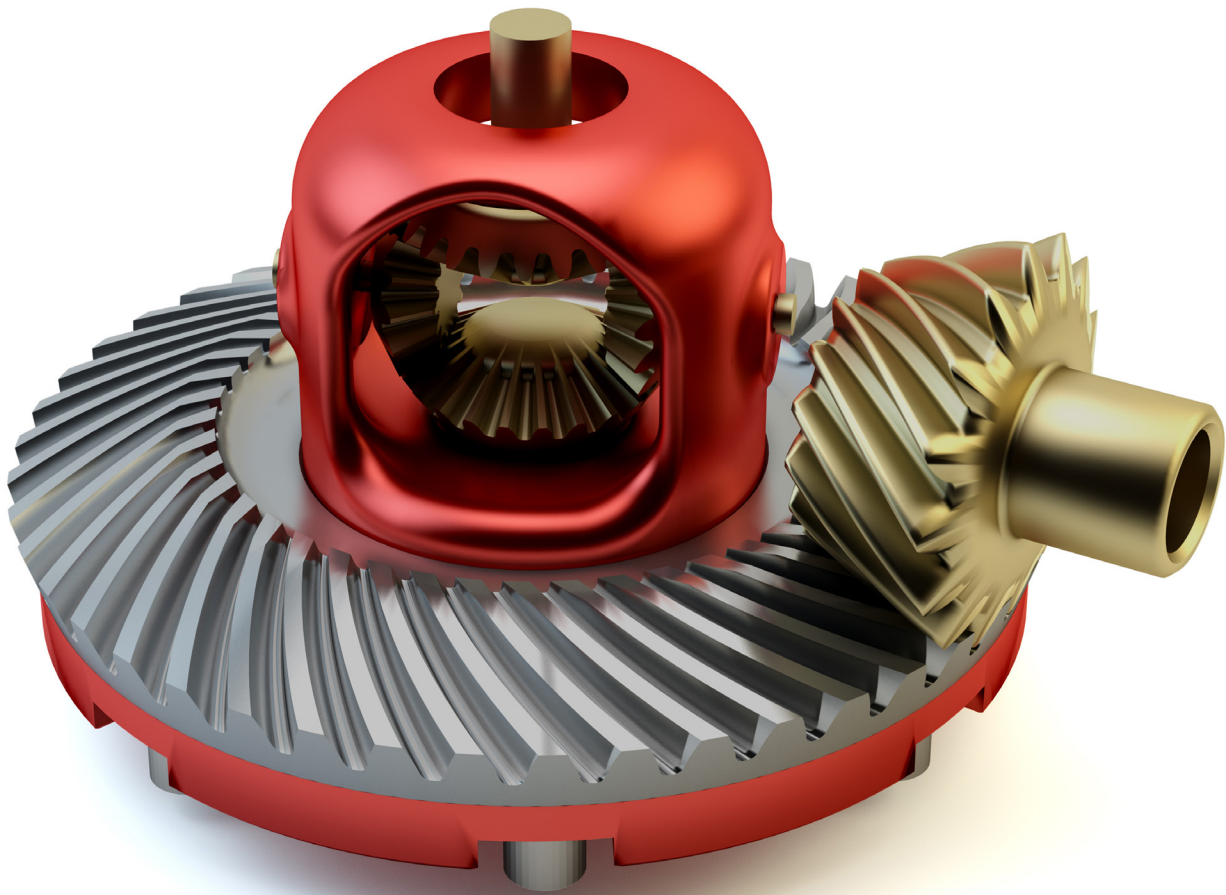


Michael J. Rider, Ph.D.

Designing with Creo® Parametric 5.0



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SKETCHER

Objectives

- ▶ Introduction to 2D sketcher
- ▶ All basic tools of sketcher explored
- ▶ Using sketcher constraints
- ▶ Modifying all dimensions at once (scaling)
- ▶ Sketching according to design intent
- ▶ Practice using the tools of sketcher
- ▶ Saving a copy of a 2D sketch
- ▶ Modifying an existing 2D sketch
- ▶ Sketcher exercise and the power of parametric models
- ▶ Sketcher design problems to reinforce concepts

▶ Sketcher Explored

Sketcher is the main creation tool of Creo Parametric. The sketcher toolbar ribbon is located at the top of the window. This section explores the many options of sketcher. Its basic icons are shown in Figure 3-1. The LMB (Left Mouse Button) is used to select geometry or to select a location when creating geometry.

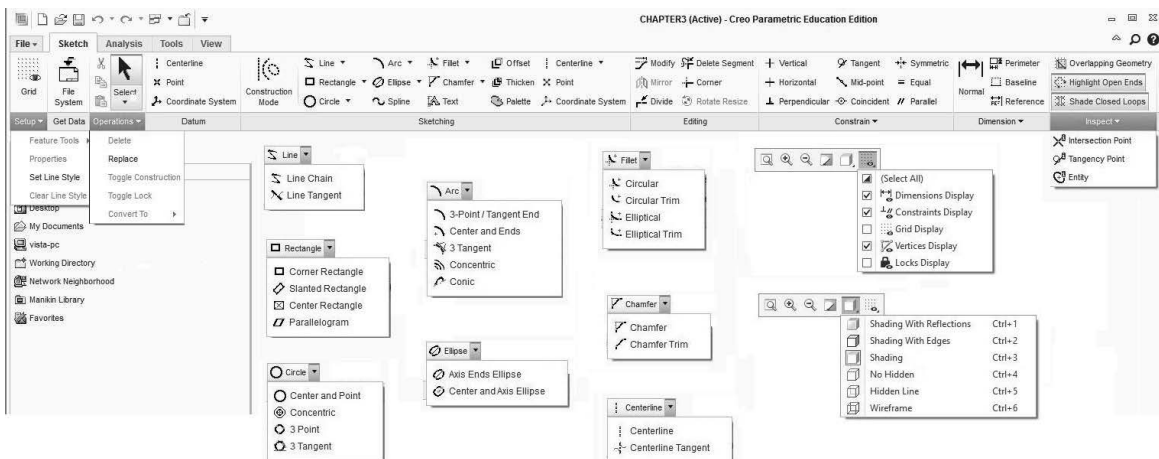


Figure 3-1 Sketcher Ribbon

Where multiple tools are present, the additional tools can be selected by moving the cursor on top of the small arrow icon (▼), then pressing the LMB. The row of options will appear. Move the cursor to the desired tool icon, then press the LMB again to select the tool.

As you create geometry, sketcher will place enough dimensions on the sketch to completely define it. These original dimensions are referred to as weak dimensions. The value of a weak dimension can be changed, thus making it a strong dimension, or a weak dimension can be replaced by a strong dimension added by the user which matches design intent.

Reminder:

LMB = press the Left Mouse Button down; used to select points or features.

RMB = press the Right Mouse Button down; used to search through a series of features or used to bring up a pop-up menu.

MMB = press the Middle Mouse Button down; used to cancel a command, place a dimension, or accept the current value.

Entering Sketcher Directly from Creo Parametric



Select the NEW FILE icon from the ribbon at the top of the screen. Save as >File>New. When the New file window opens, select **Sketch** as the type, then enter a valid filename without the extension. Pick **OK**.

