

Plastic Part Design Tutorial

This tutorial will demonstrate some of the plastic part design capabilities of IronCAD. It is intended to introduce basic techniques and is not meant to represent the full capabilities of IronCAD. Although it is a stand alone example, this tutorial assumes the reader is familiar with the basic modeling concepts introduced in the IronCAD User Guide.

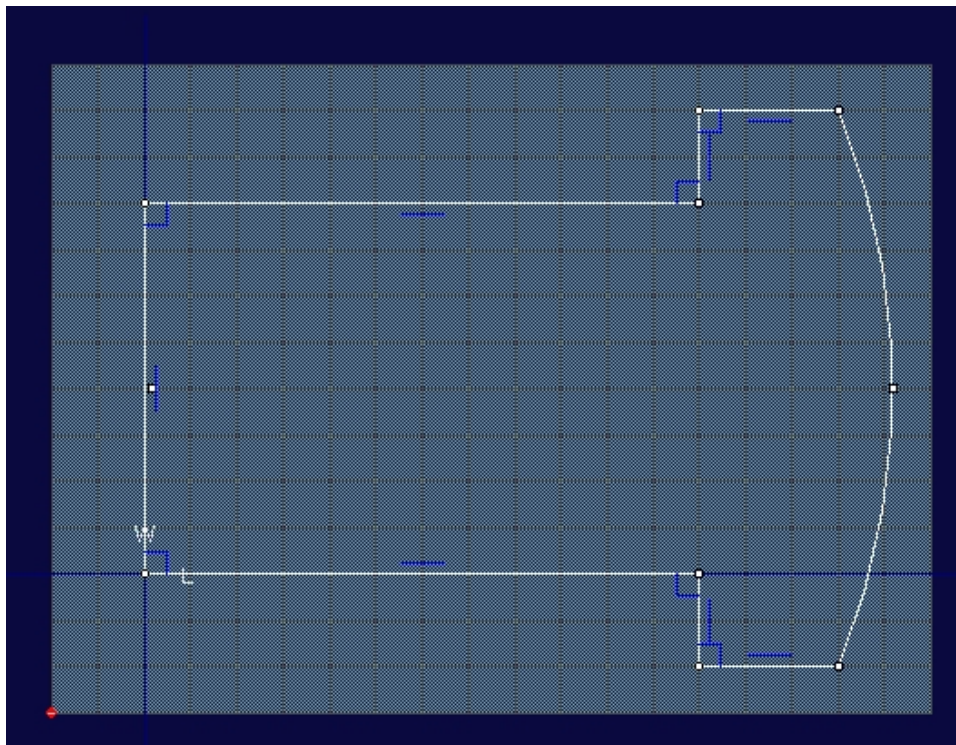
Topics covered in this tutorial :

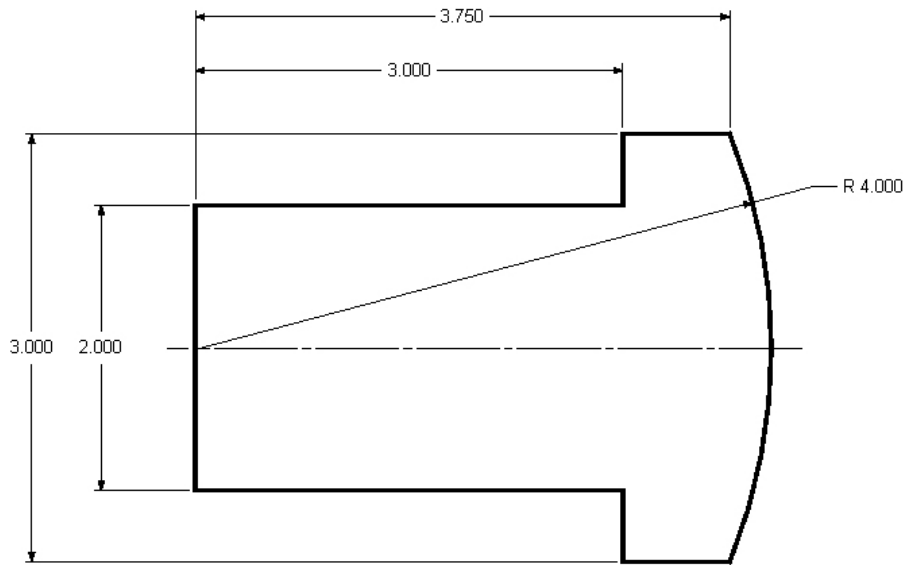
- Part Tapering
- Part Blending
- Part Shelling
- Use of Catalogs to ease design tasks
- Creating mating parts

NOTE: For this tutorial, set the Default kernel to Parasolid. From the Tools menu, select Options and the Parts tab, then select Parasolid.

