# Boolean Solids Tutorial

## Introduction

The CADMAX Solid Master Tutorial is a great way to learn about the benefits of constructive (boolean) solid modeling with CADMAX. We have assembled four typical parts that you will be able to construct using CADMAX Solid Master. Each part should take between 20-40 minutes to complete.

The tutorial assumes that you already have some familiarity with CADMAX. In you are unfamiliar with CADMAX, review the *Getting Started* section first. Here are several additional suggestions before getting started:

**!** In the tutorial there are several conventions that are used with text. The text **Draw** > **Circle** > **Center Radius** means to click on the Draw menu, hold the cursor over the Circle function until the Circle sub-menu pops, then click on Center Radius.

**!** When you see a statement that starts with the word "set", this means you are to set a modifier value in the currently displayed Function box. For example, in the Line function box, you will see a -Connect? modifier. When you see the statement, "Set -Connect? to off", this means you should set it off by clicking on the word Connect if it isn't already off.

**!** Sometimes you will be asked to set a specific value for a modifier. For example, in the Circle Function box you will see a modifier called -Radius? with a combo box below it and a history arrow. When you see the statement, "Set -Radius? to .25", this means you should click in the -Radius? text field and type in the value or use the history arrow to select previous value.

**!** When an instruction includes an arrow key ( → ← ↑ ↓ ) followed by a number, hold down the Ctrl key before pressing the specified arrow. For example, <Ctrl+→>1 means hold down the Ctrl key and press →, then release the keys and press 1. After you type in the value, press **Enter**.

**!** Context sensitive prompts are available for each function in the message area at the bottom of the screen. Help is available buy pressing F1.

**!** In the tutorial, we recommend when you should zoom in or out on the drawing to get better views of your work. In addition to our recommendations, you should feel free to use the View menu to Zoom around the drawing as you feel is appropriate. Press F10 to go to previous zooms or F11 to make a new zoom or use the View shortcut icons on the right side of the display.

**!** To load a specified drawing, press F4 to Open a drawing. The drawings that you will be asked to open are located in the \SAMPLES sub-folder under the folder that you installed CADMAX.

# Connector Screw

In this section, you will use CADMAX Solid Master to build the screw for the 27-pin DIN connector that you constructed in the previous section.  By now you have started to become familiar with CADMAX's operation for geometry selection using **Select** and right clicking to select and modify geometry.  The following instructions will have more brevity.  For example, instead of saying "Click **Select** > **Select Element**, then select the line", the instructions will read "Select the line".  You have already learned that there are several ways you can select the line, and it will be your option to choose the best method.

 Your final part should look similar to this:
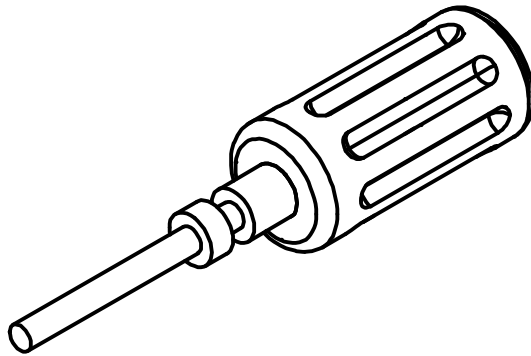
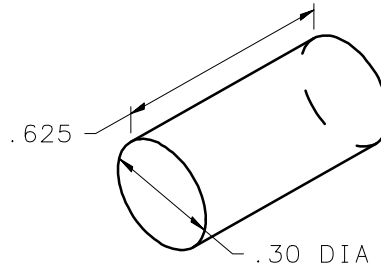## Draft connector screw body components

1. Open the drawing CONNECT, and click on
   **Edit > Surfaces and boolean solids** or click on
   the 3D icon on the Ribbon bar.

2. Click **CPL > Align to View**, and click on the
   front view.

3. Click **Draw > Circle > Center Radius**, set
   -Radius? to '.15', set -Circle Plane? to
   'CPL/Pnts', set -Attribute Name? to 'visible'
   and insert the circle in the top view.

4. Select the circle.
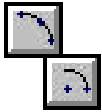
5. Click **Modify > Sweep > Linear Surfaces**, set
   -Sweep? to 'end only', and set -Make Solids?
   'on'. Click anywhere as a pickup point and type
   < Ctrl+ PageDown> .625 as the put down point.
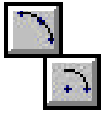
6. In the top view, to the left of the cylinder,
   Click **Draw > Arc > Center Start Angle**,
   set -Angle? to '-90', and set -Arc Plane? to
   'view'. Click anywhere as the center point of
   the top arc of the profile in the illustration and
   type < Ctrl+ ←> .03125.
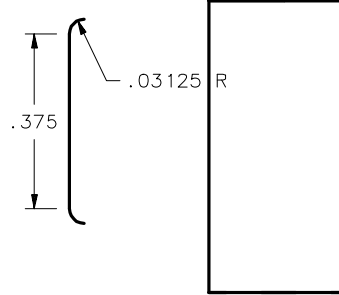
7. Click **Draw > Line**, click on the end point of
   the arc and type < Ctrl+ PageUp> .375.

8. Click **Draw > Arc > Center Start Angle**,
   set -Angle? to '90', type < Ctrl+ →> .03125
   as the center point of the arc and click on the
   endpoint of the line as the arc start point.

9. Select the profile just drafted.

10. Click **Modify > Sweep > Rotate Surface**,
    set -Continuous? 'off', -# Steps? to '3', -
    Angle? to '360', -With? to 'arcs' and -Make
    Solids? to 'yes'. Click on the open endpoint of
    one arc as the first rotate about point, and click
    on the open endpoint of the opposite arc to
    define the rotate about line and form a solid.
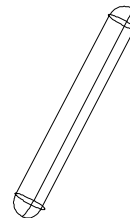
11. Click **Erase** to remove the original profile.