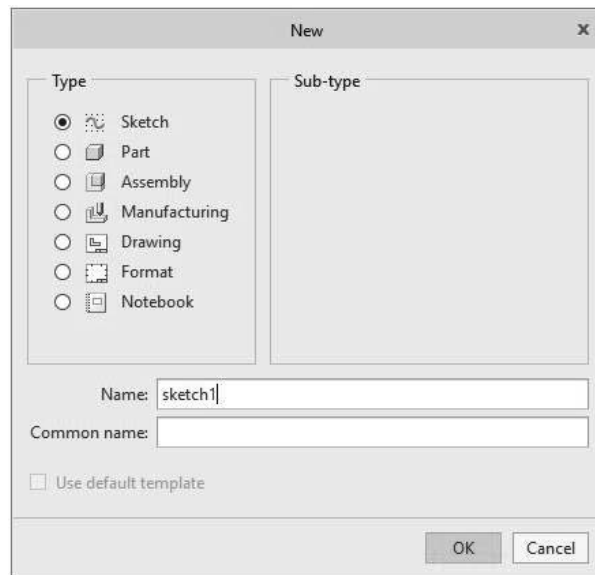


Figure 3-2 New Sketch Named

Entering Sketcher from Part Mode within Creo Parametric

This section assumes you are in part creation mode already.



To enter sketcher mode from part mode, *select* the SKETCHER icon at the top of the screen. *Pick* the plane that you wish to sketch on followed by orienting another plane toward the right, top, left, or bottom of the screen. *Pick* the **Sketch** button. If necessary in the tool bar at the top of the window, *select* the SKETCH VIEW icon to orient the sketching plane parallel to the display screen. The sketcher ribbon is slightly different at the left and right ends. These differences will be discussed later.

Sketcher Tools Explained

Now let's explain most of the sketcher icons. It is important to note that the LMB activates each of the sketcher tools.



Define the grid settings. Grid type can be either Cartesian or Polar. Grid Spacing can be either Dynamic or Static. X Spacing and Y Spacing define the distance between grid points in the X and Y directions. Grid Orientation defines the origin (0, 0) and the angular position of the X-axis relative to the horizontal.



The File System icon allows you to import data into the active sketch.



Cut, Copy, and Paste icons perform the standard cut, copy, and paste functions. Cut is <Ctrl>x. Copy is <Ctrl>c. Paste is <Ctrl>v.



Select Items tool allows you to select features already on the screen by moving the cursor over the item, then pressing the LMB. Holding down <CTRL> and selecting a feature adds the feature to the list of selected features. Selecting a feature already in the list of selected features removes the feature from the list. Selecting a blank area of the screen removes all features from the list of previously selected features.

Datums



Geometry Centerline—Draw an infinitely long geometric centerline by selecting two points with the LMB. This series of center lines can be continued until a new sketcher tool is selected or the user presses the MMB once. These centerlines can reference existing geometry as well as be available outside of sketcher.



Geometry Point—Create a geometry point using the LMB. This point can reference existing geometry as well as be known and visible outside sketcher.

This series of geometry points can be continued until a new sketcher tool is selected or the user presses the MMB once.



Geometry Coordinate System—Create a geometric coordinate system at the specified point by pressing the LMB. This coordinate system is known and is visible outside sketcher. This series of coordinate systems can be continued until a new sketcher tool is selected or the user presses the MMB once.

Sketching Tools

There are numerous sketching tools with limited space for them at the top of the screen; therefore many of these tools are stacked. If the tool is stacked, then a small down arrow icon appears to the right of the current icon tool. To see the stacked tools, pick the down arrow with the LMB. To select a stacked tool, move the cursor onto the desired tool, then press the LMB again.



Construction Mode—Toggles the creation of new geometry between construction line creation and solid geometry line creation. You can change the type of line segment created afterward by picking the line segments with the LMB, then pressing and holding the RMB until a pop-up menu appears. In the pop-up menu select Geometry to change construction lines to solid geometry lines, or pick Construction to change geometry lines into construction lines.



Line Chain—Draw a solid line from first LMB pick location to second LMB location pick. After the first line has been drawn, the cursor can be moved to a new location, then the LMB pick will draw a solid line from the end of the previous line to the new location. To start a new series of solid lines, press the MMB once, then begin anew. When the cursor gets near the end of an existing line, a small circle will appear around its end. Pressing the LMB will cause the end point of the newly created line to be exactly the same as the endpoint of the existing line. This series of solid lines can be continued until a new sketcher tool is selected or the user presses the MMB twice. After the series is complete, enough weak dimensions will appear on the sketch to completely define the sketch. These dimensions can be changed or replaced according to design intent.



Line Tangent—Draw a solid line tangent between two arcs or circles, which are selected using the LMB. This series of solid lines can be continued until a new sketcher tool is selected or the user presses the MMB once.



Corner Rectangle—Sketch a rectangle by selecting two opposite corners of the rectangle using the LMB. This series of rectangles can be continued until a new sketcher tool is selected or the user presses the MMB once.



Slanted Rectangle—Sketch a slanted rectangle by sketching one side of the slanted rectangle using the LMB twice, then moving perpendicular to this side to create the slanted rectangle's size. This series of 3-point slanted rectangles can be continued until a new sketcher tool is selected or the user presses the MMB once.



Center Rectangle—Sketch a rectangle by selecting the center point of the rectangle, then one of its four corners using the LMB. This series of rectangles can be continued until a new sketcher tool is selected or the user presses the MMB once.



Parallelogram—Sketch a parallelogram by sketching one side of the parallelogram using the LMB twice, then moving away from this side to create the parallelogram's shape. This series of 3-point parallelograms can be continued until a new sketcher tool is selected or the user presses the MMB once.



Center and Point Circle—Draw a circle by selecting the location of the circle's center with the LMB, then moving away from that point to create its radius. When the desired radius is shown, press the LMB to set it. This series of circles can be continued until a new sketcher tool is selected or the user presses the MMB once. The symbol **R** will appear by the newly created circle and an existing circle when their radii match.



Concentric Circle—Draw a new circle using the same center point as an existing circle. First the existing circle must be selected with the LMB, then a new circle appears. The new circle's radius follows the cursor and uses the center of the selected circle. The LMB sets the new circle's radius. Immediately another new circle appears using the same center point. The LMB will set this circle's radius or the MMB will cancel the creation of this new circle. After the MMB is pressed, then the user can repeat the entire sequence after picking a new reference circle. This series of concentric circles can be continued until a new sketcher tool is selected or the user presses the MMB once more.



3-Point Circle—Draw a circle through three points which are selected by pressing the LMB three times. This series of 3-point circles can be continued until a new sketcher tool is selected or the user presses the MMB once.



3-Tangent Circle—Draw a circle tangent to three features which are selected by pressing the LMB three times. This series of tangent circles can be continued until a new sketcher tool is selected or the user presses the MMB once.



3-Point/Tangent End Arc—Draw a circular (constant radius) arc by selecting its two endpoints using the LMB, then moving the cursor to size the arc's radius or make one end of the arc tangent to an existing feature. A plus sign (+) will appear at the center of the arc. This series of 3-point arcs can be continued until a new sketcher tool is selected or the user presses the MMB once.



Center and Ends Arc—Draw a circular (constant radius) arc by first selecting its center point using the LMB, then moving the cursor to size the arc's radius. Pressing the LMB sets the arc's radius. A plus sign (+) will appear at the center of the arc. Now the cursor will only move in a circular fashion around the arc's center point. Press the LMB to select the other end of the arc. This series of 3-point arcs can be continued until a new sketcher tool is selected or the user presses the MMB once.



3-Tangent Arc—Draw an arc tangent to three other features. First select a feature where the arc begins and where the arc must be tangent to using LMB. Next select a feature where the arc ends and where the arc must be tangent to using LMB. Third, select a feature that the arc must be tangent to using the LMB. If it is possible to create an arc tangent to all three features, one will be created. It is not necessary to select the exact tangent point on the features. Sketcher will adjust the end points of the arc to make the arc ends tangent to the features. This series of 3-point arcs can be continued until a new sketcher tool is selected or the user presses the MMB once.