Profile layout and part tapering

- From the **File** menu, select **New**, and then **Scene**. From the Workspace(English) tab, select the template "**Blank Scene**" and then select **OK**. You will need to leave the units in **Inches** for the entire tutorial.
- From the **Create** menu, select **Intellishape**, **Extrude** and click inside the viewport. Or use the **Extrude** icon in the Intellishape Creation toolbar.
- In the Extrude Shape Wizard, select Stand alone, **Next**, **Next**, "To specified distance" equal to **.75**, **Next**, "Do you want to display the drawing grid" **Yes**, grid spacing **.25** for both horizontal and vertical and **Finish**.
- Create a profile similar to the one below and select **Finish Shape** (see drawing on the next page for dimensions):

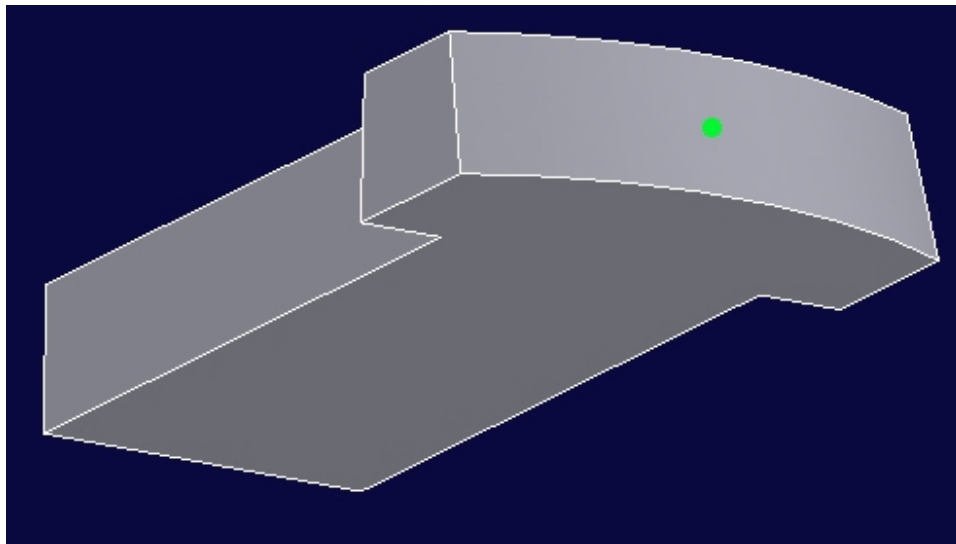




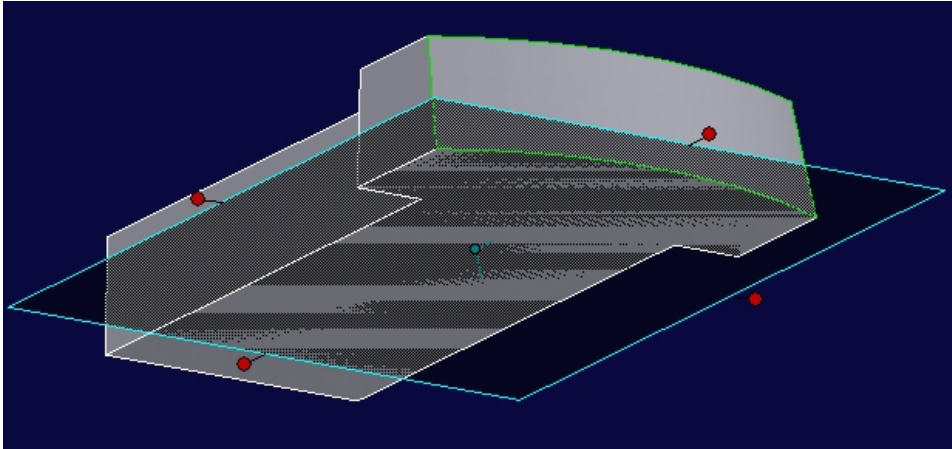
Tapering the part

The Taper command provides the capability to put “draft” on your part. Draft is required by most molding processes to facilitate part removal from the mold.

