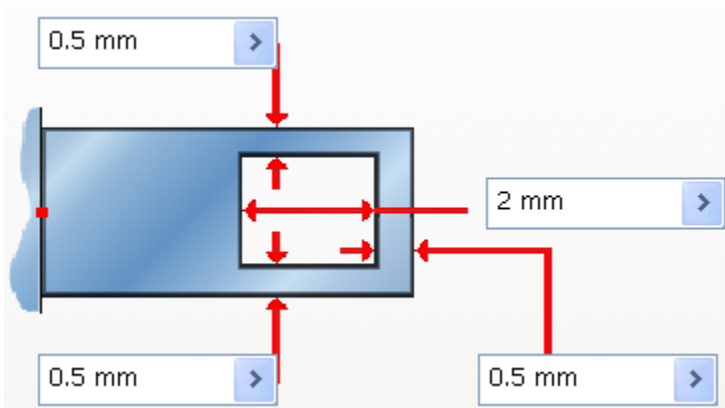
7. On the **Clip** tab.

- For **Clip length**, enter 4 mm.
- For **Clip width**, enter 5 mm.
- For **Clip thickness at wall**, enter 0.5 mm.
- For **Clip thickness at top**, enter 0.3 mm.

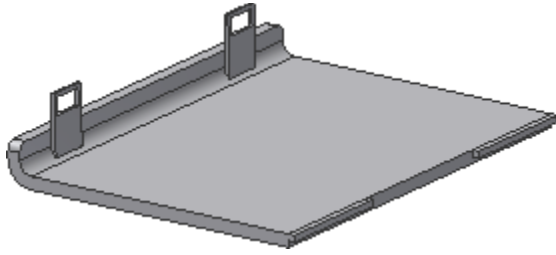


8. On the **Catch** tab:

- For **Catch width on both sides**, enter 0.5 mm.
- For **Catch opening length**, enter 2 mm.
- For **Catch width at top**, enter 0.5 mm.

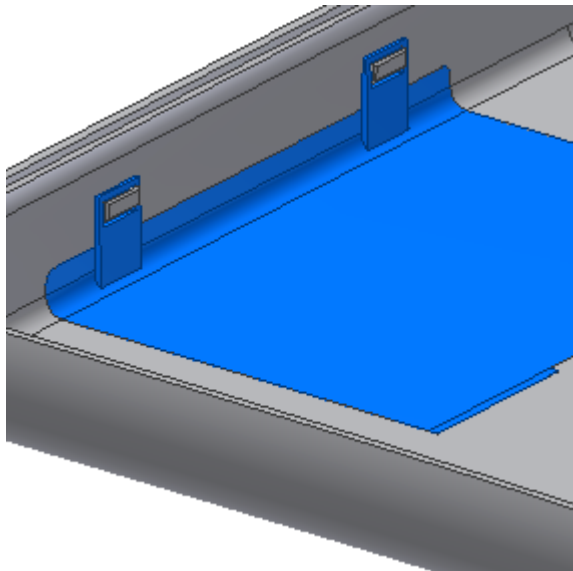
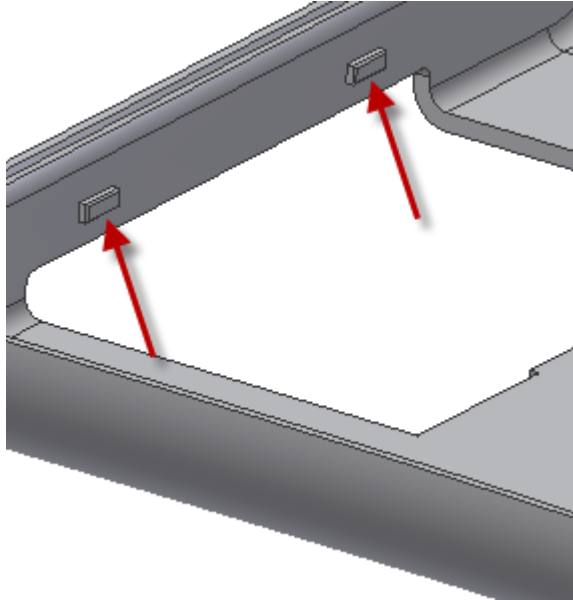


9. Click **OK** to create clips.



A trap mechanism on the bottom solid can be modeled by creating a 2D sketch on the inside face of the bottom body and using the projected edges of the catch features to create an extrusion. Fillets are added to the extrusion in this image to facilitate cover insertion and removal.