Concentric Arc—Draw a concentric arc using the same center point as an existing circle or arc. First the existing circle or arc must be selected using the LMB. Next one end of the arc is selected using the LMB. The cursor will only move in a circular fashion around the selected center point. As the cursor moves, the new arc appears. Use the LMB to select the other end of this arc. Immediately the beginning of another new arc appears using the same center point. The LMB will set this arc's endpoint and radius or the MMB will cancel the creation of a new arc. After the MMB is pressed once, then the user can repeat the entire sequence after picking a new reference circle or arc. This series of concentric arcs can be continued until a new sketcher tool is selected or the user presses the MMB once more.



Conic Arc—Draw a conic (variable radius) arc by selecting its two endpoints using the LMB, then moving the cursor to size the conic arc. The conic arc's shape can be changed by changing its tangent angle on either end or the value of the conic arc. This series of 3-point conic arcs can be continued until a new sketcher tool is selected or the user presses the MMB once.



Axis Ends Ellipse—Draw an ellipse by selecting the end points of the major or minor axis, then moving perpendicular to this axis to size the other axis. The ellipse is shown as the cursor is moved perpendicular to the axis drawn first. The second axis's size is set by pressing the LMB. This series of 3-point ellipses can be continued until a new sketcher tool is selected or the user presses the MMB once.



Center and Axis Ellipse—Draw an ellipse by selecting its center using LMB, one end of its major or minor axis using LMB, then moving perpendicular to this axis to size the other axis. The ellipse is shown as the cursor is moved perpendicular to the axis drawn first. The second axis's size is set by pressing the LMB. This series of 3-point ellipses can be continued until a new sketcher tool is selected or the user presses the MMB once.



Spline—Draw a free-hand spline curve by selecting spline points using the LMB. The spline curve must go through these spline points. If the curve is closed, then the curve's beginning slope will match the curve's ending slope. This series of spline curves can be continued until a new sketcher tool is selected or the user presses the MMB once.



Circular Fillet—Draw a circular fillet or round tangent to two features. The radius of this tangent arc is displayed after its creation. If the two features are intersecting lines, then the line segments at the intersection are trimmed back to the point where the arc is tangent to the lines. *The location of the intersecting lines is marked with a dot.* If neither feature is not a line, the features are not trimmed. This series of arcs tangent to selected features can be continued until a new sketcher tool is selected or the user presses the MMB once.



Circular Trim Fillet—Draw a circular fillet or round tangent to two features. The radius of this tangent arc is displayed after its creation. If the two features are intersecting lines, then the line segments at the intersection are trimmed back to the point where the arc is tangent to the lines. *The location of the intersecting lines is NOT marked with a dot.* If neither feature is not a line, the features are not trimmed. This series of arcs tangent to selected features can be continued until a new sketcher tool is selected or the user presses the MMB once.



Elliptical Fillet—Draw an elliptical fillet or round tangent to two features at the two points selected on the features using the LMB. If the two features are lines, then the line segments in the area of the elliptical arc are trimmed back to the selected point and the arc is tangent to the line at that point. *The location of the intersecting lines is marked with a dot.* If neither feature is not a line, the features are not trimmed. This series of elliptical arcs tangent to selected points on existing features can be continued until a new sketcher tool is selected or the user presses the MMB once.



Elliptical Trim Fillet—Draw an elliptical fillet or round tangent to two features at the two points selected on the features using the LMB. If the two features are lines, then the line segments in the area of the elliptical arc are trimmed back to the selected point and the arc is tangent to the line at that point. *The location of the intersecting lines is NOT marked with a dot.* If neither feature is not a line, the features are not trimmed. This series of elliptical arcs tangent to selected points on existing features can be continued until a new sketcher tool is selected or the user presses the MMB once.



Chamfer Fillet—Draw a chamfer between two intersecting lines starting at the points selected using the LMB, then remove the line segments in the area of the intersection of the two lines. *The location of the intersecting lines is marked with a dot.* Two weak dimensions are created that reflect the length of the two legs of the chamfer. This series of chamfers can be continued until a new sketcher tool is selected or the user presses the MMB once.



Chamfer Trim Fillet—Draw a chamfer between two intersecting lines starting at the points selected using the LMB, then remove the line segments in the area of the intersection of the two lines. *The location of the intersecting lines is NOT marked with a dot.* Two weak dimensions are created or modified that reflect the length of the two remaining line segments. This series of chamfers can be continued until a new sketcher tool is selected or the user presses the MMB once.

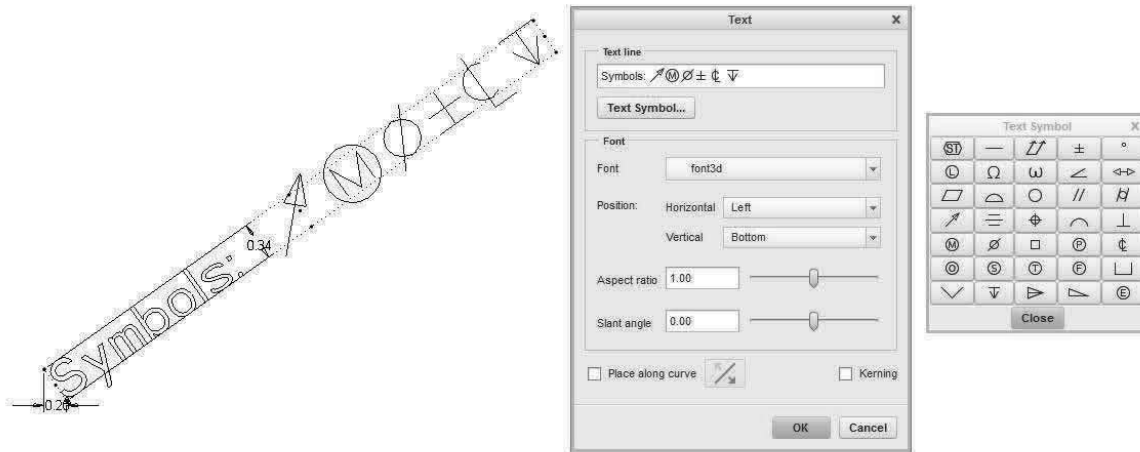


Figure 3-3 Text and Symbols




Text—Create alpha characters and symbols on the sketch. Use the LMB to select a start point on the sketching plane and a second point which will set the text height and orientation. The length of the construction line determines the height of the text, while the angle of the line determines the text orientation. The two weak dimensions can be changed or replaced using the dimensioning tool. The Text dialog box opens. A red diamond appears at the start point of the text. Type a single line of up to 79 characters of text. Selecting the Text Symbol button brings up a new window containing many symbols which can be incorporated into the line of text. See Figure 3-3. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB once.

The start point, by default, is the left bottom point of the text. The Left reference can be changed to Center or Right. The Bottom reference can be changed to Middle or Top. The aspect ratio affects the width to height ratio. Increasing this value spreads the characters out. The Slant angle tilts the characters making them appear more like italicized characters.

When you finish entering or modifying the text, *pick* the **OK** button. Double-clicking on the text using the LMB will bring this dialog box back so additional modifications can be made.



Text Along A Curve—To place text along a curve create the start and height points as before, then check the “Place along curve” box (Figure 3-4) followed by selecting the curve to follow using the LMB. Type the text as before.

Use the  button to change the direction of the text along the curve.