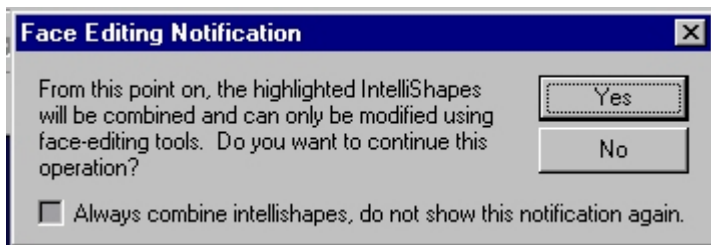
- Select the part’s Intellishape, right-click and select **Surface Reshaping Properties...**
- Select “Which Surface?” **Side, Taper**, set **Tilt angle** to -5 , select **OK**.
- Rotate the viewport as shown, select the **Taper Face** icon from the Face/Edge Edit toolbar and click on the curved surface.



- Select the **Create draft plane** icon from the Taper toolbar and click the bottom face of the part. The draft plane should appear as shown.



- Enter -10 for the angle and select the green OK dot.



Answer Yes to this question, our single Intellishape will combine with the new taper and allow us to continue with our design.

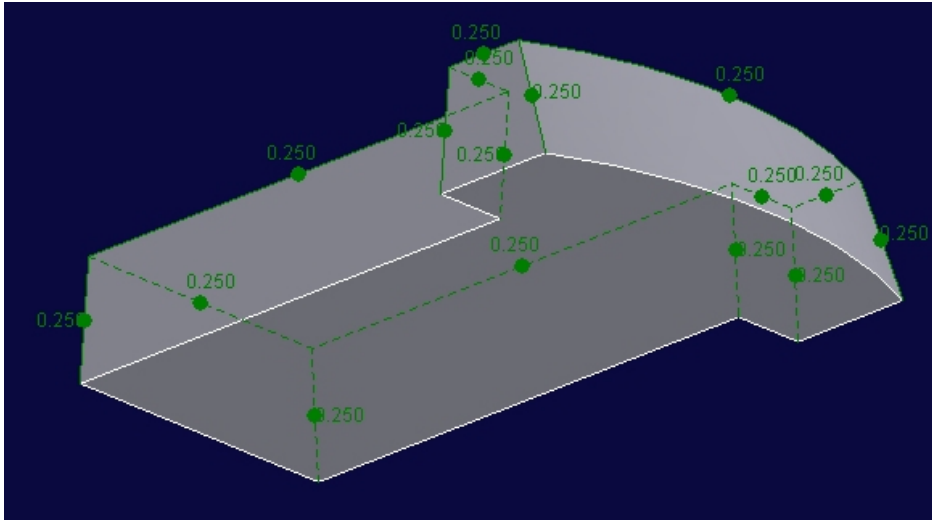
The curved face now has a 15 degree taper while the other faces retain their 5 degree taper.

Blending the part.

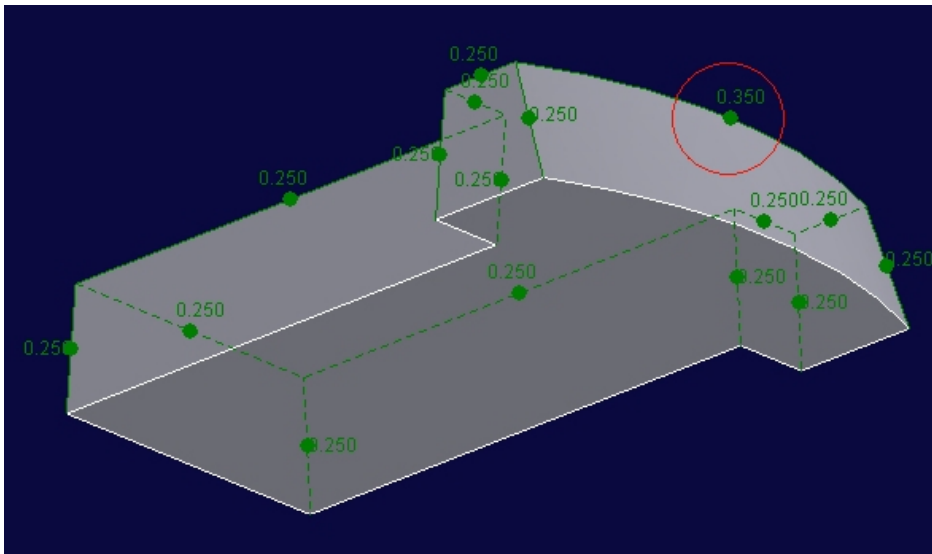
IronCAD's Blend command is flexible and easy to use for the important blending needs of plastic parts for smooth flow of material during the molding process.

