Assembly-in-Place Tutorial

Introduction

The CADMAX Solid Master Tutorial is a great way to learn about the benefits of feature-based parametric solid modeling with CADMAX. We have assembled several 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. If you are unfamiliar with CADMAX, review the Getting Started section first in CADMAX Help or in the CADMAX Users Manual. 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.

! Most functions are available through drop-down menus and icon tool bars. Throughout the tutorials the written instructions tell you how to do things from the drop-down menus and the icons to the left show you how to accomplish the same operation using icons. The first icon is the icon to click on the main tool bar, and the second indented icon is the icon on the vertical function bar.

    < Locate this icon on the top tool bar and click on it.

    < Then click on this icon on the vertical tool bar that pops up on the left side.

! 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", 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 (6 7 8 9) followed by a number, hold down the Ctrl key before pressing the specified arrow. For example, <Ctrl+6>1 means hold down the Ctrl key and press 6, 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 or the Quick View tool bar to zoom around the drawing as you feel is appropriate. Press F10 to go to previous zooms or F11 to make a new zoom.

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

Assembly-in-Place Modeling

In this tutorial you will create a basic fixture using Assembly-in-Place modeling. Assembly-in-Place lets you model multiple parts simultaneously in a single drawing, which gives you the ability to dynamically create relationships between these parts as they are created. Then as revisions are applied to the design, associated parts will change automatically without the need for defining explicit variables.

You will also learn how to place parts into an assembly. Additional parts can be placed from parts created in the current drawing or from other drawings using parts libraries.
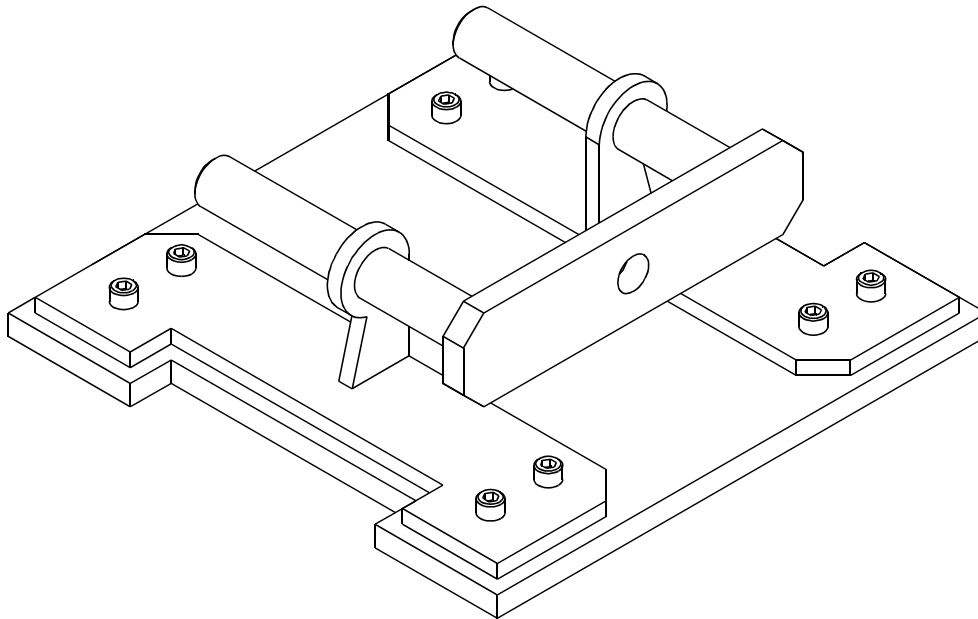
Construct a Mounting Bracket

In this section you are going to start with a initial part called a *mounting base* and construct a bracket directly on the *mounting base*. You will start by making a sketch of the bracket on the top face of the base part.
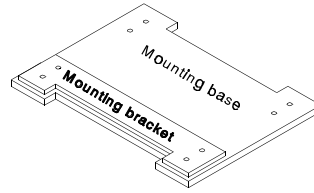
1. Open the drawing *mountingbase.cxd* from the \Samples\Fixture folder which is a subfolder under the folder where you installed CADMAX.

2. Click on **Edit > Feature Modeling** or click on the Feature Modeling icon on the Ribbon bar to make sure you are in Modeling mode.

In the next series of features you will create a *mounting bracket* which will be based on the geometry in the *mounting base*. By using CADMAX's assembly-in-place technology you can easily establish relationships between the parts. If one part changes the other parts that relate will automatically change with it.
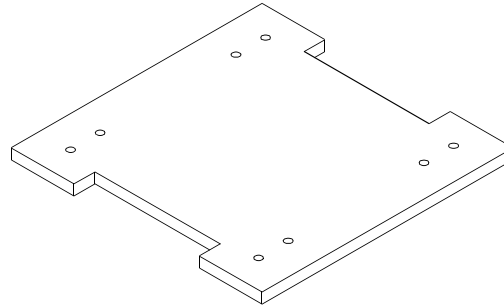
3. Click on **Edit > Sketch** or click on the Sketch icon on the ribbon bar. This starts a new sketch and displays the Work Plane function box.

    The Work Plane function box shows different methods to locate the sketch plane, depth, rotation and twist.

    Once the work plane is defined, the **Sketch** function bar will open and place you in the **Sketch Line** function. The Sketch function bar contains many of the functions you will use to draw and dimension a sketch.
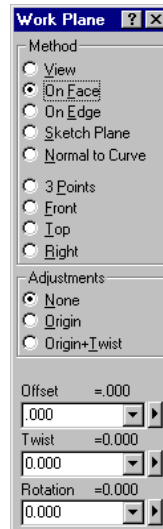
4. Set the -Method? to 'On Face',
-Adjustment? to 'None', and click on the top
face of the mounting base in the top or
isometric view. Now, no matter which view
you work in, as you sketch your object, the
geometry will always be oriented to the top
view on the top face of the mounting base.
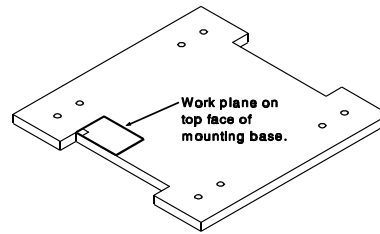
Note: You will find it easier if you zoom in on
the top view to for the next several steps.
Use View > Zoom In or click on the Zoom In
icon on the vertical tool bar along the right
edge of the display and then click on two
point to define the area to zoom in on.

5. Click on **Draw > Line.** Working in the top
view, sketch an object as shown.

Notice that CADMAX displays dynamic
construction lines as guides to help you
make vertical and horizontal lines.