Offset—Create duplicate geometry an offset distance from the selected geometry. After selecting the Offset tool, the Type dialog box appears. In the type dialog box, select either single (one item), chain (series of interconnected items), or loop (series of interconnected items that make a closed loop). If you select single, then the feature selected with the LMB is offset a specified distance in the direction of the yellow arrow. If you select chain, then all features between the first feature picked with the LMB and the second feature picked are offset the specified distance. If it is not clear which line segments are selected, then a query box appears asking if this is the desired chain. You can accept selection, go to the next

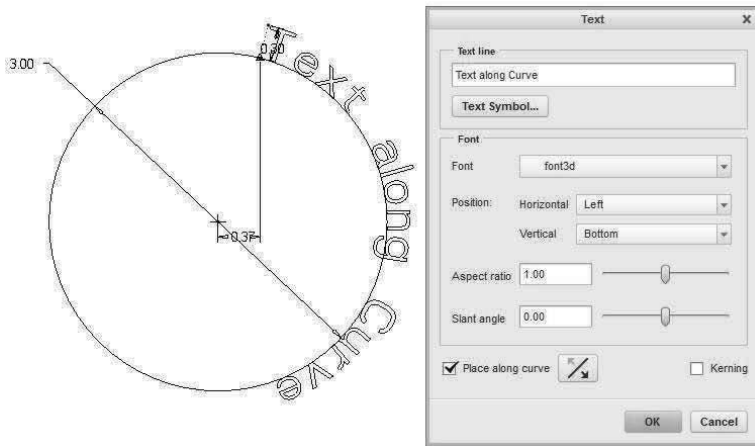


Figure 3-4 Text along a Curve

selection, back up to the previous selection, or quit. If you select loop, then all features in a loop that contains the first feature picked with the LMB are offset the specified distance. This tool continues to operate until a new sketcher tool is selected, the user closes the dialog window, or the user presses the MMB twice.

If you want to change the type, leave the offset value blank and press the <Enter> key. Then select the desired type from the Type dialog box.



Thicken—Create entities by offsetting an edge or a sketched entity on two sides. The 2-line offset may have open, flat, or rounded ends. The user is prompted for the thickness, which is the distance between the two new lines, then the offset in the direction of the arrow for the further of the two new lines. The thicken edge can be a single line, a chain of lines, or a closed loop.



Palette—Provides you with a customizable library of predefined shapes that you can readily import onto the active sketch plane. These shapes are presented in a palette. You can resize, translate, and rotate the shape after placing it on the screen.

The sketcher palette has tabs representing categories of sections. Each tab has a unique name and contains at least one section of a certain category. The four tabs with predefined shapes are:

- ▶ **Polygon**—contains regular polygons such as Triangle (3), Square (4), Pentagon (5), Hexagon (6), Heptagon (7), Octagon (8), Nonagon (9), Decagon (10), Dodecagon (12), Hexdecagon (16), and Icosagon (20).
- ▶ **Profiles**—contains common profiles such as C-shape, I-shape, L-shape, and T-shape.
- ▶ **Shapes**—contains miscellaneous common shapes such as Arc racetrack, Cross, Oval, Racetrack, Rounded rectangle, Wave 1, and Wave 2.
- ▶ **Stars**—contains regular star shapes such as 3-tip, 4-tip, 5-tip, 6-tip, 7-tip, 8-tip, 9-tip, 10-tip, 12-tip, 16-tip, and 20-tip.

Using a shape from the palette is similar to importing the corresponding section from another sketch. Each shape in the palette appears as a thumbnail with its name. These cross-sections appear using the default line style and color.

After selecting the PALETTE tool, select the worded-tab corresponding to the desired shape (Figure 3-5), then *double-click* using the LMB on the specific shape. Move the cursor to the desired location in the sketcher window and press the LMB again. The shape can also be dragged onto the screen. To do this, *select* the desired shape with the LMB. While holding down the LMB move the cursor onto the screen. The selected shape will follow. To position the shape, release the LMB. In either case, the Move & Resize window will appear. At this point the shape can be rotated or scaled in place by entering the corresponding value. On the screen the shape has two handles, one for rotation and one for scaling. Use the

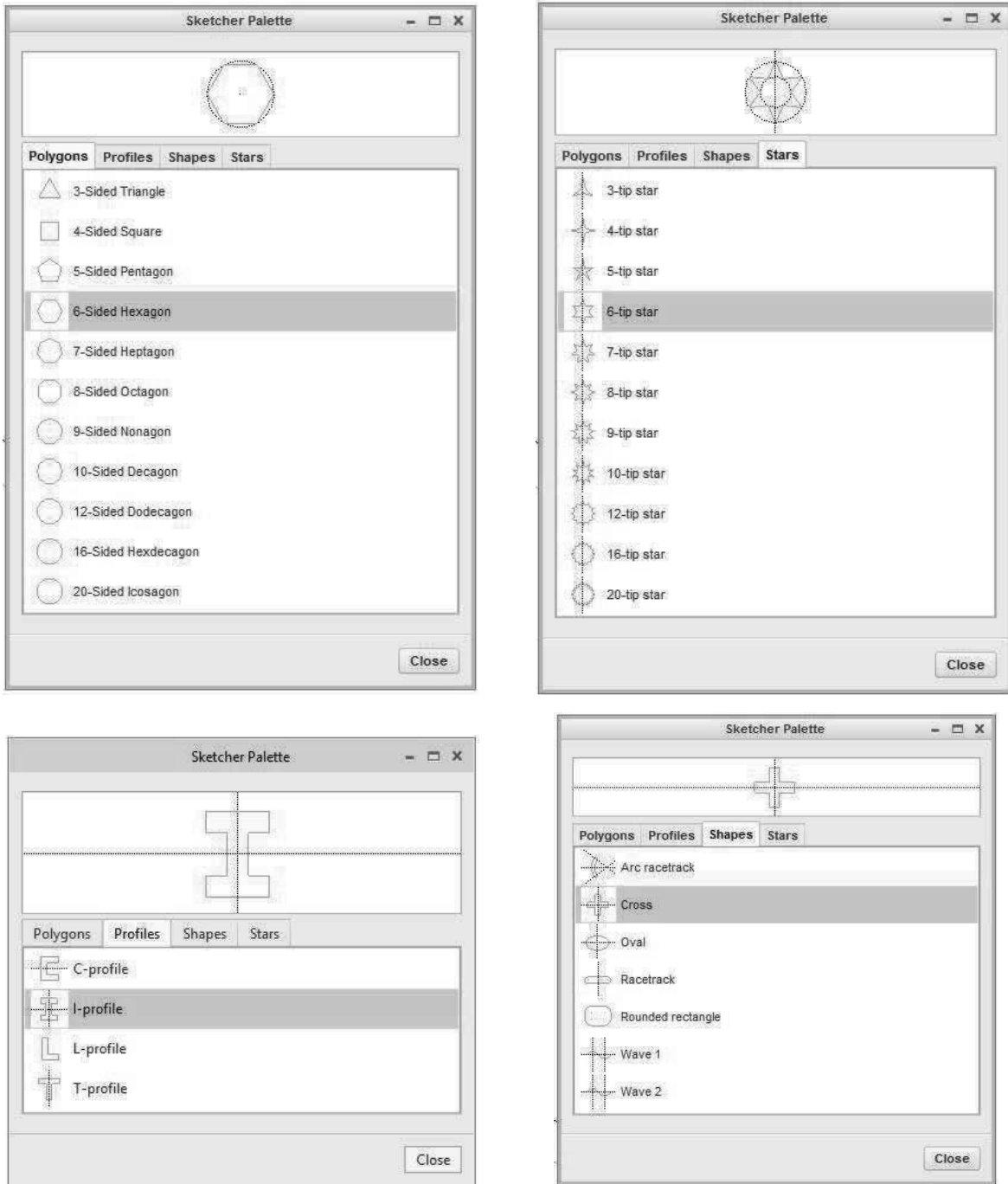


Figure 3-5 Sketcher Palette

cursor and the LMB to free-hand rotate or scale the shape. Move onto the handle, hold down the LMB, move the cursor, then release the LMB.