- Select the **Blend** icon in the Face/Edge Edit toolbar, select the **Box Select** icon in the Selection toolbar and box around the entire part.
- Enter .25 for the blend radius value.
- Now click the bottom face and select the **Clear Blend** icon, the part should appear as shown.

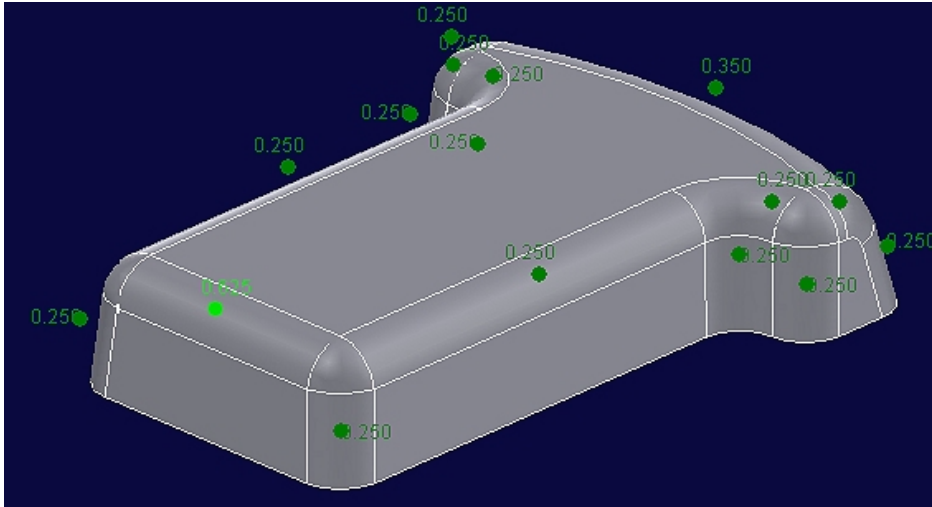




- Now select the green dot on the curved edge and change the blend value to .35.



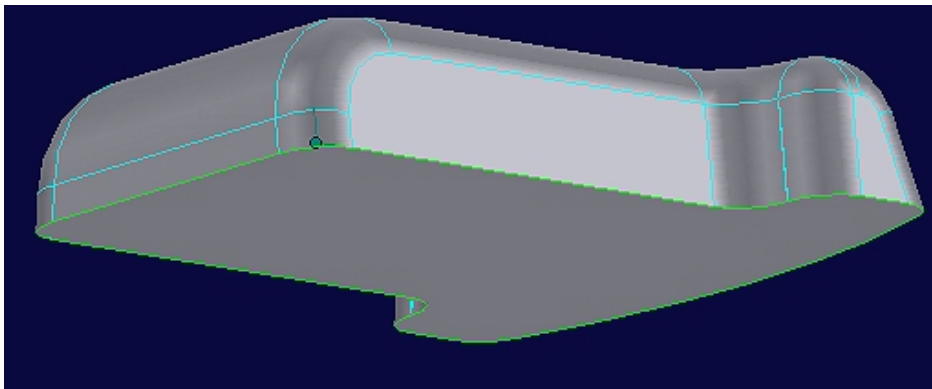
- Select the green OK dot to generate the blends.
- Now blend modifications are done easily, click one of the blend faces twice (it will highlight yellow), right-click the blend and select Edit Shape.
- Click the green dot on the back blend as shown, change the value to .625 and select the green OK dot.



Shelling the part

The Shell command provides fast, intelligent shelling for parts.