The steps to create the trap feature are not covered in this exercise.

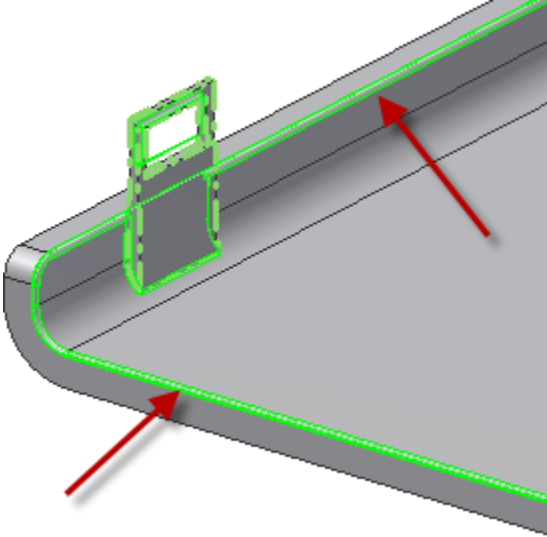


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## Add a Rule Fillet to a Feature

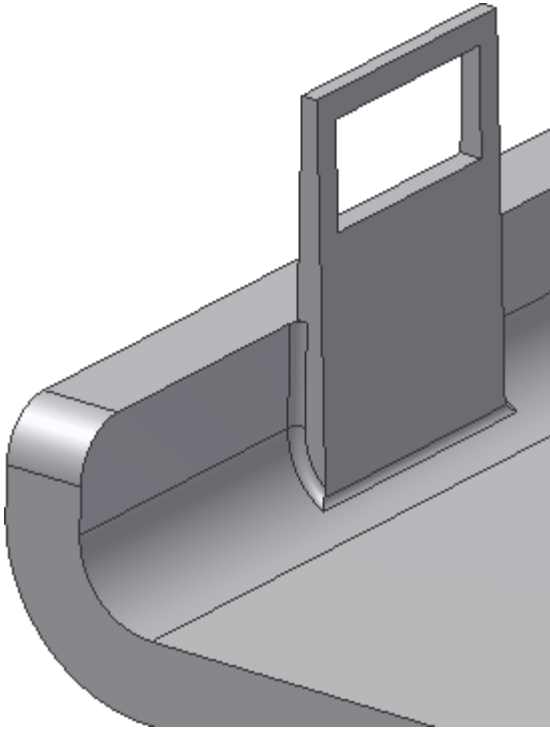
We now want to fillet the edges at the intersection between the Snap Fit and the battery cover solid. A Rule Fillet can do the job.

1. Click the **Rule Fillet** command.
  - For **Source**, select Feature.
  - Select the **Snap Fit** feature.
  - For **Radius**, enter 0.2 mm.
  - For **Rule**, select **Against Part**.



We do not want the fillet around the top edges (red arrows). The edges are included because they share the curved face of the Snap Fit that coincides with the curved face of the battery body. They are called "merged faces." We can skip such merged faces (and all the edges they share) by doing the following:

2. Click **More** to expand the dialog box.
3. Check the **Remove Merged Faces** box to enable the option.
4. Click **OK** to create the rule fillet. Both clips are filleted since they both belong to the same feature.
5. **Save** the file.



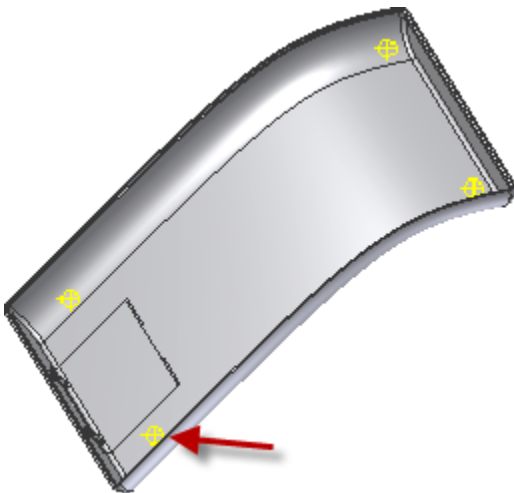
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## Create a Boss

This exercise consists of two procedures: creating screw mounting bosses on the top and bottom bodies, and then creating mating bosses for the thread portion of the fasteners.

