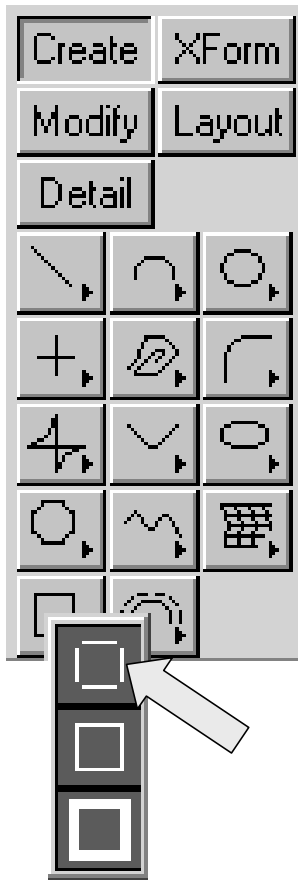


Chapter One

Creating a Simple Plastic Shroud

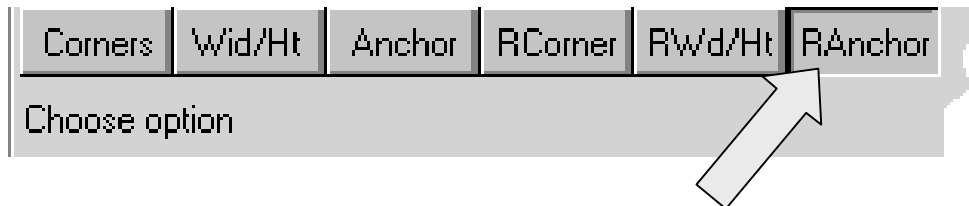
Let's create the simple plastic shroud illustrated to the right. We'll use several new CADKEY tools in the process of creating the part.



Let's start with a new file in View 1.

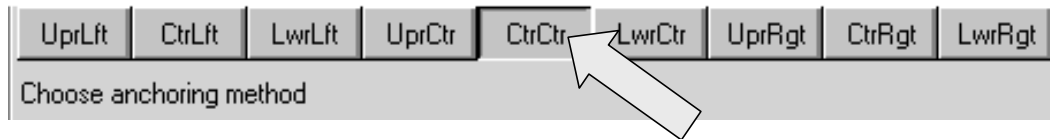
Click on the CREATE RECTANGLE LINES Icon in the CADKEY Application Menubar. Notice that the Conversation Bar now has a selection of options instead of only the CORNERS and WID/HT Options of previous releases!

Your Conversation Bar should loOK like the illustration below. Click on the RANCHOR Option.



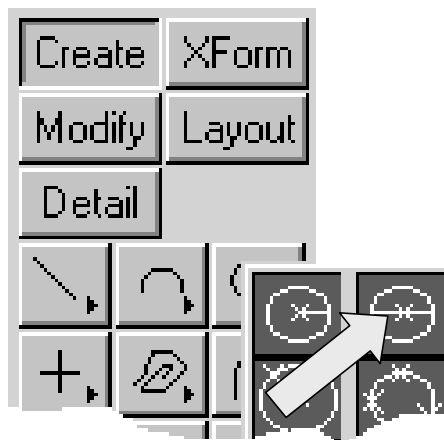
You are presented with a wide range of anchor options. (Traditionally, when you placed a rectangle, you were confined to using the lower left corner of the rectangle. In CADKEY19, you can select any of the four corners, any of the four side midpoints, or the body center of the rectangle.)

We are going to select the body center, which is designated as CTRCTR.



You are now prompted to enter a corner radius. Enter 0.5
You are next prompted to enter a width. Enter 3.
Finally, you are prompted for a height. Enter 3.

Now, use the KEYIN Option and enter 0 for X, 0 for Y, and 0 for Z.



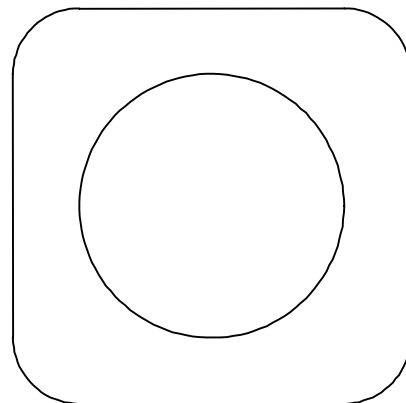
Now, click on the CREATE CIRCLE DIAMETER Icon and enter 2 for the Diameter.