In particular, when you draw the second
horizontal line at the bottom of the sketch,
make sure you use the dynamic construction
line generated by the first horizontal line. If
the dynamic construction line doesn't
appear, wave over the first horizontal line
until the dynamic construction line appears.
Using this procedure, it will only be
necessary to vertically dimension one line,
since both lines are implicitly linked.
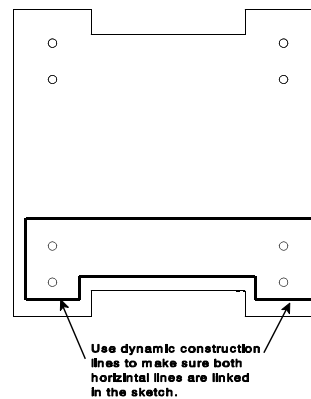
6. Click on **Draw > Circle > Center, Point.** Set
-Chain? 'on'.

7. Starting with one of the four holes in the
mounting base, wave over the circle until
the center point pops up and click on the
center point. Then drag the cursor until the
echo of the circle snaps to the diameter of
the circle in the *mounting base* and click to
place the circle.
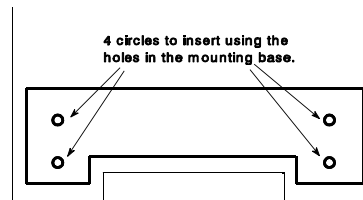
This forces the diameter of the new circle to
always be equal to the diameter of the hole
in the *mounting base*. If the diameter is
changed in the *mounting base,* then the
diameter of the hole in the *mounting bracket*
will automatically change too.

8. Since -Chain? was set 'on', all ensuing circles will have an equal diameter to the first circle. Place three more circles by waving over each succeeding circle until the center point pops up and then click on the center point to place each circle.

Adding Dimensions

Dimensions are used to make geometry the desired size. CADMAX does not require that you add dimensions before you extrude the model. You can always return to the sketch and add dimensions or change the sketch later in the design. However, in this case, the size of the mounting bracket is defined relative to the mounting base. All edges are .25" in from the edge of the mounting base and the overall height of the mounting bracket is 2.50".

1. Click on **Dimension > Linear.**

2. Click on the points A and B to define the dimension shown. Drag the cursor to the desired text position and click to place the dimension.

   Note: It may be necessary to set -Arrow Position? to 'inside' or 'outside' to make the dimension appear like the figures.

   As you move the cursor to the right or left, CADMAX calculates a vertical dimension or as you move up and down a horizontal distance is calculated. Use the Ctrl key to hold a particular dimension type when attempting to place the text.
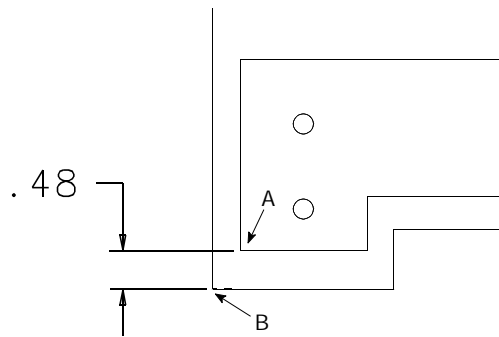
3. Type in .25 and press **Enter**. This assigns a value of .25" for the dimension and automatically resizes the geometry according to the assigned value.

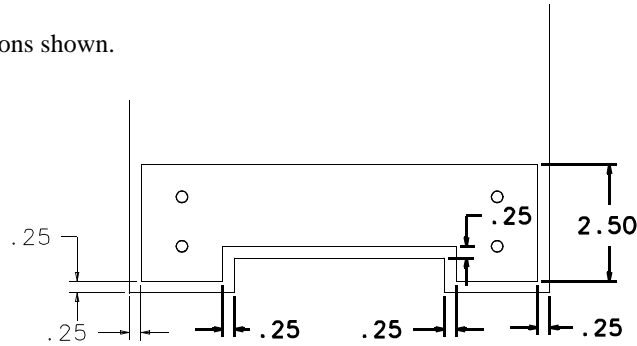   You can always change this value at a later time using the **Equation** function.

   If the sketch geometry was constrained correctly, both bottom horizontal lines should have moved when the dimension was changed.
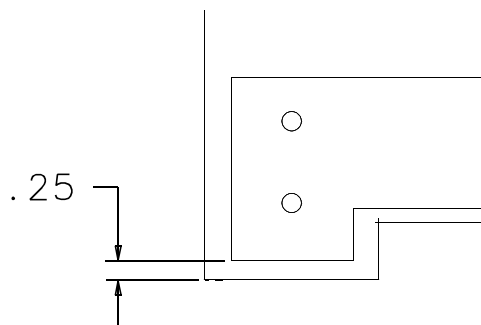
4. Repeat the above steps to place the
    horizontal dimension.
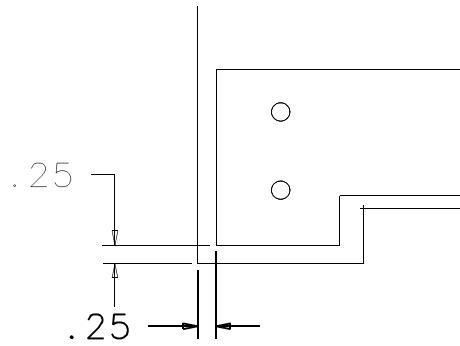
5. Add the remaining dimensions shown.

Note: Once the dimensions are added, CADMAX reported in the Status Area at the bottom of
the display that the sketch was *well-defined* with the correct number of dimensions. This
means that you have provided the correct dimensions and constraints to fully control this
sketch. Many of the constraints that are part of this sketch were invisible to you. The
lines were constrained to be parallel and the angles were constrained to be perpendicular.
These constraints were added as a function of using the dynamic construction lines.
Without those constraints, it would have been necessary to add angular dimensions to
make the sketch *well-defined*.

CADMAX does not require that each sketch be *well-defined*. You can add only the
dimensions and constraints needed and continue modeling. You can set your **Sketch
Preferences** to display the status of sketch geometry. See Sketching in the *CADMAX
Solid Master Help* for more information on colors and display status.

Extrude the Mounting Bracket

The mounting bracket is a separate part from the mounting base. When the sketch is extruded, use Base to create a new solid. Before the extrusion, you can assign a new part name and a different geometry attribute to the part.
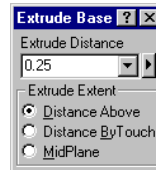
1. Click on **Feature > Extrude Sketch** or click on the **Extrude Sketch** icon on the top tool bar.

    The Extrude dialog box appears so you can specify the -Boolean Choices? to use. Click on 'Base'.
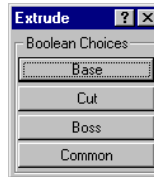
2. Set -Extrude Distance? to '.25' and -Extrude Extent? to 'Distance Above'.