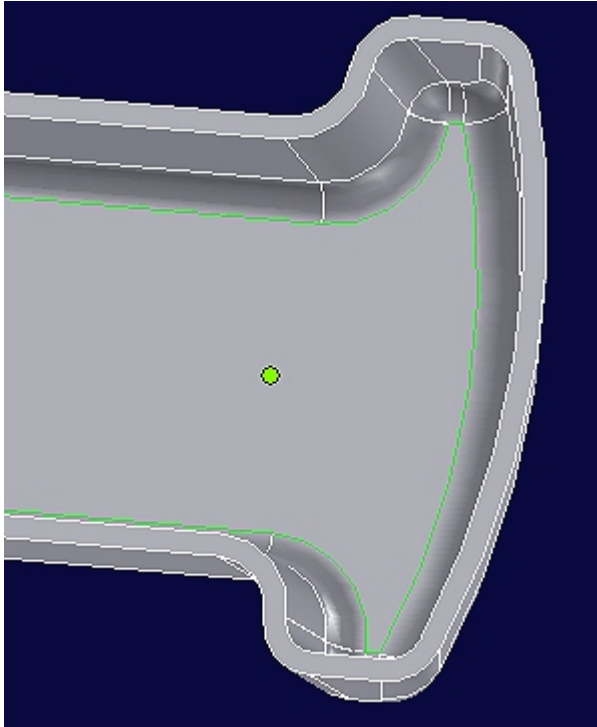
- Select the part (blue highlight), right-click the part and select Shell.
- Click on the bottom face as shown, change the Thickness to .1 and select the green OK dot.



Using Catalogs

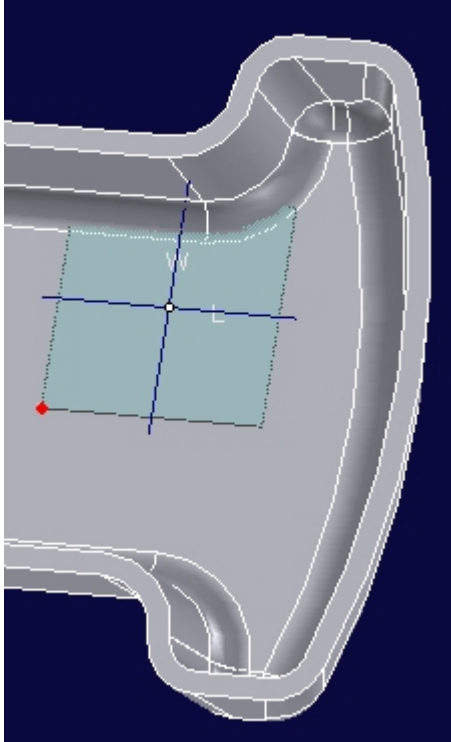
Designers very often use features that are common to most plastic parts. Fastener bosses, snap fits, etc. IronCAD's Catalogs are a great way to capture those features in a drag-and-drop ready library.

- Select the **Extrude** icon from the Intellishape Creation toolbar, click the center of the arc face as shown.



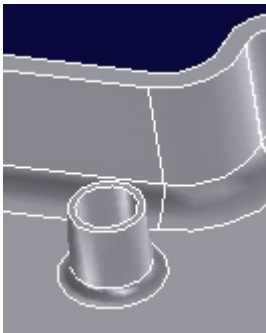
- In the Extrude Shape Wizard, select **Add Material, Next, Next**, “To specified distance” equal to **.375, Next**, “Do you want to display the drawing grid” **No** and **Finish**.
- Now select the Triball tool, move the drawing grid **.4** as shown and deselect the Triball.





