KeyCreator Lesson KC2857

A Molded Plastic Bottle Handle

Every so often I come across a really wild part to model that is part of a common, everyday item. The molded plastic handle illustrated to the right was part of a five gallon bottled water assembly. The handle snaps into formed indentations in the clear water bottle. The part features midplane draft and an interesting rib geometry that makes it lightweight yet extremely rigid.





Let's start with a new file in View 2. (The Front View.)

Create a circle that is 12 inches in diameter centered at the origin and a second, concentric circle that is 10.5 inches in diameter. Create two more concentric circles 8.5 and 7.75 diameter.





Next, create a single vertical line centered on the circles.

Then click on the CREATE LINE AT AN ANGLE Icon. Type 20 for the Angle and click on the bottom end of the vertical line.

Then, using the CtrMid Option, click on one of the circles. Repeat, creating an oblique line at -20 degrees.



Your screen should look like this:





Click on the TRIM DOUBLE Icon. Trim each of the four circles with the two oblique lines.

Your construction should now look like this:



Click on the CREATE LINE AT AN ANGLE Icon. Type 3 for the Angle.

Click on the bottom end of the right, oblique line and then, using the EndEnt Option, click on the bottom end of the line again.

Click on the BACKUP Button, click on the bottom end of the line you just created and click on the same spot again.

Two more times, click on the BACKUP Button, click on the bottom end of the line you just created and click on the same spot again.

You will now have four more oblique lines on the screen. (I've shown them in a contrasting color in the illustration to the right.)





Click on the FILLET WITH TRIM Icon.

Type 0.25 for the Radius.

Now create fillets at the intersections of the topmost arc and the outermost oblique lines.

Then, create fillets at the intersections of the arc just below the topmost arc and the outer, oblique lines so you get a closed profile like the CYAN one illustrated to the right.



Click on the TRIM DIVIDE Icon. Click on the third arc from the top, between the second and third long oblique lines, then click on the second and third long oblique lines.

Using the FILLET WITH TRIM COMMAND, create 0.1 fillets at the four intersections shown in the illustration to the right.

Then, create 0.05 fillets at the two intersections shown.





Click on the TRIM FIRST Icon. Click on the lower part of the leftmost, long oblique line and then on the topmost arc.

Repeat, clicking on the lower part of the rightmost, long oblique line and then on the topmost arc.

Click on the CREATE LINE BY END POSITIONS Icon. Using the EndEnt Option, click on the top ends of the two oblique lines that you just trimmed.





You can delete the long vertical line left from your original construction.

You will now have two distinct profiles. To simplify the following steps the profiles should be different colors. Mine are Cyan and Deep Red.





For the next step, make sure that Restrict Chain Select and Enable Quick Chain are selected in TOOLS/OPTIONS/SELECT.

Select a third construction color.





Click on the XFORM CHAIN OFFSET Icon.

A Dialog Box appears. Select the Distance-Depth Option.

Type 0.07 for the Distance and 0 for the Depth.

Select the Fillet Arc Option under cornering and hit the ENTER Key.

Move the cursor over the profile with the two large arcs and click when the entire profile highlights.

Then, click on the vector that faces into the interior of the profile.

Select a fourth construction color and still using the XFORM CHAIN OFFSET Function, move the cursor over the smaller profile that is shaped like an inverted "T." When the entire profile highlights, click on it and then click on the vector that points into the profile.

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Distance	0.07
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Cornering Linear S Fillet Arc	



Your screen should now look like this:

Now, switch to the Isometric View. (View 7.)





Click on the EXTRUDE Icon. A Dialog Box appears.

Type 0.3 for the Length. Type 2 for the Draft Angle. Select the Outward Option and hit the ENTER Key.

Select the outer large profile with the broad arcs and hit the ENTER Key. Then, click on the rearward-facing vector.

Next, select the outer "Inverted T" profile and hit the ENTER Key. Then, click on the rearward-facing vector.

Your screen should look like this:





Next, click on the CUT Icon.

A Dialog Box appears. Select the Forward Option. Use the Through All Condition.

Type 2 for the Draft Angle and select the Inward Option. Hit the ENTER Key.

Select the larger solid. Then, select the matching profile that you created with the Chain-Offset Function. Click on the rearward-facing vector.

Repeat this process with the inverted, "T-Shaped" solid and the matching profile there.

Your screen should now look like this:





Next, click on the XFORM ROTATE COPY Icon. Click on the "T-shaped" solid and hit the ENTER Key.

Type 1 for the Number of Copies. Using the CtrMid Option, click on the bottom, front circular edge of the large solid and then the bottom, rear circular edge of the large solid.

Type -12 for the Angle.



Your screen should now look like this:



Click on the TRIM SOLID TO SOLID Icon. A Dialog Box appears.

Click on the First Body Only Option and the All Portions Option and hit the ENTER Key. Now, click on one of the "T-shaped" solids and then on the large solid. Next, click on the other "T-shaped" solid and then on the large solid.

Click on the SEPARATE Icon.

Then, click on each of the "T-Shaped solids." (These solids were turned into disjoint, non-manifold solids by the previous trim operation. This breaks them into individual solids.)



Now, delete the lower piece of the left, "T-shaped" solid and the four top sliver pieces.





Switch to the Front View. (View 2.)

Then, click on the XFORM MIRROR COPY Icon.

Select all of the pieces from the trimmed "T-shaped" solids and hit the ENTER Key.

Click on the 1 Pos V Option on the Conversation Bar. Using the CtrMid Option, click on one of the large arcs on the main solid.





This will duplicate the trimmed solids on the left side of the part.

Click on the BOOLEAN UNION Icon.

Select the main solid. Then click on the ALL DSP Option and the ALL Option and hit the ENTER Key.

Our next task is to add 0.03 constant radius blends to every internal corner on the part. To do this quickly, first, switch to the Top View. (View 1.)





Next, click on the CONSTANT RADIUS BLEND Icon. Type 0.03 for the Radius.

Using the Window, Part In Option on the Conversation Bar, drag a selection window like the one illustrated below and hit the ENTER Key. All of the blends are done in one shot!





Next, click on the XFORM MIRROR COPY Icon.

Select the solid and hit the ENTER Key. Click on the 1 Pos H option and click on the top edge of the solid using the EndEnt Option.

This gives you the rear half of the part.

Use the BOOLEAN UNION Function again to join the two halves into one solid body. (Note: We used this approach because this part is molded with equal draft in both directions from a middle symmetry plane. Because the cores are drafted in the opposite direction from the outer surface, we had to use the cut operation to create those features. This is an easy way to create parts like this.)



Switch back to the Isometric View. (View 7.)





Your part should look like this:

You can use the CONSTANT RADIUS BLEND Tool to add additional cosmetic blends on the two, outer edges of the part.