3. On the Ribbon bar set the -Part Name? to 'Mounting bracket' and the -Geometry Attribute? to 'visible'.

    Hint: The Ribbon bar is the second horizontal tool bar. Type in the Part Name and use the drop down arrow to choose a geometry attribute.

    A Geometry Attribute is used to control the color, line style, and pen number for the plotter. The Geometry Attribute 'visible' makes the geometry orange.

4. The direction of the extrusion is defined by the side of the sketch that you click on. Working in the front, side or isometric view, click above the object, to create the initial feature of the mounting bracket.

    Note that the feature has been named "mounting bracket" in the feature tree and the geometry color should be orange.
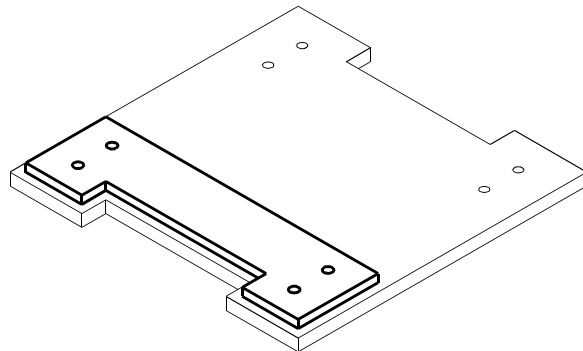
Test Part Associativity

The next series of steps shows how the parts relate to each other. As changes are made to the variables used to define the *mounting base*, the *mounting bracket* will change with it, keeping the relationships that were established between the parts. This is the power behind CADMAX's assembly-in-place construction. As a function of the design process, geometry between parts relate automatically for relationships that don't have to be explicitly defined.

1. In the Feature tree double-click the *Mounting base* feature to display the Variables dialog box or right click the *Mounting Base* feature to display the pop up menu and click **Edit Variables**.
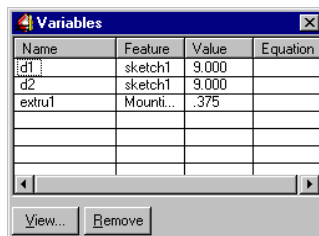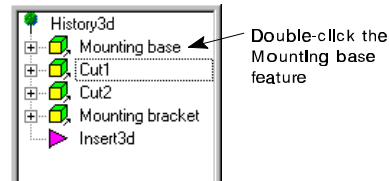
2. The Variables dialog box shows the variables that were used to define the initial mounting base feature. d1 and d2 specify the length and width. Change the value of d1 to 12, by clicking on the 9.000 and then enter the new value. It is not necessary to press **Enter** or type in the decimal or trailing zeros.

3. Click **Feature > Rebuild Model**.

    Both parts rebuild based on the new value for d1. Notice that the mounting bracket maintained its original relationship with the mounting base.
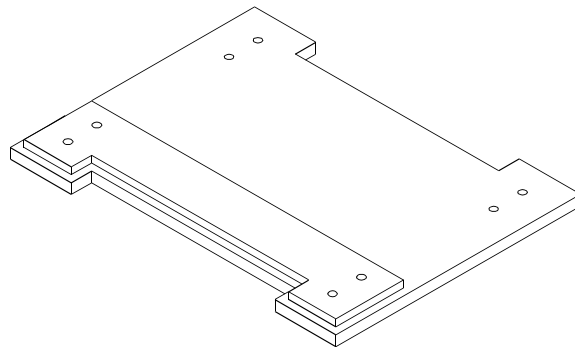
4. Let's make a couple more changes.

    Double-click on *Cut1* and change d5 and d6 to 2. Then double-click on *Cut2* and change diam1 to .75.

5. Click **Feature > Rebuild Model**. Note how the mounting bracket is driven by the changes to the features of the mounting base.

6. Change the mounting base back to its original dimensions. Double-click on *Mounting base* and change d1 to 9. Double-click on *Cut1* and change d5 and d6 to .75. Double-click on *Cut2* and change diam1 to .25.
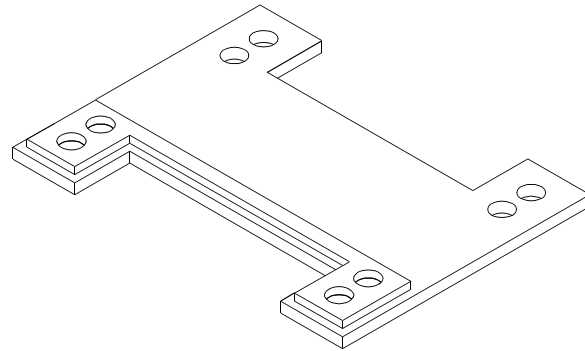
7. Click **Feature > Rebuild**.

8. Right-click on any of the features in the Feature Tree that are bold, and if Show is checked, click Show to uncheck it and make the sketch no longer display.
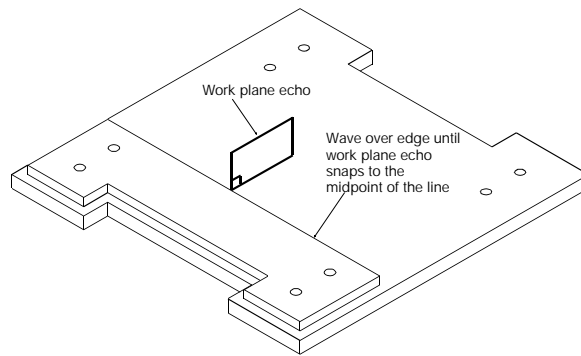
Complete the Mounting Bracket

In the next series of steps you will add the clip and chamfer to the mounting bracket.

1. Click on **Edit > Sketch** or click on the **Sketch** icon on the ribbon bar to start a new sketch.

2. In the Work Plane function box, set the -Method? to 'Right', -Adjustment? to 'None'. Click on the midpoint snap button in the snap tool bar on the right.

3. In the isometric view, wave the cursor over the back edge until the work plane snaps to the midpoint of the line. Click to place the work plane. This sets the work plane to be centered on the object and aligned to the side view.

4. Click on **Draw > Sketch**.

5. Working in the side view, add three lines as shown. Start by waving the cursor over point B, to make the vertical dynamic construction line display. Then click on points A, B, C, and D. Make sure that point C is only along the horizontal line and

Work plane echo

Wave over edge until work plane echo snaps to the midpoint of the line
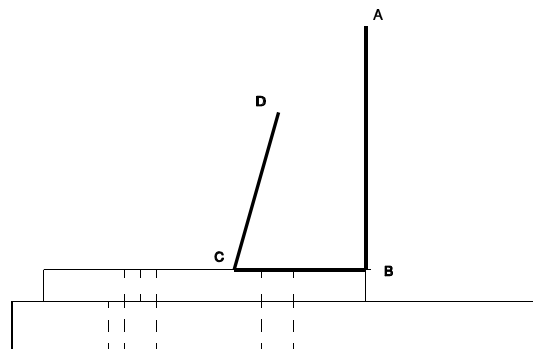
doesn't snap to any pop up end points, otherwise point C will be constrained to that location. Also, make sure point D doesn't snap to any dynamic construction lines.