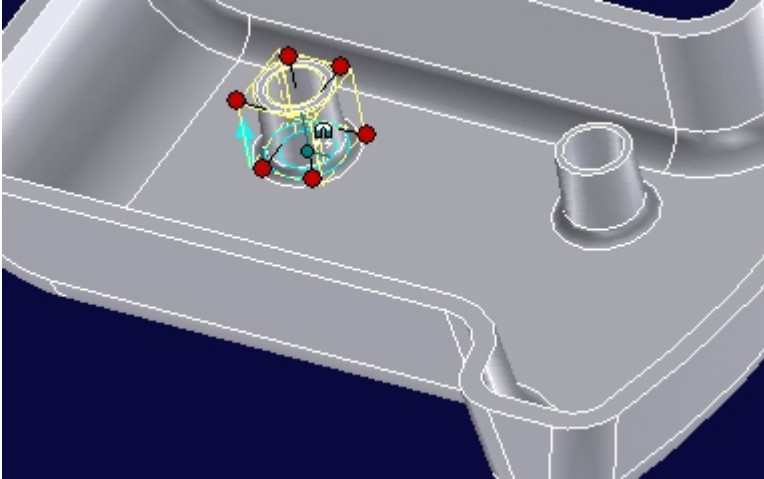
- Create two circles at the intersection of the blue axis, .125 and .185 radius and select Finish Shape.
- Select the new Intellishape, right-click and select **Surface Reshaping Properties...**, Select “Which Surface?” **Side**, select **Taper** and set **Tilt angle** to -2 .
- Go to the **Bevel** tab, “Which edges?” select **All Intersection Edges**, **Blend**, **Radius** set to .063 and select OK.

Now you have a boss that can be drag-and-dropped into a new Catalog and used in other places on this part or on a new part.

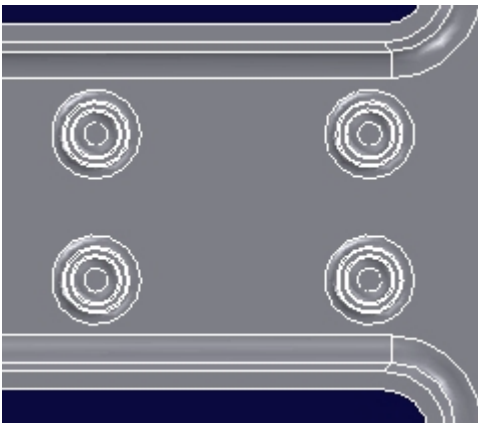


- Go to the Catalogs menu and select New.
- Select the new Intellishape, drag-and drop it into the new Catalog.

- Go to the new Catalog and drag-and-drop the new shape onto the inside face as shown.



- Now drag-and-drop two more shapes and position them approximately as shown. You can use the TriBall to position them accurately.

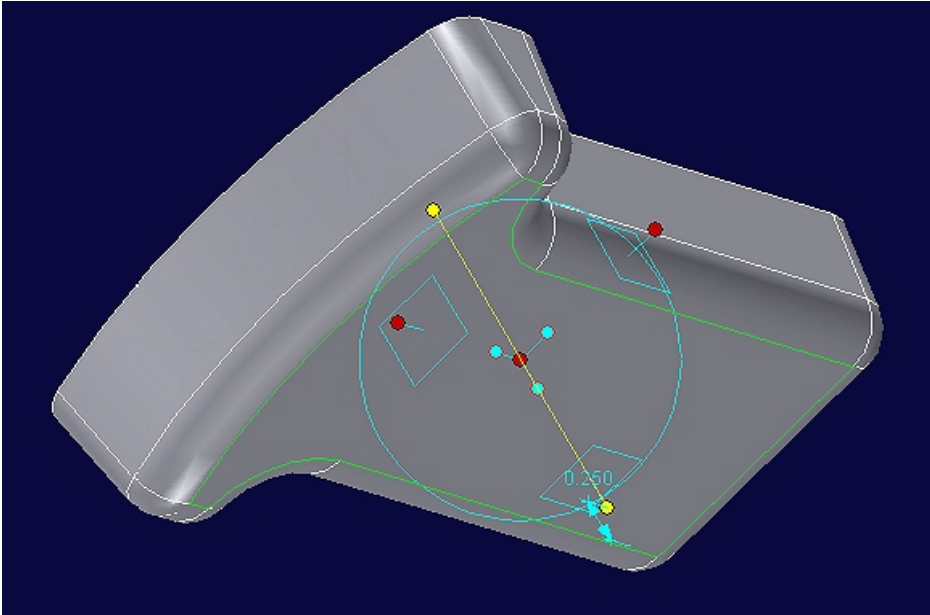


Creating mating parts

A great benefit of solid modeling is the ease in which mating parts can be created from existing part geometry.