First, we use work points to model the screw mounting bosses on the top and bottom bodies.

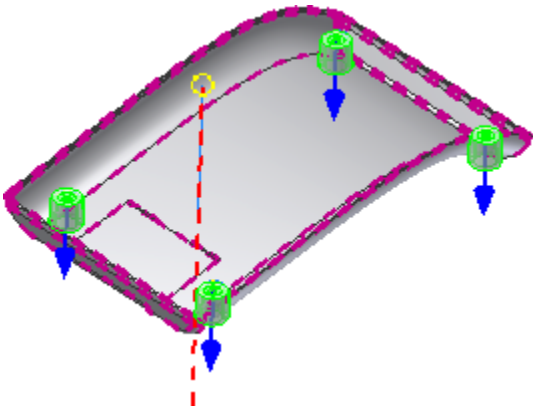
1. Turn **on** the **Visibility** of the bottom body and Work Points 1-4. The work points are located at the termination position of each boss.



2. On the ribbon, click Model tab ➤ Plastic Part panel ➤ Boss.
3. Make sure that the **Head** button is selected.

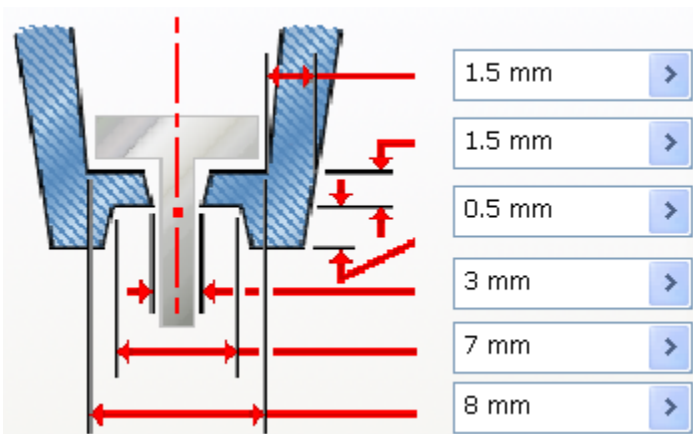
4. On the **Shape** tab:

- In the **Placement** area, select **On Point** from the drop-down menu.
- Select the four work points as the **Centers**.
- For **Direction**, select the **Y Axis**. **Flip** the direction if the arrows do not point towards the body.
- For the target **Solid**, select the bottom solid.
- For **Fillet**, enter 0.3 mm.



5. On the **Head** tab:

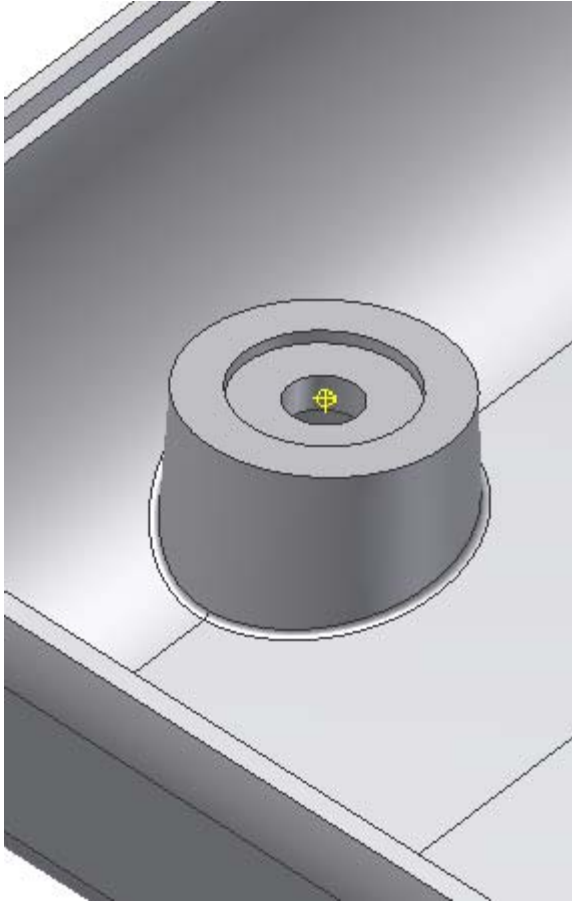
- For **Wall Thickness**, enter 1.5 mm.
- For **Shank Height**, enter 1.5 mm.
- For **Clamp Height**, enter 0.5 mm.
- For **Shank Diameter**, enter 3 mm.
- For **Clamp Diameter**, enter 7 mm.
- For **Head Diameter**, enter 8 mm.
- For **Draft Options**, enter 2.5 deg in each field.