6. Click on **Draw > Arc >3 Points** . Click on point A as the start of the arc and point D as the end of the arc, then drag the cursor up and click to place the arc.

7. Click on **Draw > Constrain > Tangent**. Click on the arc, then click on the vertical line at point A to make the arc tangent to the line.
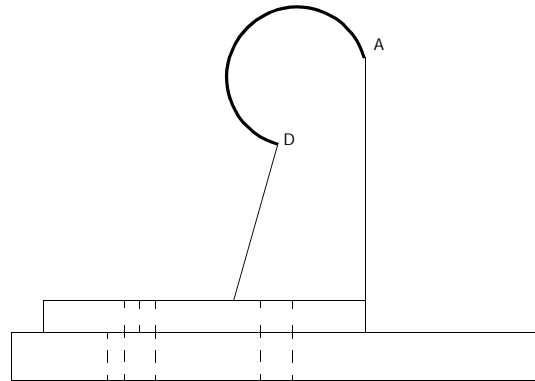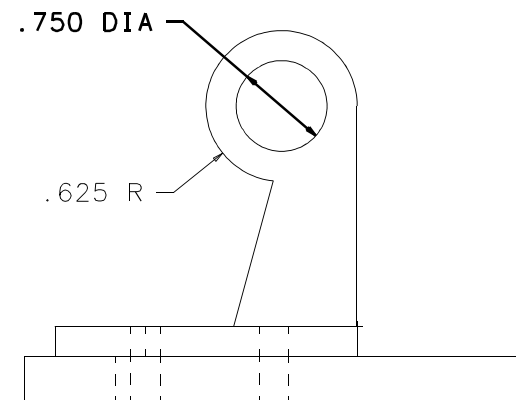
8. Click on **Dimension > Radius**. Set -# Auto Decimal Places? to '3'.

9. Wave the cursor over the arc until it turns red and an echo of the radial dimension is displayed. Click on the arc, drag the cursor to locate the text, and click to place the radial dimension.
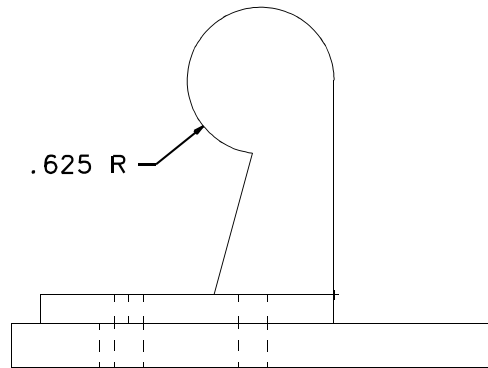
10. Type .625 and press **Enter** to set the radius of the hole to .625.

11. Click on **Draw > Circle > Center, Point.** Wave the cursor over the arc until the center point pops up. Click on the center point and drag the cursor out and click to create a circle.
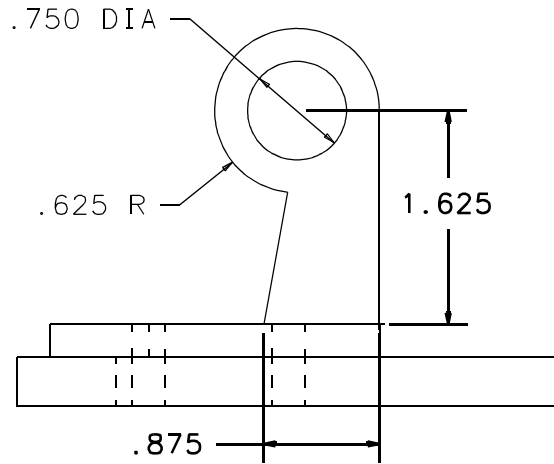
12. Click on **Dimension > Radius**.

13. Wave the cursor over the circle until it turns red and an echo of the radial dimension is displayed. Click on the circle, drag the cursor to locate the text, and click to place the radial dimension.

14. Type .75 and press **Enter** to set the diameter of the hole to .75.

15. Click on **Dimension > Linear.** Add the dimensions shown in the figure to the right. After you place the dimension text, just type in the desired size and press **Enter**.

    As a reminder, you can freeze a dimension to horizontal or linear by holding down the Ctrl key before you place the text.

16. Click on **Dimension > Angle.** Add the angular dimension shown in the figure to the right by clicking on the two lines to measure the angle between. After you place the dimension text, just type in the desired angle and press **Enter**.

    Note: Check the Status Area. CADMAX report that the sketch is well-defined. This means that the correct number of dimensions and constraints have been specified.
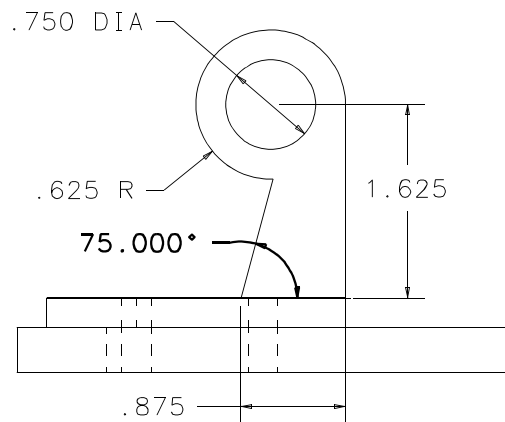
17. Click on **Feature > Extrude Sketch** to display the Extrude function box. Select **Boss** as the type of extrude. Boss means to add the new feature to the specified solids.
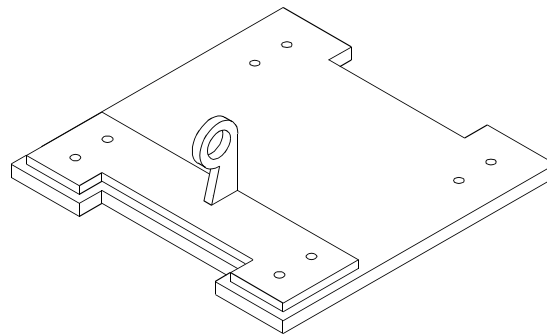
18. In the Extrude Boss function box, set:

   - • -Target Solids? to 'All'
   - • -Extrude Distance? to '25'.
   - • -Extrude Extent? to 'Midplane', which will extrude the profile one-half the -Extrude Distance? in both directions.