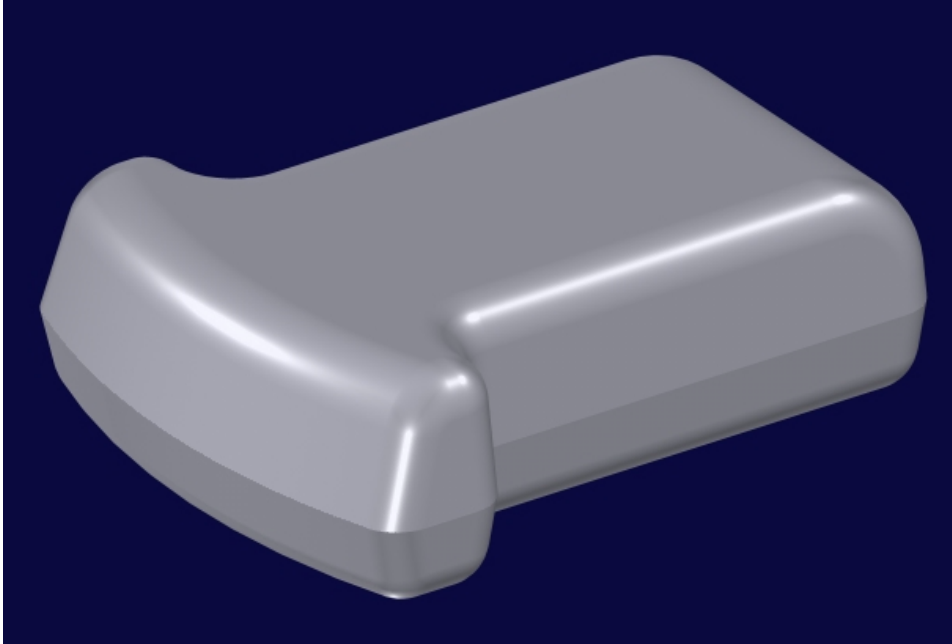
- Select the part in part mode (blue highlight), go to the **Edit** menu, select **Copy** then **Paste** to create a copy of the part.
- Go to the **Modify** menu, **Mirror, About Height**. Suppress the original part.

- Now to modify the new part, remove the four boss features, edit the blends by right-clicking the yellow highlighted blend, change the .625 and the .35 blend to .25, select the green OK dot.
- Select the **Move Face** icon in the Face/Edge Edit toolbar, select the bottom face as shown and decrease the height of the part by .25.



- Unsuppress the first part by right-clicking it's icon in the Scene Browser and unchecking **Suppress**.

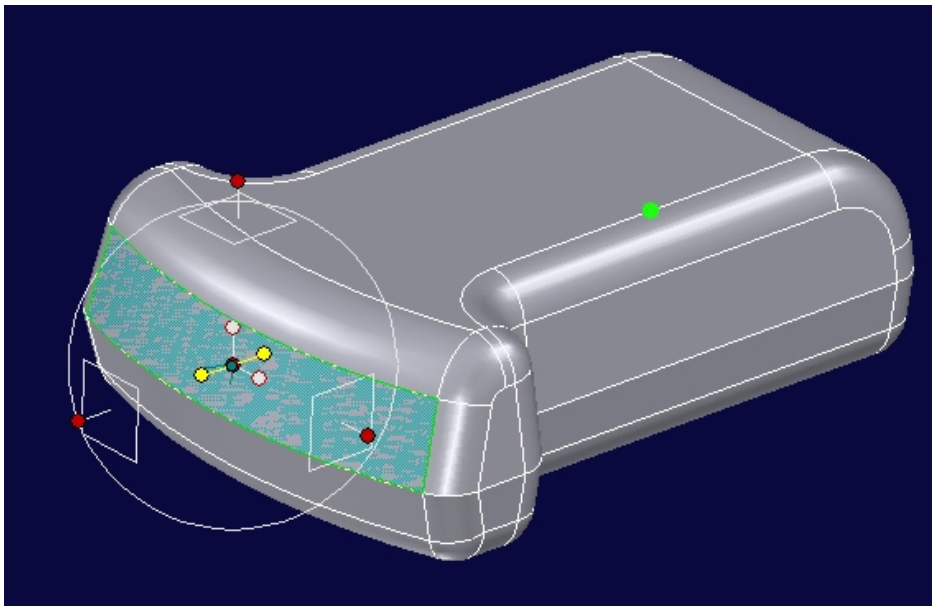
You now have a basic plastic part design that should look like the one below.