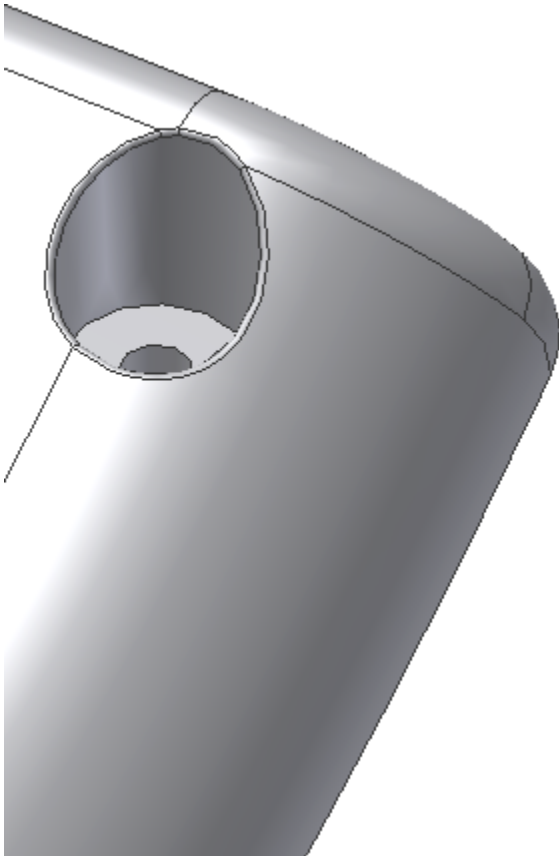
- Select the **Counterbore** type.

6. Skip the **Ribs** tab.

7. Click **OK** to create the four bosses.



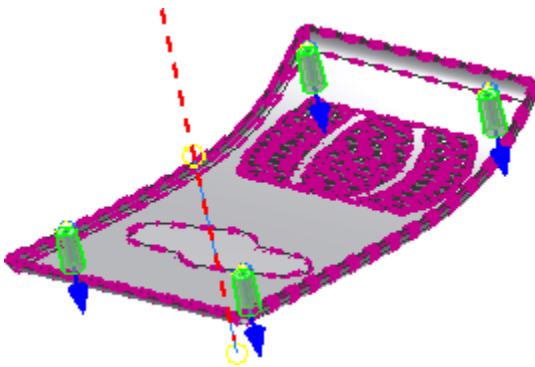
8. Rotate the body to see the recess for the screw head.



Next, we build the mating bosses for the thread portion of the fasteners.