Construction Centerline—Draw an infinitely long sketcher centerline by selecting two points with the LMB. This series of centerlines can be continued until a new sketcher tool is selected or the user presses the MMB once. These sketcher centerlines are not known or visible outside of sketcher.



Construction Centerline Tangent—Draw an infinitely long sketcher centerline tangent to two circles or arcs using the LMB. This series of center lines can be continued until a new sketcher tool is selected or the user presses the MMB once. These sketcher centerlines are not known or visible outside of sketcher.



Construction Point—Create a construction point using the LMB. This point is known only inside sketcher and is not visible outside sketcher. This series of construction points can be continued until a new sketcher tool is selected or the user presses the MMB once.



Construction Coordinate System—Create a construction coordinate system at the specified point by pressing the LMB. This coordinate system is known only inside sketcher and is not visible outside sketcher. This series of coordinate systems can be continued until a new sketcher tool is selected or the user presses the MMB once.

Editing

Modify the selected dimension values in a separate window (Figure 3-6). This tool can be initiated in two different ways. The common way is to highlight the dimensions to be modified using the LMB for the first and <CNTRL> and LMB for the others, then selecting the tool. A similar approach is to draw an imaginary box around the sketch and dimensions, then selecting this tool. The second way is to select the tool first, then select a dimension to be modified using the LMB. If additional dimensions are to be modified at the same time, hold down the <CTRL> key

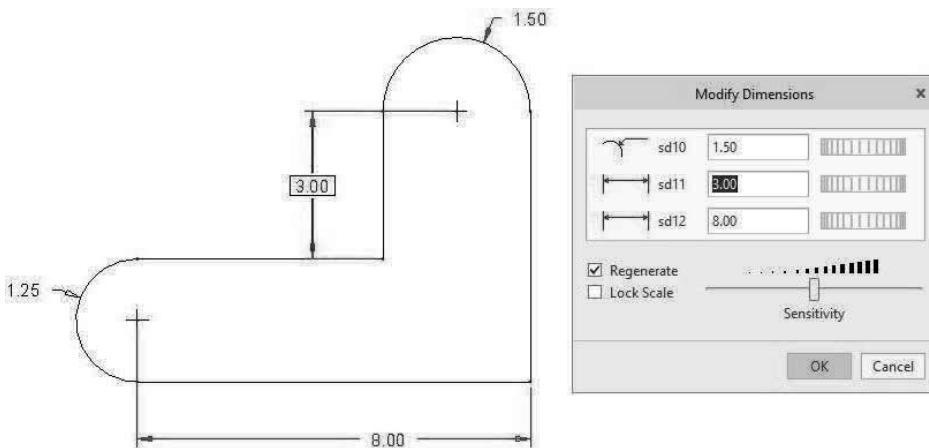


Figure 3-6 Modify Selected Dimensions

and select these dimensions with the LMB. Each dimension will be added to the list in the Modify Dimensions box as it is selected.

The Regenerate check box will cause the sketch to change shape each time a single dimension is changed. Uncheck this box if you want to change all dimensions without seeing their effect until the green check mark is picked.

The Lock Scale check box, when checked first, allows the user to change just one dimension followed by picking the green check mark and seeing all dimensions change by the same scale factor as the modified dimension. This will cause the original shape of the sketch to remain the same. This is a great way to get all dimensions close to their desired value at once if the original sketch is approximately the correct shape, but the wrong size.

Selecting the red X button cancels the request.




Mirror—Mirror a selection of features about a specified centerline, thus there must be a centerline present in the sketch. First, select the features to be mirrored either by selecting them one at a time using the LMB and <CTRL> LMB, or drawing an imaginary box around them, then pick the Mirror tool using the LMB. Next, select the centerline the features are to be mirrored about. The mirrored copy will appear on the opposite side of the centerline. Note that you cannot pick this tool unless you have already selected at least one feature.



Divide Entity at This Point—Divide a feature at the point of selection located by pressing the LMB. This will break a straight or curved line segment into two parts. It will create a starting and ending point on a circle or an ellipse.



Delete Segment—Will remove any line segment that is drawn through while holding down the LMB. They will be erased when the LMB is released. All line segments drawn through with a red line will turn red and be erased when the LMB is released. Use <CTRL>Z or the undo tool  to undo the deletions you just made if you accidentally remove a wanted line segment. Features can also be removed from the sketch by selecting the feature with the LMB, then pressing the <Delete> key.



Corner—This tool is used to trim the intersection of two line segments back to the intersection point. Select the two line segments using the LMB. The portion of the line segment where you selected the line will remain after the trim.

Another neat feature of this tool is its ability to connect at their intersection point two line segments (straight or circular) that do not touch, but would touch if extended. Select each line segment with the LMB. Each line segment will be extended or trimmed as necessary so the two line segments meet and end at their intersection point.

This tool will not work on a circle, an ellipse, or a centerline since they are considered continuous, without ends.



Rotate Resize—The Rotate and Resize tool can be used to translate, rotate, or scale the selected features. First, select the features to be modified either by selecting them one at a time using the LMB and <CTRL> LMB, or drawing an imaginary box around them, then pick the Move & Resize tool using the LMB. The Move & Resize dashboard will appear. At this point the features can be translated, rotated, or scaled by entering the corresponding value. On the screen the shape has

three handles: one for translation, one for rotation, and one for scaling. Use the cursor and the LMB to free-hand move, rotate, or scale the features. Move the cursor onto the handle, hold down the LMB, and move the cursor. Release the LMB when the modification is complete.

If you want to move the selected features parallel to or perpendicular to a line segment, place the cursor in the first box in the dashboard, then select the line segment to be used as reference.

If you want to rotate the selection relative to a point, place the cursor in the fourth box in the dashboard, then select the end point to be used as the rotational reference point.

Pick the green check mark (or press the MMB) when all modifications are complete. Pick the red X to cancel the changes made.

Constraint



Vertical Constraint—Force the selected line to be vertical. After selecting this tool, select the desired line using the LMB. If it is possible to make this line vertical, the sketch will be modified accordingly. A “V” will appear near any line that is vertical. Instead of a line segment two points can be forced to be vertically aligned by selecting the two points using the LMB. A small rectangular marker will appear near each of the two points indicating that they are vertically aligned. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB once.



Horizontal Constraint—Force the selected line to be horizontal. After selecting this tool, select the desired line using the LMB. If it is possible to make this line horizontal, the sketch will be modified accordingly. An “H” will appear near any line that is horizontal. Instead of a line segment two points can be forced to be horizontally aligned by selecting the two points using the LMB. A small rectangular marker will appear near each of the two points indicating that they are horizontally aligned. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB once.



Perpendicular Constraint—Force two selected lines to be perpendicular to each other. After selecting the tool, select the two lines using the LMB. A perpendicular symbol (\perp) will appear near the intersection of these two lines indicating their status. If the two line segments do not intersect physically, then a perpendicular symbol with a subscript (\perp_x) is placed on each of the two lines. This tool is not limited to just line segments; any feature with a tangent endpoint can be made perpendicular to another feature. For example, the endpoint of an arc can be made perpendicular to a line segment. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB once.