Use the KEYIN Option and once again enter 0 for X, 0 for Y, and 0 for Z.

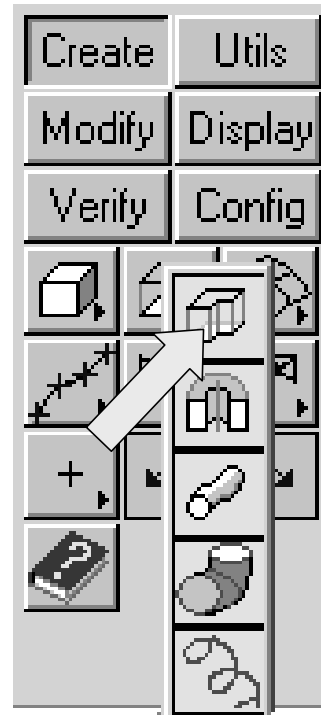
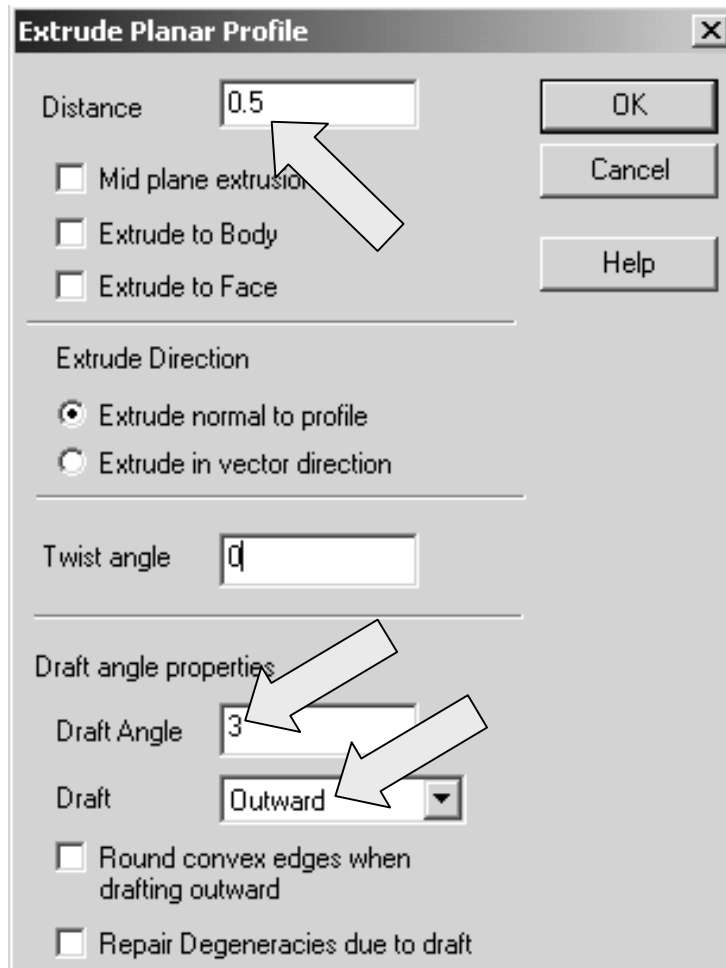
This will place a circle at the body center of the rectangle. You can use the Hot Key ALT + A to Autoscale the display.

(NOTE: In this book and all of the Doctor Walt's books, we always use the CKDOS Keyboard set as our default. The CKWIN Keyboard that automatically installs with CADKEY is less ergonomic, so most serious users quickly gravitate to the more efficient CKDOS alternative.)

Your screen should now look like this:



Next, switch to View 7 and click on the EXTRUDE Icon in the SOLIDS CREATE Menu.



A large Dialog Box appears. Enter 0.5 for the Distance.

Enter 3 for the Draft Angle.

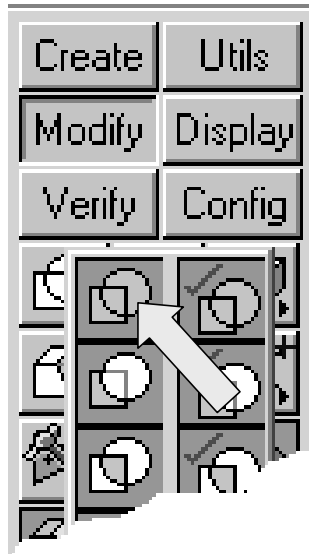
Click on the Outward Draft Option.