19. Click anywhere inside one of the views to extrude the profile.

20. Zoom in on the isometric view and change the view to be hidden-dashed.

21. Click **Select > Select Element** and click on the two short vertical edges shown in the figure to the right.
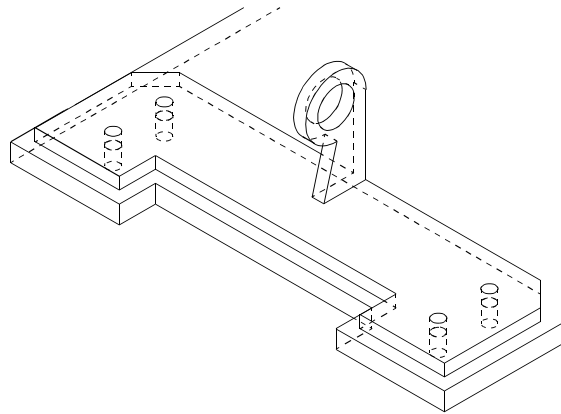
   Notice that the edges and surfaces highlight in red as you move the cursor over them identifying which item will be selected. Make sure only the edge is highlighted when you click and not the entire surface.
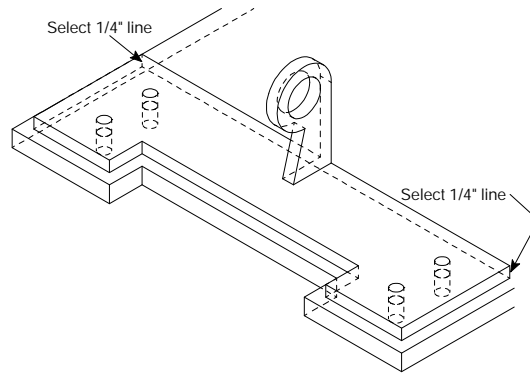
22. Click on **Feature > Chamfer, Fixed.** The Chamfer function box appears.
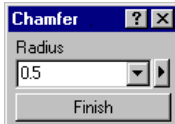
23. Set -Radius? to '.5' and click **Finish** to apply the chamfer feature to the selected edges.
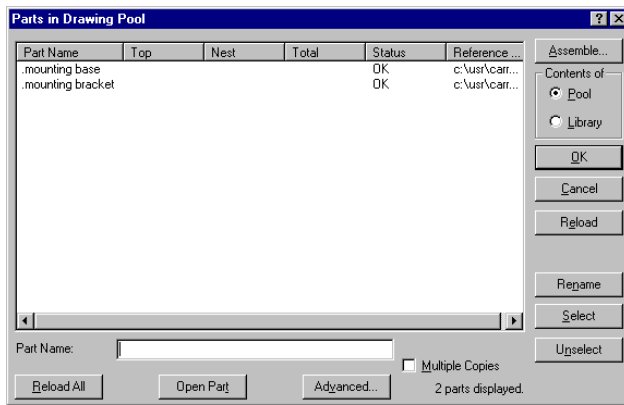

Instance the Mounting Bracket

In the next series of steps you will insert an instance of the mounting bracket on the opposite side of the mounting base.

Throughout the process of inserting parts into the assembly you should make use of the zoom functions to get a better view. Zoom and Previous are the top two icons on the vertical tool bar along the right edge of the display. You will find it easiest t work in the isometric view with a hidden dashed picture.
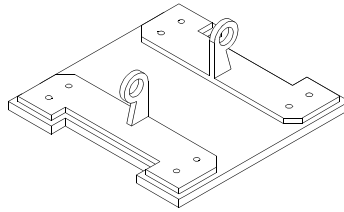
1. Click on **Tools > Parts** or click on the **Parts** icon on the main tool bar.

2. In the Parts dialog box make sure -Contents of? is set to 'Pool'. When -Contents of? is set to 'pool', the Parts dialog show the parts in the current drawing, which in this case are the *mounting base* and *mounting bracket*.

3. Click on *mounting bracket* and then click on **Assemble** to begin the process of placing another instance of the *mounting bracket* into the drawing.

4. An instance of the *mounting bracket* is attached to the cursor and the Initial Place dialog box is displayed. Drag the *mounting bracket* into the isometric view and put it down towards the rear of the *mounting base*. This is just an initial placement. In the following steps you will use additional functions to position the *mounting bracket* precisely on the *mounting base*. When you place the instance, it will turn violet.

Hint: It will be easier to work if you zoom into the isometric view and set it to be hidden-dashed.
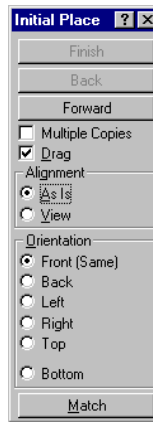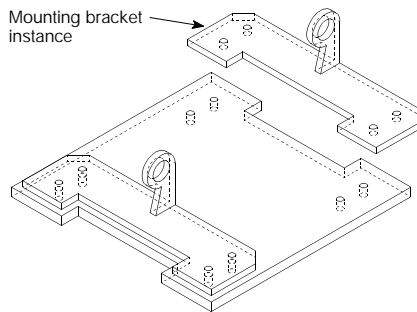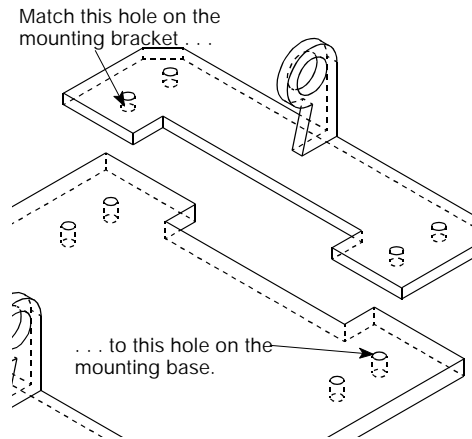
Mounting bracket instance →

5. Once initial placement has been completed, the Orient From dialog box is displayed. Use Orient From to define the work plane and origin of the current instance that will be used to position it on the assembly. The plane that is selected here will be matched to a plane on the assembly in the next step.

Set -Method? to 'On Edge'. As you wave over the instance of the *mounting bracket*, the work plane echo shows the origin and the orientation of the work plane.

When using 'On Edge', the work plane is oriented to a specific face which is defined by the edges of the face. If two planes are shared by an edge, the plane that is defined is determined by the side of the line that the cursor is positioned.
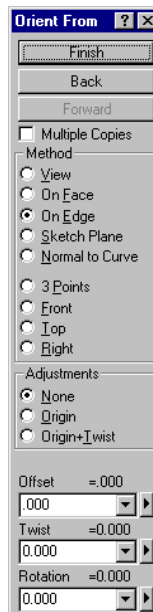
6. In order to position the mounting bracket, it will have to be rotated 180 degrees from its current position. You will accomplish this by anchoring matching holes between the *mounting base* and the *mounting bracket*, and then pivot the *mounting bracket* into place to a second set of matching holes.