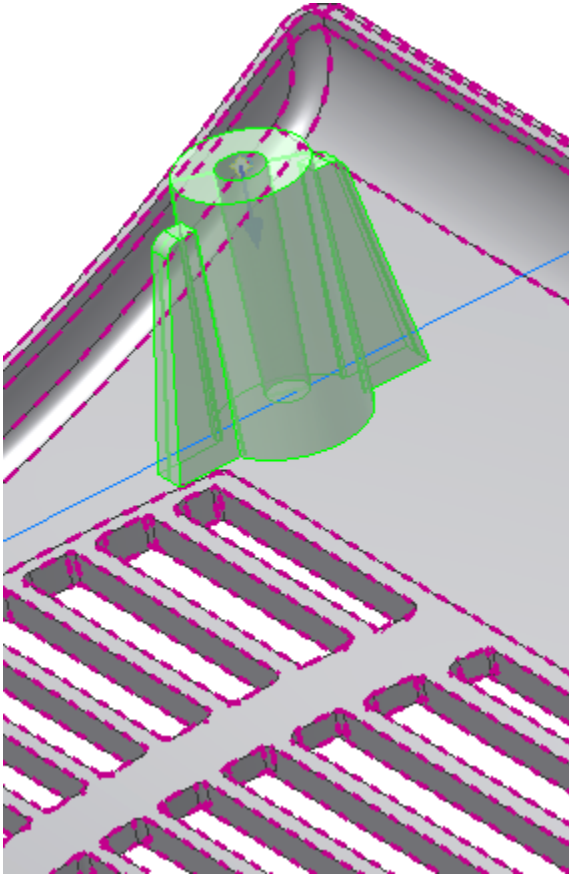


1. Turn off the **Visibility** of the battery cover and the bottom solid, and turn on the **Visibility** of the top solid.
2. Click the **Boss** command.
3. Make sure that the **Thread** button is selected.
4. On the **Shape** tab:
  - In the **Placement** area, select **On Point** from the drop-down menu.
  - Select the four work points as the **Centers**.
  - For **Direction**, select the **Y Axis**. **Flip** the direction if the arrows do not point towards the body.
  - For the target **Solid**, select the top.
  - For **Fillet**, enter 0.3 mm.

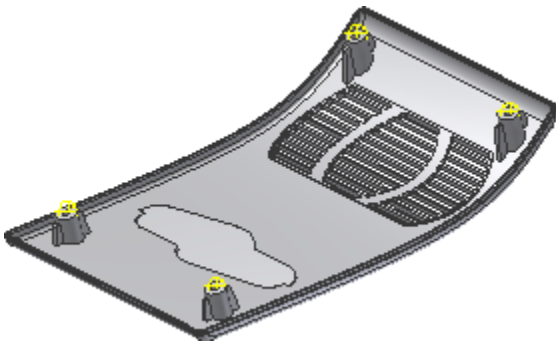


5. On the **Thread** tab:
  - Ensure that **Hole** is selected, and select **Full Depth** from the drop-down menu.
  - For **Thread Diameter**, enter 8 mm.
  - For **Thread Hole Diameter**, enter 3 mm.
  - For **Inner Draft Angle**, enter 1 deg.
  - For **Outer Draft Angle**, enter 2 deg.
6. On the **Ribs** tab:
  - Ensure that **Stiffening Ribs** is selected, and select **2 ul** from the drop-down menu.
  - For **Rib Thickness**, enter 1.5 mm.
  - For **Rib Draft**, enter 1.5 deg.
  - For **Shoulder Length**, enter 6 mm.
  - For **Top Offset**, enter 2 mm.
  - For **Shoulder Radius**, enter 1 mm.
  - For **Shoulder Flare Angle**, enter 10 deg.
7. Expand **Fillet Options** on the **Ribs** tab:
  - For Rib Radius, enter 0.2 mm.

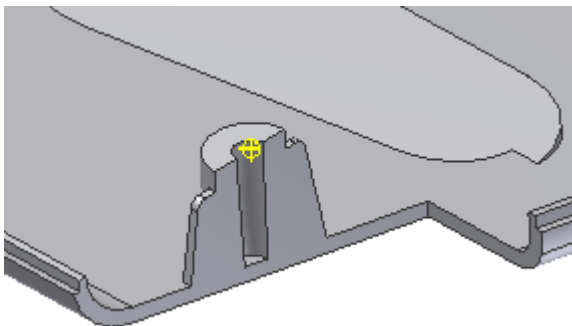
- For **Blend Radius**, enter 0.2 mm.
- For the **Ribs Start Direction Angle**, enter 0.
- For **Direction**, select the **X Axis** in the **Origin** folder.



8. Click **OK** to create the four bosses with stiffening ribs.



The following image shows a cutaway view of a boss.



9. If you didn't change the body colors earlier in the exercise, right-click each body in the browser,

and then select **Properties** from the context menu. Assign a unique body color in the **Body Color Style** drop-down menu.