Then, click on the OK Button.

Using the CHAIN Option on the Conversation Bar, select the filleted, rectangular profile. After confirming the selection, click on the upward facing Vector to indicate the desired direction of the extrusion.

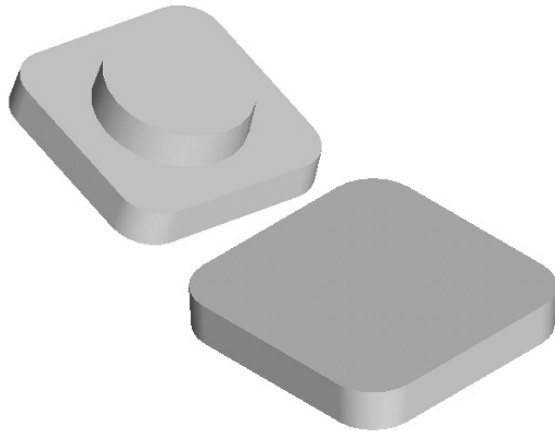
Next, click on the BACKUP Button to re-access the Dialog Box. Change the draft direction option to Inward and click on the OK Button.

This time, using the SINGLE Option, click on the 2 inch circle and then after clicking the ACCEPT Button, click on the downward-facing Vector.



You should now have two separate solids on the screen.

Click on the BOOLEAN UNION Icon in the SOLIDS MODIFY Menu and select first one solid, then the second one to unite them into one body.



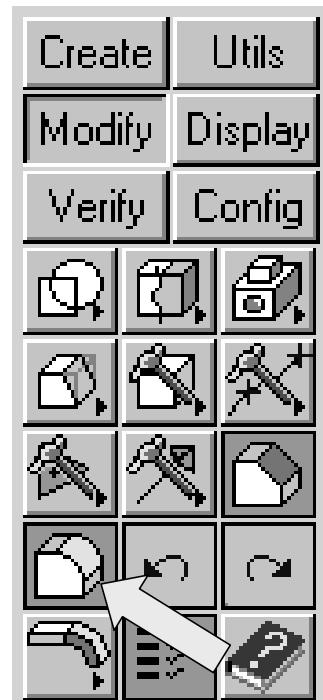
Your part should now look like this:
(I've shown it in two orientations so you can see the conical portion located on the lower end of the part.)

Next, let's click on the BLEND Icon in the SOLIDS MODIFY Menu.

Using the Constant Radius Option, enter 0.25 for the Radius. Make sure that the Blend Along Smooth Edges Option is checked.

Now, click on the circle that defines the intersection of the cone and the rectangular portion of the part. Notice that a fillet propagates around this intersection.

Also click on the outer edge of the part on the same surface as the circle that you previously selected. This will propagate a blend completely around the perimeter. (This is why we needed the blend around smooth edges checked.)





Click on the Local Operations Editing Menubar Icon in the Solids Modify Menu.

Then, click on the SHELL Icon.

A Dialog Box appears. Use the Shell Inward Option and Select Faces to be Opened Option.

Enter 0.05 for the wall thickness.

Click on the OK Button.

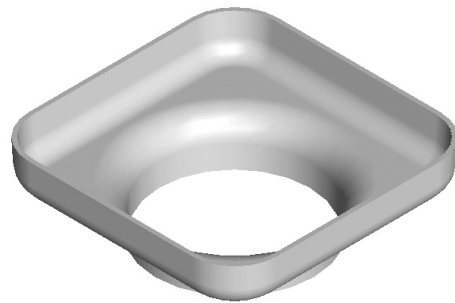
Now, move the cursor over the top surface until it highlights. Depress the CTRL Key while you click on the highlighted surface.

Now, move the cursor over the lower, circular face of the part. (This is presently hidden from the camera view so the face will not immediately highlight.)

Release the CTRL Key, and touch the ENTER key once while you keep the cursor in the center of the circle that defines the lower face. This indexes the face selector to the lower, circular surface. Notice that the circular surface highlights.