Match this hole on the mounting bracket . . .
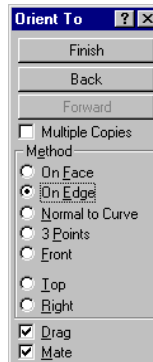
. . . to this hole on the mounting base.

Wave the cursor over the *mounting bracket* instance until the work plane echo snaps to the matching hole on the bottom of the *mounting bracket* and then click to set the work plane to this location.
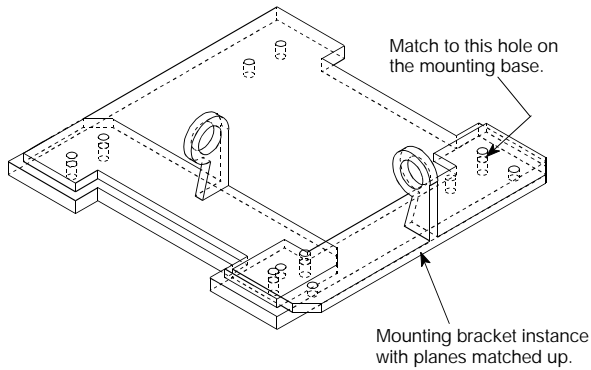
Work plane echo on the matching hole on the bottom face.
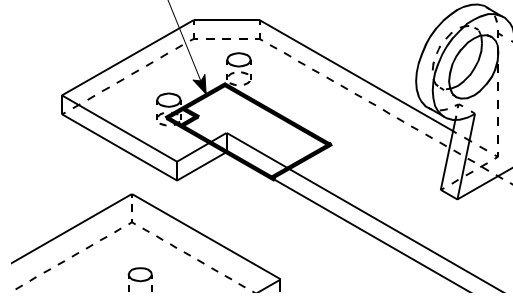
7. As soon as the Orient From work plane is defined, the Orient To dialog box is displayed. Use Orient To to identify the plane that will match to the Orient From plane. Only planes on the assembly can be used with Orient To.
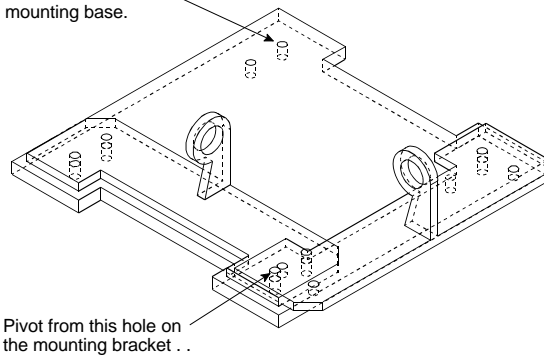
Wave the cursor over the *mounting base* until the work plane echo snaps to the matching hole on the top of the *mounting base* and then click to position the *mounting bracket* to this location.
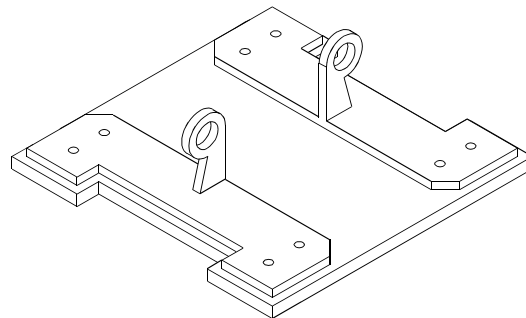
Orient To
- Finish
- Back
- Forward
- ☐ Multiple Copies
- Method
  - ○ On Face
  - ● On Edge
  - ○ Normal to Curve
  - ○ 3 Points
  - ○ Front
  - ○ Top
  - ○ Right
- ☑ Drag
- ☑ Mate

Match to this hole on
the mounting base.

Mounting bracket instance
with planes matched up.

8. Once the work planes are matched, you are
ready to pivot the *mounting bracket* into
place. Parts are pivoted about their origin.
Click on **Pivot** in the Align From dialog box.
Then click on the hole in the *mounting
bracket* that will pivot to the *mounting base*,
then click on the hole in the *mounting base*
to pivot to.

. . . to this hole on the
mounting base.

Pivot from this hole on
the mounting bracket . .

Align From

Finish
Back
Forward

Align
Pivot

Move X
Move Y
Move Z

X Offset   =.000
.000
Y Offset   =.000
.000
Z Offset   =.000
.000
Angle   =0.000
0.000

☐ Reverse
☐ Multiple Copies
☑ Drag

9. Click **Finish** to complete placement of the
*mounting bracket* instance.

10. Save your work.

Adding Parts from Other Drawings

In next section you will add an *insert* between the two mounting brackets. The *insert* is a part that comes from a separate drawing. Parts are registered with a Part Libraries.
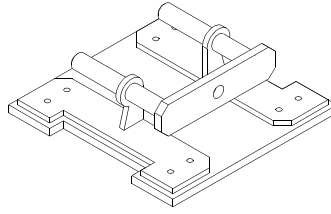
1. Click on **Tools > Parts** or click on the **Parts** icon on the main tool bar.

2. In the Parts dialog box set -Contents of? to 'Library'. The Parts dialog box expands to show the Part libraries. Locate 'assy2' from the list of libraries and click on the library name.
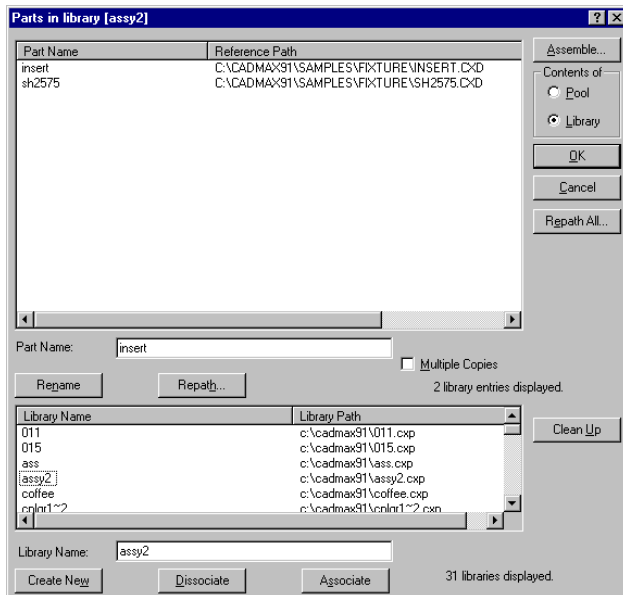
When -Contents of? is set to 'library', the Parts dialog box expands to show a list of part libraries. In this case the parts in the assy2 library are *insert* and *sh2575*.

3. Click on *insert* and then click on **Assemble..** to begin the process of placing an instance of the *insert* into the drawing.