Tangent Constraint—Force a line, an arc, or a circle to be tangent to an arc or a circle. After selecting the tool, select the two features using the LMB. The symbol “T” will appear near the intersection of these two features indicating their status. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB once.



Mid-point Constraint—Force a point to locate itself at the midpoint of a line segment or arc. After selecting the tool, select the approximate location of the midpoint using the LMB, then select the line segment or arc using the LMB. The point will move to the midpoint of the selected line or arc. The symbol “M” will appear at this location. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB once.



Coincident Constraint—Make two points coincident, that is, the exact same point. After selecting the tool, select the two points using the LMB. If the endpoints are governed by weak dimensions, then the common point is somewhere between the two original points. If the endpoints are constrained by strong dimensions, then a resolve sketch window will appear showing all possible necessary changes to accomplish the task.

In the example (Figure 3-7) there are seven possible modifications, four of them currently not visible in the scroll window. The possible solutions are highlighted. The possible changes to the sketch are:

1. Delete or make the 2.40 dimension a reference dimension.
2. Delete or make the 2.60 dimension a reference dimension.
3. Undo or delete the requested coincident point request.
4. Make the two vertical lines unequal by deleting left constraint L_1 .
5. Allow the horizontal top line to become non-horizontal by deleting constraint.
6. Allow the horizontal bottom line to become non-horizontal by deleting constraint.
7. Allow the vertical left line to become non-vertical by deleting constraint.
8. Allow the vertical right line to become non-vertical by deleting constraint.
9. Make the two vertical lines unequal by deleting right constraint L_1 . (Same as 4.)

Once a resolution has been reached, sketcher continues. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB once.

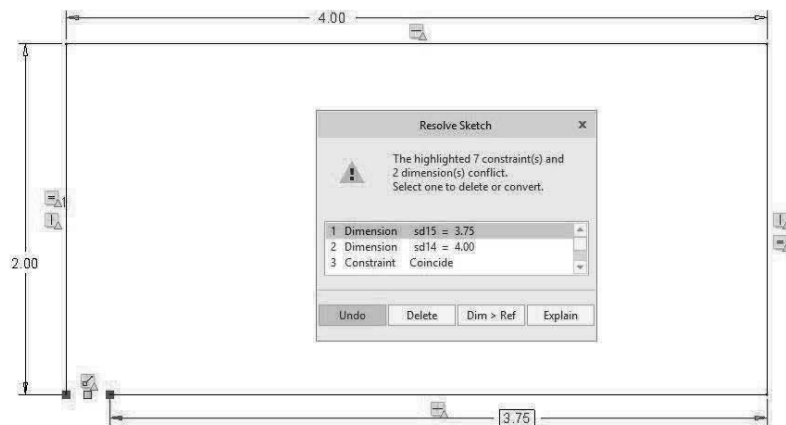


Figure 3-7 Resolve Sketch Constraints



Symmetric Constraint—Force two points to be symmetric about the selected centerline. There must be a centerline present to use this command. After the tool is selected, pick the governing centerline using the LMB, then select the two points (one on each side of the centerline) using the LMB. The two points will become symmetric about the centerline. In Creo Parametric you can select the two points and the centerline in any order. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB once.



Equal Constraint—Force two or more features to be equal size. After the tool is selected, select the governing feature (line length or radius) using the LMB, then using the LMB select all features that you want to be the exact same size. The symbol “Lx” will appear next to line segments of the same length. The symbol “Rx” will appear next to radii of the same size. Line segments and radii cannot be made equal to each other using this constraint.

Press the MMB once to select a new governing feature (line length or radius). This tool continues to operate until a new sketcher tool is selected or the user presses the MMB twice.



Parallel Constraint—Force two lines to be parallel. After the tool is selected, select the governing line segment using the LMB, then using the LMB select all other line segments that you want to be parallel to the governing line segment. The symbol “//x” will appear next to line segments that are parallel.

Press the MMB once to select a new governing line segment. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB twice.

Dimension



Normal Dimension—Add a strong dimension to the existing sketch. A weak dimension will disappear. If the dimension added causes a conflict with existing dimension, then a conflict box will appear allowing you to resolve the conflict. The new dimension or an existing dimension can be made into a reference dimension or deleted at this time. Extension and dimension lines will be automatically created for the new dimension. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB once.

To change the value of a dimension double-click on it using the LMB, then type the new value followed by the <ENTER> key. The dimensioned feature will adjust its size to the new value if possible.



To dimension a single line segment, select the line using the LMB, then move perpendicular to the line segment. Position the cursor where you want the dimensional value to appear and press the MMB.



To dimension the distance between two points, use the LMB to pick each of the two points, then position the cursor where you want the dimensional value to appear and press the MMB. If you want the vertical distance between two offset points, move the cursor horizontally away from the two points before pressing the

MMB. If you want the horizontal distance between two offset points, move the cursor vertically away from the two points before pressing the MMB. If you want the length of the slanted line between the two points, move the cursor perpendicular to the slanted line before pressing the MMB.



To dimension the radius of an arc or circle, select the arc or circle by pressing the LMB, then position the cursor where you want the dimensional radius to appear and press the MMB.



To dimension the diameter of an arc or circle, select the arc or circle by pressing the LMB twice, then position the cursor where you want the dimensional diameter to appear and press the MMB.



To dimension the major or minor axis radius of an ellipse or an elliptical arc, select the ellipse by pressing the LMB, then press the MMB. A dialog box will appear asking if you want the major or minor axis. Select one of them, then pick the **Accept** button. Rx will appear in front of the first (major) axis's radius and Ry will appear in front of the second (minor) axis's radius. Note that the major axis is defined by the first axis line you drew when creating the ellipse; it is not necessarily the larger of the two axes nor is it necessarily horizontal or vertical.



To dimension the angle between two line segments, select the first line using the LMB, select the second line using the LMB, then position the cursor where you want the dimensional angular value to appear and press the MMB.

Figure 3-8 below shows four different ways to dimension an angle between two line segments. If you move the cursor between the line segments, then you get the angle between the line segments such as 80 degrees. If you move the cursor opposite to between the line segments, then you get the angle outside the line segments such as 280 degrees. If you move the cursor to the left of the line segments, then you get the angle between the line segments on the left side such as 100 degrees. If you move the cursor to the right of the line segments, then you get the angle between the line segments on the right side such as 100 degrees.

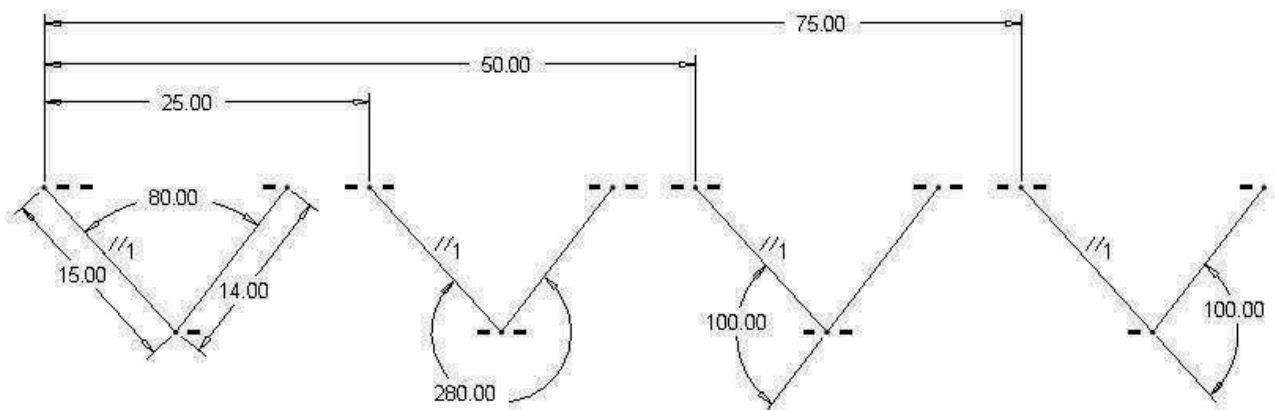


Figure 3-8 Dimensioning an Angle



To dimension the length of an arc, select one end of the arc using the LMB, select the arc using the LMB, select the other end of the arc using the LMB, then position the cursor where you want the dimensional arc length value to appear and press the MMB.