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## Assemble connector screw body

1. Zoom in on the isometric view.

2. Click **Draw** > **More Lines** > **1 Pt Construction**, set -System? to 'model', set -Construction Size? to 'fill view', and set -Angle? to '0'. Click on the **Center point** snap icon in the Quick Snap tool bar and click on the circle of the main body as shown in the illustration.

Snap to circle centerpoint
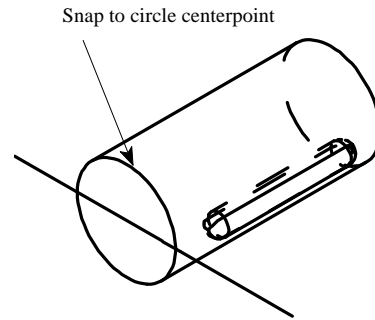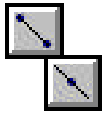
3. Select the small solid object to the side using **Select** > **Select** > **Solid**.

4. Click **Modify** > **Move or Copy**, click on the small object at its nearest tip as the pick up point and put it down at the intersection of the circle and line on the main body, then type < Ctrl+ PageDown> .10 to slide the part along the main object as shown in the illustration.

5. Click **Modify** > **Sweep** > **Rotate**, set -Sweep? to 'none', set -# Steps? to '8', and set -Angle? to '360'. Click on the **Center point** snap icon and click on one of the circles of the main body as the first point of the rotate about line, click on the **Center point** snap icon and click on the opposite circle of the main body as the second point of the rotate about line to complete the circular array.
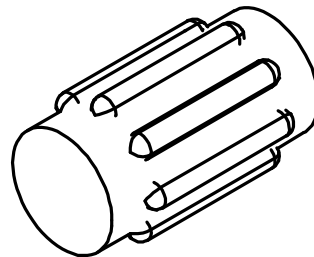
6. Click **Erase** to remove the original object.

7. Press Ctrl+ A to select all editable levels, then click **Select** > **Unselect** > **Solid** and click on the main body.

8. Click **Draw** > **Solid** > **Subtract**, and click on the main body as the solid to change.
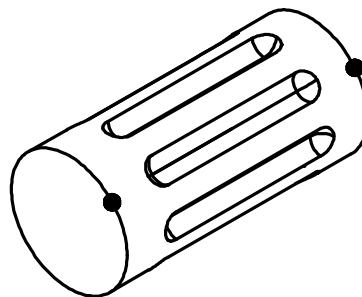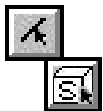
9. Click **Modify > Corner > Fillet/Chamfer Advanced**, set -Radius? to '.03125', set -Type? to 'Fixed radius', click **Add**, then click on the two circles indicated in the illustration, then click **Done**.
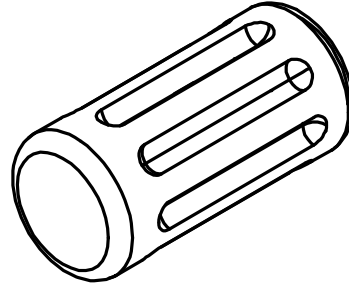
10. Right click, click **Erase,** then click on the construction line to erase it.
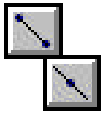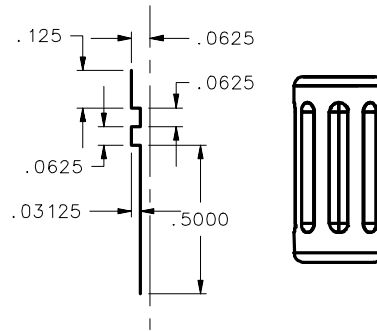
## Draft connector screw

1. Zoom in on the top view.

2. Click **Draw > Line**, click anywhere in the top view to start the lower point of the profile shown in the illustration. Then at the keyboard enter:
   < Ctrl+ PageDown> .5
   < Ctrl+ ←> .03125
   < Ctrl+ PageDown> .0625
   < Ctrl+ →> .03125
   < Ctrl+ PageDown> .0625
   < Ctrl+ ←> .03125
   < Ctrl+ PageDown> .125

3. Click **Draw > More Lines > 1 Pt Construction**, set -System? to 'view', set -Construction Size? to 'fill view', and set -Angle? to '90'. Type < Ctrl+ →> .0625 to place the construction line .0625 to the right of the profile.
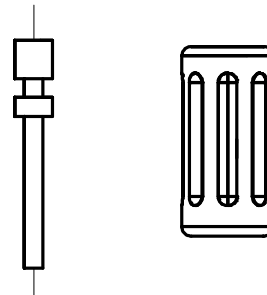
4. Select the profile but not the construction line.

5. Click **Modify > Sweep > Rotate Surfaces**, set -Sweep? to 'ends only', set -Continuous? 'on', -# Steps? to '1', -Angle? to '360', and -With? to arcs. Click on any two points on the construction line as the rotate about line to form a solid of the bottom of the screw connector.

6. Click **Erase** to remove the original profile, then click on the construction line to erase it.

1. Zoom in on the isometric view.

2. Click **Select** > **Select** > **Solid** and then click on the connector screw.

3. Right click, click **Move**, click on the **Center point** snap icon and click on the circle at the top of the screw connector as the pick up point. Click on the **Center point** snap icon and click on circle at the bottom of the screw body as the put down point.

4. Click **Draw** > **Solid** > **Add** and click on the main body.

5. Save the drawing as CNTSCREW unless you are running Test Drive in which case you can make a temporary snapshot of your working by clicking **File** > **Save Snapshot**.