10. **Save** the file.

The plastic features portion of this tutorial is complete.

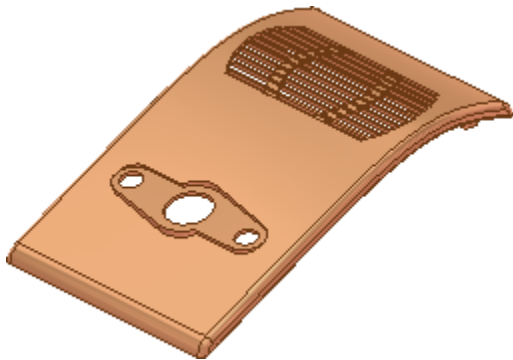
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## Add Holes to a Single Body

In the next section, we create new holes in a single body using the **Hole** command.

1. Turn off the **Visibility** of all solid bodies except the top cover.
2. Create two **10-mm** holes with a depth of **2 mm** on the ends of the rest feature.
3. Create one **20-mm** hole with a depth of **2 mm** at the center of the rest feature.

Notice Autodesk Inventor selects the body being referenced as the default participant.



**Note** To add participants to the **Hole** operation, use the **Solids** selector and pick additional bodies. An example of this could be if you want multiple bodies to participate in a “Through All” operation.

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## Insert a Toolbody Using Derive

You can create solid bodies within a part file and use them as toolbodies in the **Combine** command for a cut, join, or intersect operation. You can also use another component placed in the part file using the **Derive** command.

We will now use the **Derive** command to import another part file to use as a cutting tool later in the exercise.

**Note** If you skipped the plastic features sections, you can open the file **PFTutorial\_Revolve\_Combine.ipt** and begin the tutorial here.

1. On the ribbon, click Manage tab ➤ Insert panel ➤ Derive.

2. In the file open dialog box, select the file **Control\_Button\_Solid.ipt**.
3. In the Derived Part dialog box, you can select any of the solid body options, but do not select the surface feature option.  
**Note** If the component you are inserting is an assembly, and you choose to maintain each solid as a solid body, the result is multiple bodies in the **Solid Bodies** folder.
4. Click **OK** to finish.
5. The new body is inserted in the part. Rotate the part to view the new body.



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## Moving Bodies

In this section, we will use the **Move Bodies** command to position the toolbody accurately we imported.

The drop-down menu in the Move Bodies dialog box offers three methods for moving a body:

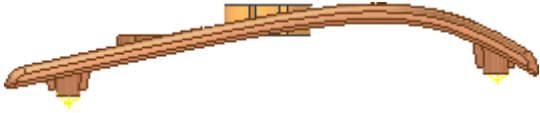
- **Free drag** (default) - Use to drag the selection in any direction or specify precise x,y,z values.
- **Move along ray** - Axial move only. Use to drag or specify a precise value along an axis.
- **Rotate about line** - Use to drag the selection or specify a precise angle around a central axis.

1. View the part from the side to expose the toolbody.
2. On the ribbon, click Model tab > Modify panel > Move Bodies.
3. Select the imported body as the body to move.
4. The body shows a 10 mm offset in the preview. Do not drag the preview.



5. Input the following values:
  - For **X Offset**, enter 35 mm.
  - For **Y Offset**, enter 0 mm.

- For **Z Offset**, enter 0 mm.
6. Select **Click to add**, and input the following values:
    - For **X Offset**, enter 0 mm.
    - For **Y Offset**, enter 24 mm.
    - For **Z Offset**, enter 0 mm.
  7. Click **OK** to move the body and exit the command.



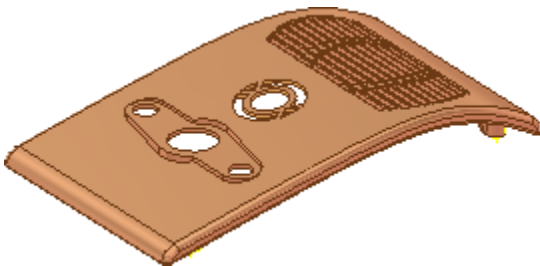
**Note Move Body** appears in the browser as a feature, and the move was calculated as a single feature. Using **Click to add** consumes the least amount of resources as opposed to doing multiple moves as individual features. You can edit, delete, or suppress individual body moves.