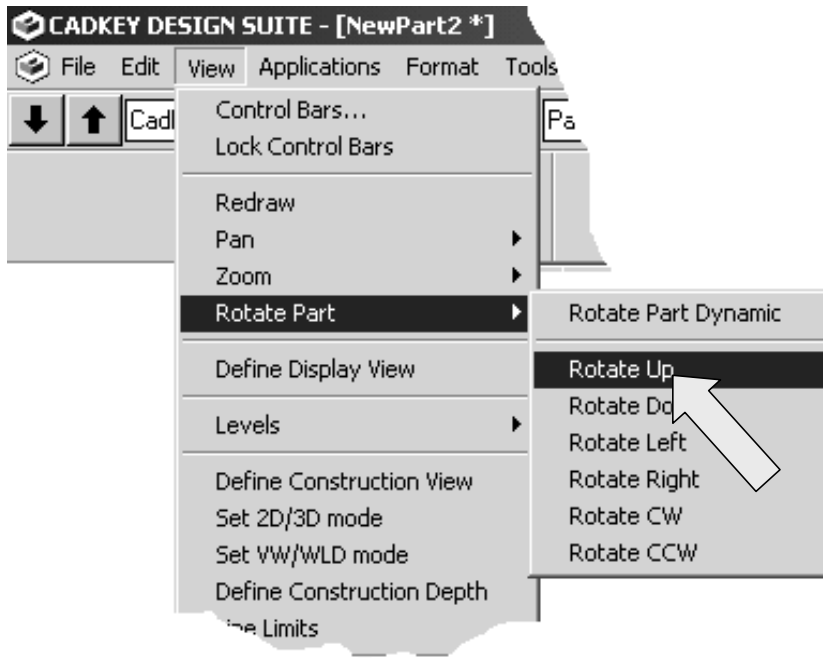
Depress the CTRL Key and click on the highlighted, lower surface. Then, click on the DONE Button on the Conversation Bar.

You should now have a shelled part that looks like this:

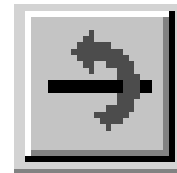


That completes the construction of our simple part. We took advantage of some of the new options in the Create Rectangle Function to simplify the construction. Now, let's look at several, exciting new tools in CADKEY that we can use when manipulating a part such as this.

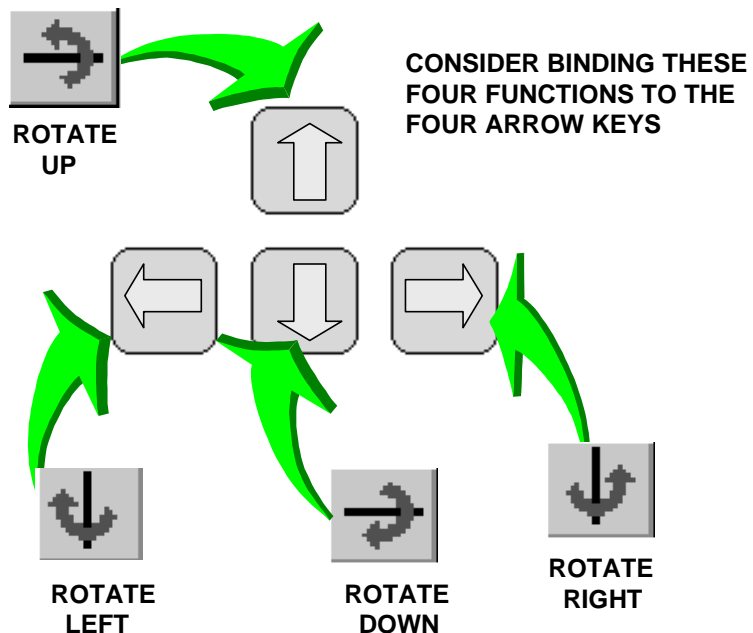
First, click on the VIEW Pulldown Menu. Next, click on the ROTATE PART Option. Then, click on the ROTATE UP Option. Notice that the part rotates up a few degrees.



You'll also see a ROTATE UP Icon appear on the history line. It looks like this:



Try experimenting with the ROTATE DOWN, ROTATE LEFT, and ROTATE RIGHT Options. If you do a large amount of solid modeling, you'll find these quick nudge tools extremely useful. You can drag the Icons up to an empty space on one of your toolbars.



Better yet, why not consider binding them to the UP, DOWN, LEFT, and RIGHT Arrows on your keyboard?

(NOTE: If you are a novice and don't know how to assign new Hot Keys, I recommend my Doctor Walt's CADKEY Starter Kit.)