4. An instance of the *insert* is attached to the cursor and the Initial Place dialog box is displayed. Drag the part into the work area and click to put it down. This is just an initial placement. In the following steps you will use additional functions to position the *insert* precisely on the fixture assembly. The violet color is used to denote that this is the part being added to the assembly.
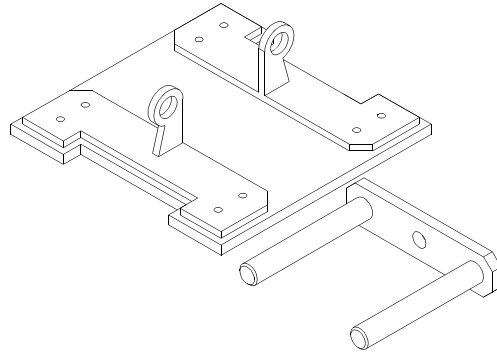
5. Once initial placement has been completed, the Orient From dialog box is displayed. Use Orient From to define the work plane and origin of the part that will be used to position it on the assembly. The plane that is selected here will be matched to a plane on the assembly in the next step.

   Set -Method? to 'On Edge'. As you wave over the instance of the *insert*, the work plane echo shows the origin and the orientation of the work plane to be defined.

   When using 'On Edge', the work plane is oriented to a specific face which is defined by the edges of the face. If two planes are shared by an edge, the plane that is defined is determined by the side of the line over which the cursor is positioned.
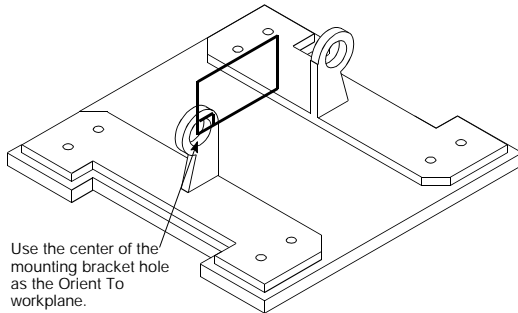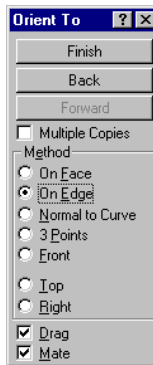
6. On the *insert* the Orient From plane is going to be the plane on the end of the cylinder with the origin as the center of the cylinder.

   Wave the cursor over the *insert* instance until the work plane echo snaps to the center of the cylinder as shown and then click to set the work plane to this location.

   Orient from workplane is the plane on the end of the cylinder.
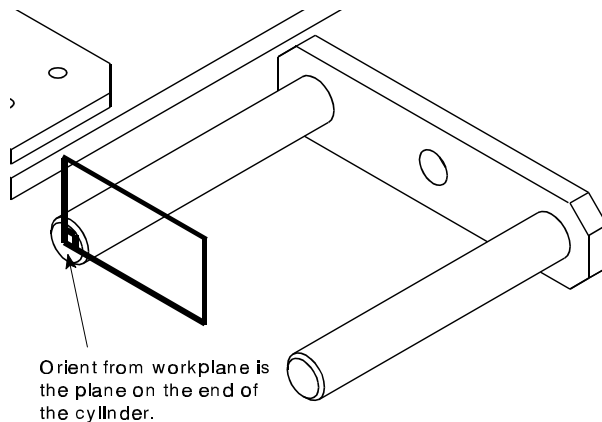
7. As soon as the Orient From work plane is defined, the Orient To dialog box is displayed. Use Orient To to identify the plane on the assembly that will match to the Orient From plane. Only planes on the assembly can be used with Orient To.
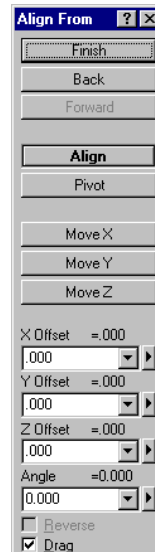
   Wave the cursor over the assembly until the work plane echo snaps to the matching hole on the top of the *mounting bracket* and then click to position the *insert* to this location.

Orient To

Finish
Back
Forward
☐ Multiple Copies

Method
○ On Face
◉ On Edge
○ Normal to Curve
○ 3 Points
○ Front
○ Top
○ Right
☑ Drag
☑ Mate

Use the center of the mounting bracket hole as the Orient To workplane.

Initial placement of the insert using the Orient From and Orient To workplanes.
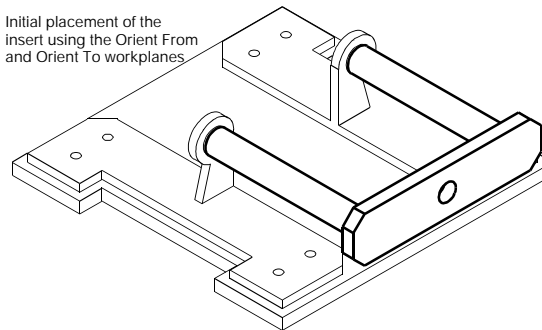
8. Once the initial placement of the *insert* has been completed, you are ready to put the part in its final position. The Align From dialog box contains a number of different operations that can be applied to the part. In this case, you will move the *insert* along one axis to slide the part further into the *mounting bracket* holes using Offset.

Align From

Finish
Back
Forward

Align
Pivot

Move X
Move Y
Move Z

X Offset   =.000
.000
Y Offset   =.000
.000
Z Offset   =.000
.000
Angle   =0.000
0.000
☐ Reverse
☑ Drag

Choosing the offset axis is determined by
the work plane used to position the part.
The work plane locator echo defines the axis.


Work plane locator echo
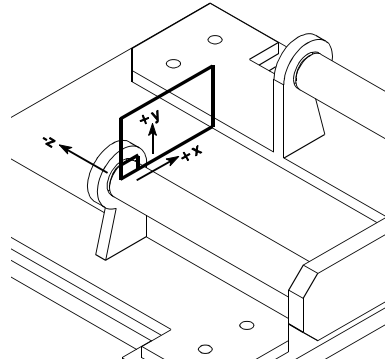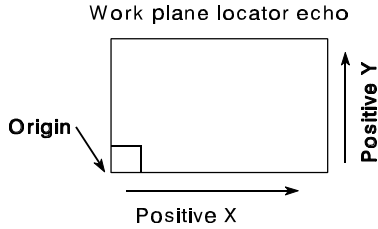
As shown, the *insert* is going to be offset in
the negative z direction by a distance of '3'.

Set -Offset Z? to '-3.0'. A red echo previews
the offset position of the part.

Offset is used to move the part being added
to the assembly a specific distance along
one axis. The **Move** functions are used to
move the part along an axis by choosing a
pick up point on the part and a put down
point on the assembly. The part then move
the distance between the two points in the
specified axis.