To convert an arc length dimension to an angular dimension, select the arc length dimension using the LMB, then press and hold the RMB to bring up a pop-up menu. Pick “Convert to Angle” from this list. To convert this angular dimension back to an arc length dimension, select the dimension with the LMB, press and hold the RMB to get the pop-up menu, then pick “Convert to Length” from the list.



To dimension a conic arc, select the conic arc by pressing the LMB, then position the cursor in the area where you want the dimension to appear and press the MMB.



Perimeter—Add a perimeter dimension to the sketch after selecting a dimension which can vary when the perimeter dimension is modified. First select the geometry that makes up the perimeter using the LMB. Hold down the <CTRL> key to select multiple features. Press the MMB once when finished. Using the LMB, select the dimension which will vary to keep the perimeter dimension correct. Type the desired perimeter dimension followed by the <Enter> key.

A perimeter dimension is used to dimension the total length of a chain or loop of entities. You must select a dimension that the system can adjust to obtain the desired perimeter. This dimension is called the varying dimension and will have the letters “var” after its value. When you modify the perimeter dimension, the system modifies the varying dimension accordingly. You cannot modify the varying dimension because it is a driven dimension. If you delete the varying dimension, the system deletes the perimeter dimension as well. You cannot create perimeter dimensions for parallel blends and variable section sweeps.



Baseline—Create an ordinate dimension baseline, either vertical or horizontal, representing 0.00. The baseline dimension is typically placed on a drawing to locate the origin of a part when N/C machining is involved in its production. Add a baseline dimension of 0.00 to your sketch by selecting the feature using the LMB, then moving the cursor to the desired location for the baseline value and pressing the MMB. Select Horizontal or Vertical in the pop-up window. This tool continues to operate until a new sketcher tool is selected or the user presses the MMB.



Reference Dimension—Add a reference (driven by other values) dimension to the existing sketch. Reference dimensions are added the same way as regular dimensions. (See adding dimensions above.) These dimensions cannot be changed by selecting them, then typing a new value. This tool continues to operate until a new sketcher tool is selected, or the user presses the MMB once.

Reference dimensions appear on sketches for information purposes only. Therefore, they are read-only and cannot be used to modify the shape. They are automatically updated during regeneration if changes are made to the sketch.

Inspect



Overlapping Geometry—Pick the Overlapping Geometry icon to highlight the sketcher geometry that overlaps so that you can correct the problem.



Highlight Open Ends—Pick the Highlight Open Ends icon to highlight using green dots, the line segments that are not connected to anything, thus they are open ends. (Should be active.)



Shade Closed Loops—Pick the Shade Closed Loops icon to fill in all closed figures so you can see which sections of your sketch are not closed for one reason or another. (Should be active.)

Figure 3-9 below shows a rectangle that is properly closed on the left. The middle figure has red highlighted lines because of overlapping line segments. This could be caused by duplicate line segments where at least one of the red lines is duplicated. The triangular figure on the right has an open-ended line segment (green dots) and overlapping line segments in its lower right corner.

Save Sketch

File>Save saves the sketch under the name you selected prior. *Pick **OK*** to save the file.

Close Sketcher from Part Mode

If you entered sketcher from part mode, then the following two icons appear at the far right side of the ribbon.



Accept the changes made in the sketcher and exit sketcher.



Cancel the changes made in sketcher and exit sketcher.

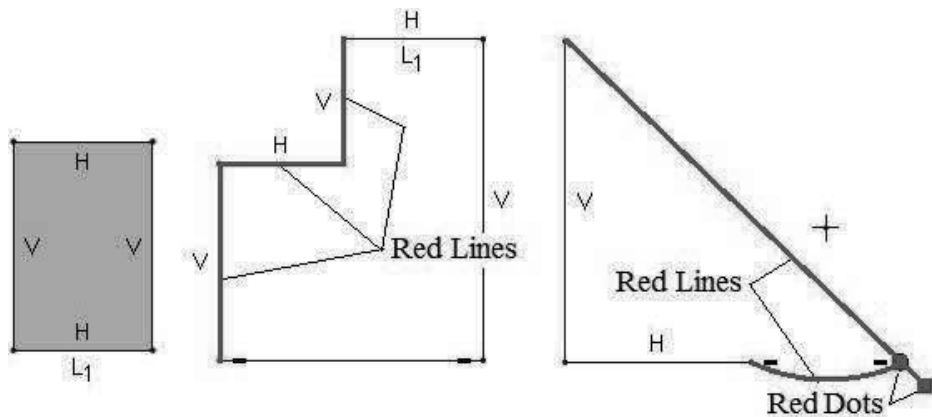


Figure 3-9 Sketched Sections

Set up Sketcher from Part Mode

After selecting the sketching plane and the orientation plane, the user typically orients the sketching plane parallel to the display screen using the *Sketch View* icon in the upper left corner of the window. See Figure 3-10.



Sketch Setup—Allows you to redefine the sketch plane and the sketch orientation. *Pick* the **Sketch** button to continue with changes or **Cancel** to continue without changes.



Select References—From previous work to be used when defining the current sketch. These references can be used for constraints or dimensions. *Select* the references using the LMB. *Pick* the **Close** button when done.



Sketch View—Orients the sketching plane parallel to the display screen. This icon is also found in the graphics toolbar just below the ribbon.

Sketcher Graphics Toolbar

The sketcher graphics toolbar is located just below the ribbon in the graphics area. It contains many of the commands that are used regularly. See Figure 3-11.



Refit—Adjust the zoom level to fully display the object on the screen.