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## Use the Combine Command

The **Combine** command provides a means to perform Boolean operations within a part file. In this exercise, we will use the imported toolbody to cut out a portion of the upper body.

1. On the ribbon, click Model tab > Modify panel > Combine.
2. Select the upper body as the **Base**.
3. Select the imported body as the **Toolbody**. Make sure that **Keep Toolbody** is unchecked.  
**Note** You can select more than one body to use as toolbodies. The number in parentheses indicates the total number of selected toolbodies. If **Keep Toolbody** is unchecked, the toolbody is consumed and cannot be used for further operations. If it is checked, the toolbody is available for further operations.
4. Select the **Cut** operation.
5. Click **OK** to cut the toolbody from the top cover and finish the command.

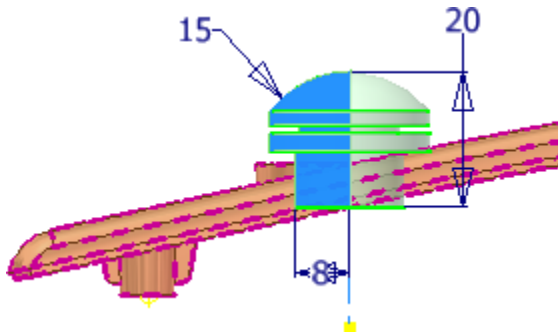


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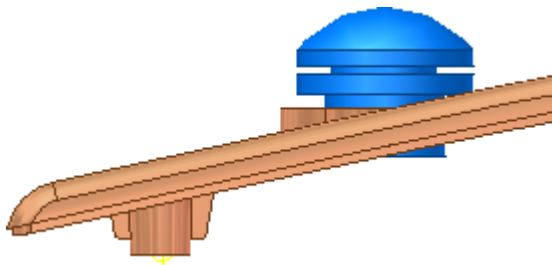
## Create a Body Using Revolve

In this exercise, we use the **Revolve** command to create a body in the part file.

1. In the browser, turn on the Visibility of the **Revolve\_NewBody** sketch.
2. Start the **Revolve** command, and select the profile and axis if necessary.



3. Select **New Solid** as the modeling operation to perform.
4. Click **OK** to finish the command and create the solid.
5. Select the new body in the browser, and change the body color.



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## Export the Design as Individual Parts

A multi-body part is a single part file. While it is a great way to control a design, you cannot create a parts list for the bodies in a multi-body part. You can use the **Make Part** or the **Make Components** command to export bodies as part files.

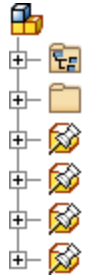
In this final segment, we will use the Make Components command to export all the bodies as derived parts into an assembly.

1. On the ribbon, click Manage tab ► Layout panel ► Make Components.
2. Select all four of the **solid bodies** for export. You can select the bodies in the graphics area or in the **Solid Bodies** folder in the browser. Make sure that all bodies to be exported are visible.  
**Note** Pay attention to which body highlights as you add selections. It helps you to assign the proper part names when you export them.
3. Assign the target assembly a unique name. Leave all the other selections at the default settings and select **Next** to continue.
4. Assign each body a new part name.
5. If you completed the tutorial in sequence:
  - The first body is the top cover

- The second body is the battery cover
- The third body is the bottom cover
- The fourth body is the revolved control button.

6. Click **OK** to create the assembly.

The assembly opens in a new window. All the bodies are present in the browser as grounded part files.



The part file controls the master design. If changes are made to a body in the originating part file, the part will update in the assembly.



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## Summary

In this tutorial, you:

- Used **Split** to create new bodies.
- Created plastic features using the Plastic Part commands.
- Used the **Hole** command to modify a single body.
- Inserted a toolbody using the **Derive** command.
- Used the **Move Bodies** command to position a body.
- Used the **Combine** command to perform a cut operation on a body.
- Created a body in place using the **Revolve** command.

- Used **Make Components** to export bodies into an assembly as individual part files.

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