Modifying existing parts

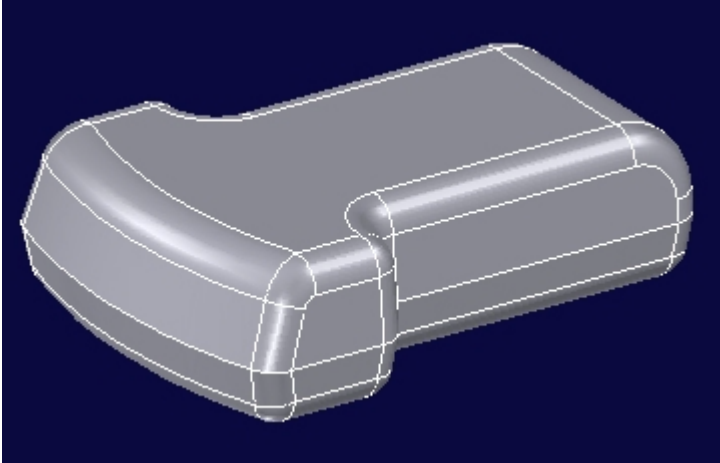
As a designer, you have probably worked on someone else's design, and sent your designs to other team mates. With IronCAD, collaboration becomes enjoyable like never before. Most CAD systems make it difficult at best to work with someone else's design due to constraint limitations and history dependence. With IronCAD you don't have to know the design history to make productive use of the design. Our Direct Face modeling allows you to make virtually any change to any geometry.

- Select the **Move Face** icon, click the face shown, select the Triball (F10), press the space bar to temporarily detach the Triball from the part (it highlights white), now right-click the Triball orientation handle shown, select **Parallel to Edge** and click the edge shown.



- Press the space bar to reattach the Triball and pull the face out .5.
- Repeat the above steps for the mating face on the lower part.

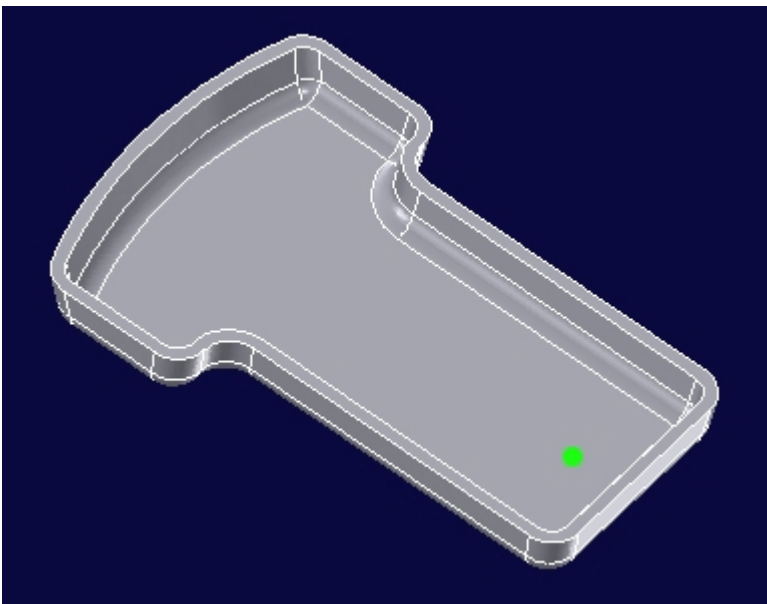
You should now have the two parts as shown below.



Adding ribs

IronCAD has a nice tool to create intelligent ribs. We'll add a rib to the smaller of the two parts. Let's rename them first.

- Select the Scene Browser and rename the larger part **top cover** and the smaller part **bottom cover**.
- **Suppress the top cover.**
- Select the **Tools** Catalog, drag-and-drop the **Rib** shape onto the bottom cover at the green dot shown.



- Set the following sizes in the **SmartRib** dialog: Rib Thickness: .075
Rib Height: .25
Tapering Angle: 2
Rib Direction: Along Width
- Use the Triball to make 2 copies of the rib spaced .85 apart. The bottom cover should now look like the part below.