9. Click **Finish** to complete placement of the
   insert instance.
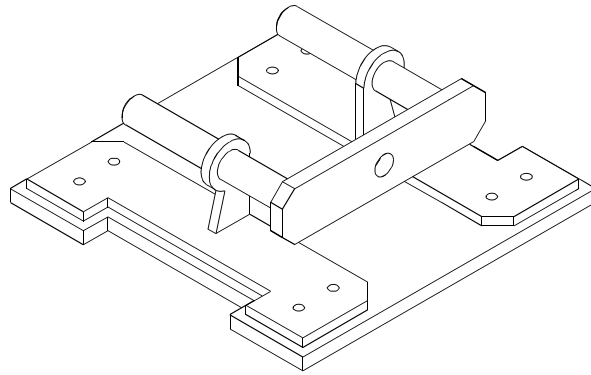
Insert Socket Head Screws

In next section you will complete the fixture by adding socket head screws to secure the *mounting bracket* to the *mounting base*. The socket head screw comes from a separate drawing.
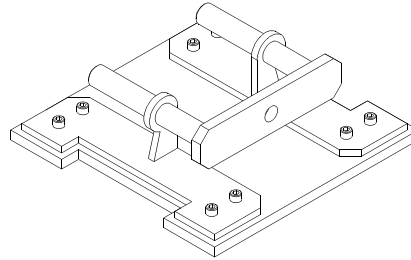
1. Click on **Tools > Parts** or click on the **Parts** icon on the main tool bar.

2. In the Parts dialog box set -Contents of? to 'Library'. The Parts dialog box expands to show the Part libraries. Locate 'assy2' from the list of libraries and click on the library name.

    When -Contents of? is set to 'library', the Parts dialog box expands to show a list of part libraries. In this case the parts in the assy2 library are *insert* and *sh2575*.

3. Click on *sh2575* and then click on **Assemble..** to begin the process of placing an instance of the first instance of *sh2575* into the drawing.

4. An instance of *sh2575* is attached to the cursor and the Initial Place dialog box is displayed. Check -Multiple Copies?, set -Orientation? to 'Top', and click to perform the initial placement of the socket head screw. When you place the instance, it will turn violet.

    Note: The Orientation option helps you to visually relate the part to the assembly, but in no way controls its final position.

    Note: When Multiple Copies is checked, a new instance of the part is automatically inserted when Finish is clicked, using the original Orient From work plane.
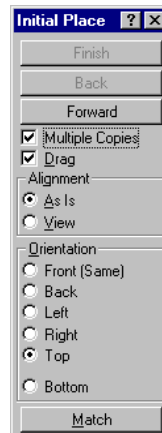
5. Once initial placement has been completed, the Orient From dialog box is displayed. Use Orient From to define the work plane and origin of the part that will be used to position it on the assembly. The plane that is selected here will be matched to a plane on the assembly in the next step.

   Set -Method? to 'On Edge'. As you wave over *sh2575*, the work plane echo shows the origin and the orientation of the work plane to be defined.

   Note: When using 'On Edge', the work plane is oriented to a specific face which is defined by the edges of the face. If two planes are shared by an edge, the plane that is defined is determined by the side of the line that the cursor is positioned.
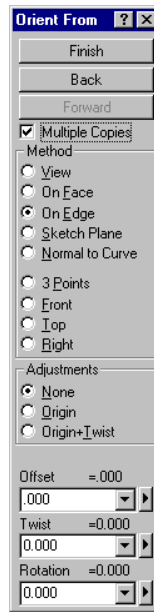

6. The Orient From plane is going to be top of the cylinder where it meets the socket head with the origin as the center of the cylinder.

   Wave the cursor over the sh2575 part until the work plane echo snaps to the center of the cylinder as shown and then click to set the work plane to this location.
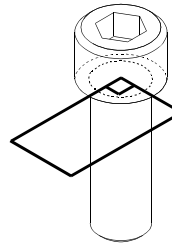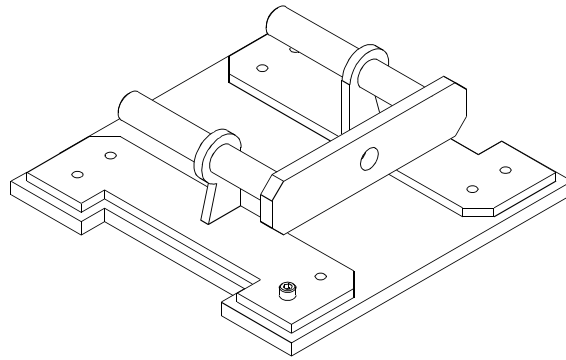

7. The Orient To dialog box is displayed. Use it to identify the plane on the assembly that will match to the Orient From plane. Only planes on the assembly can be used with Orient To.

   Wave the cursor over the assembly until the work plane echo snaps to one of the screw holes on the *mounting bracket* and then click to position the socket head screw to this location. If the part position is mated incorrectly, then click on -Mate? to reverse the mating direction before clicking on the put down point in the assembly.
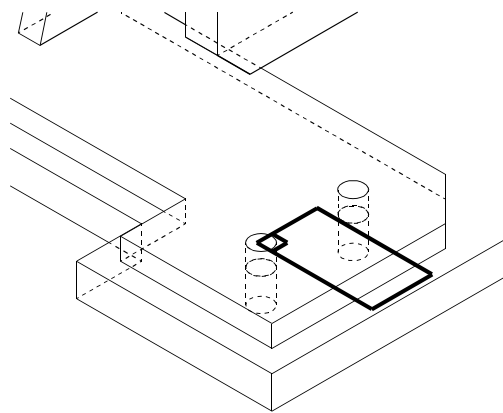
8. No additional steps are needed to position the socket head screw. Click **Finish** in the Align From dialog box to complete insertion of the first instance of the socket head screw.

9. Since -Multiple Copies? was checked, a new instance of the part is inserted into the drawing and the Align To dialog box is displayed. It is not necessary to make an initial placement or to define an Orient From work plane. CADMAX reuses the original Orient From work plane, so now all you have to do is place the part.

   Wave the cursor over the assembly until the work plane echo snaps to one of the screw holes on the *mounting bracket* and then click to position the socket head screw to this location. If the part position is mated incorrectly, then click on -Mate? to reverse the mating direction.

10. Click **Finish** in the Align From dialog box to complete insertion of the next instance of the socket head screw.

11. Continue inserting additional instances of the socket head screw until all eight parts are in place. Before you click **Finish** on the last screw uncheck -Multiple Copies? to stop adding additional copies of the socket head screw. Otherwise the function will keep looping making it confusing for you.

12. This completes the fixture. Save your work.