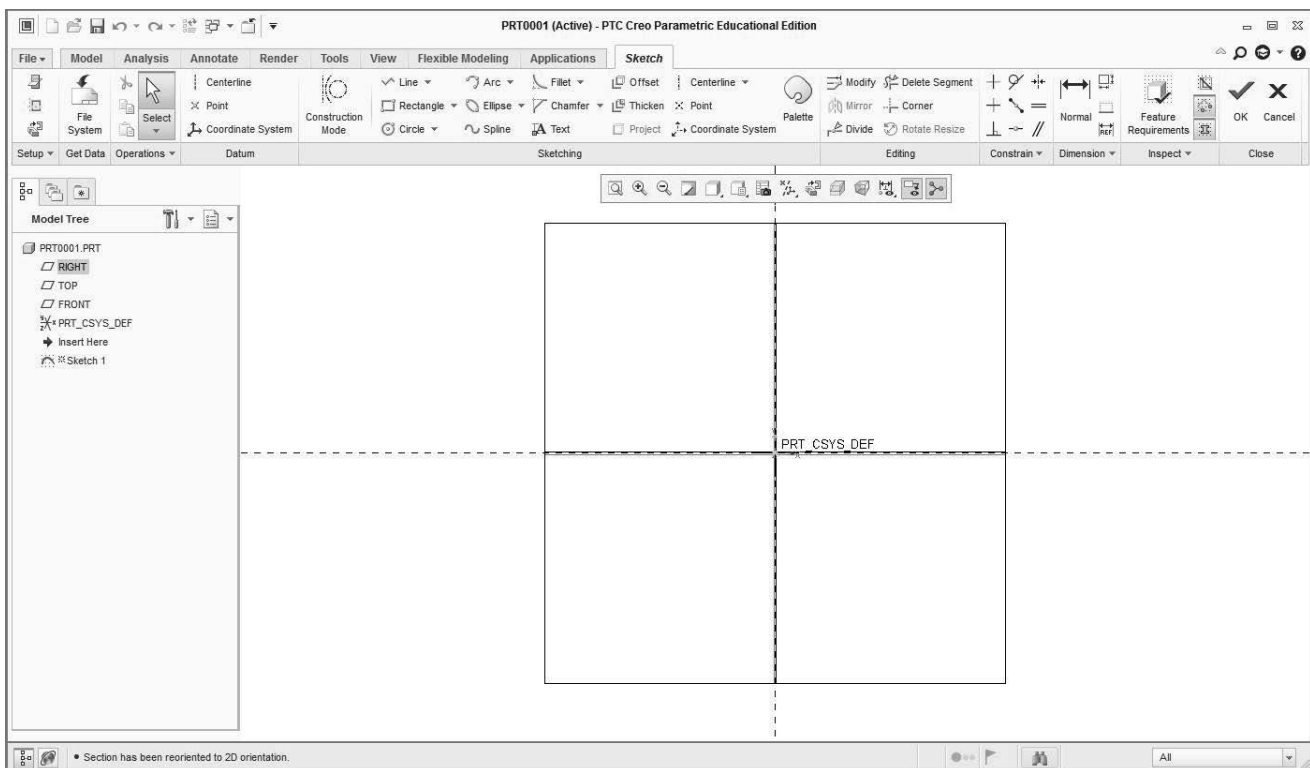


Figure 3-10 Sketcher Window when Entered from Part Mode

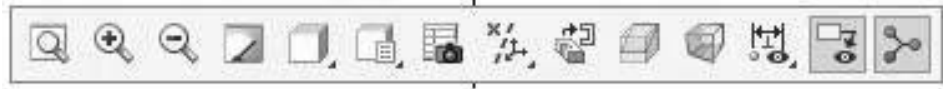







Figure 3-11 Sketcher Graphics Toolbar for Part Creation


 **Zoom In**—Zoom in on target geometry to view it in greater detail.


 **Zoom Out**—Zoom out to gain a wider perspective on the geometry.

 **Repaint**—Redraw the current view (refresh).

 **Display Style**—Determines how the object is to be viewed. See Figure 3-12.


 **Saved Orientations**—Selects the view orientation including user-defined orientations. See Figure 3-13. The *reorient...* tool at the bottom of the pull-down menu lets the user create new user-defined orientations.


 **View Manager**—Lets the user create, set, or delete simplified representations; create cross-sections; create, redefine, or set layers; create, set, or delete views; and create, redefine, or set combined states.

 **Datum Display Filter**—Allows the user to control the display of datum axes, datum points, datum coordinate systems, and datum planes. See Figure 3-14.

 **Sketch View**—Orient the sketching plane parallel with the screen.

 **Clip Model**—Hide the model geometry in front of the sketching plane.

 **Perspective View**—Toggle perspective view.

 **Sketcher Display Filter**—Allows the user to control the display of sketcher dimensions, sketcher constraints, the sketcher grid, and sketcher vertices (end points). This tool only shows up when you are in sketcher. See Figure 3-15.

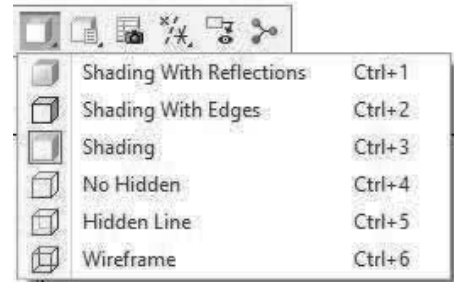


Figure 3-12 Object Viewing Options

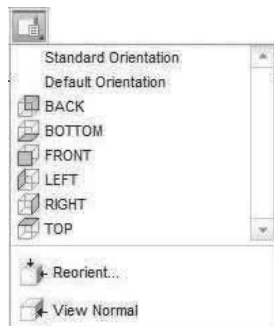


Figure 3-13 Named View List

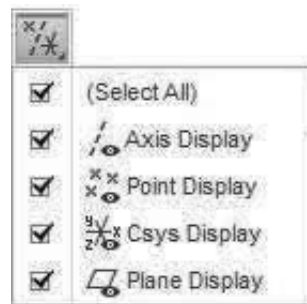


Figure 3-14 Datum Display Control

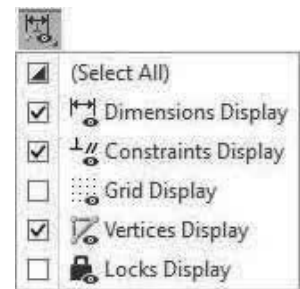


Figure 3-15 Sketcher Display Control



Annotation Display—Allows the user to turn on or turn off 3D annotations and annotation elements.



Spin Center—Allows the user to show or hide the spin center. The object will rotate about the spin center if the spin center is visible. The object will rotate about the current position of the mouse cursor if the spin center is hidden.

► Sketcher Practice

This section allows you to practice the necessary skills needed to use sketcher in Creo Parametric. Do each of the exercises below as quickly as possible. Go back and read about the sketcher command if you have trouble using the tool or forgot how to use it.

Step 1: Start Creo Parametric by *double-clicking* with the LMB on the CREO PARAMETRIC icon on the desktop, or from the Program list: Creo Parametric.

Step 2: Set your working directory, by *selecting the **Select Working Directory*** icon in the Home ribbon. Locate your working directory, and then *pick OK*. See Figure 3-16.

Step 3: Create a new sketch, named “sketcher_exercises” (Figure 3-17).

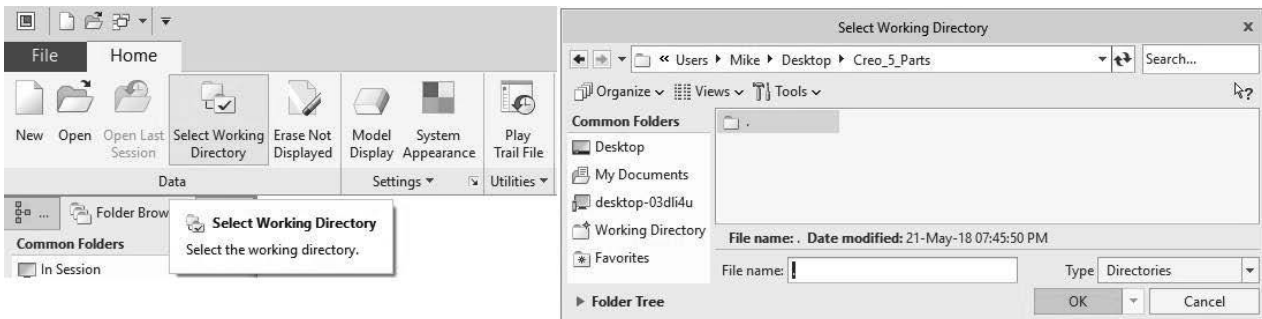


Figure 3-16 Set Working Directory

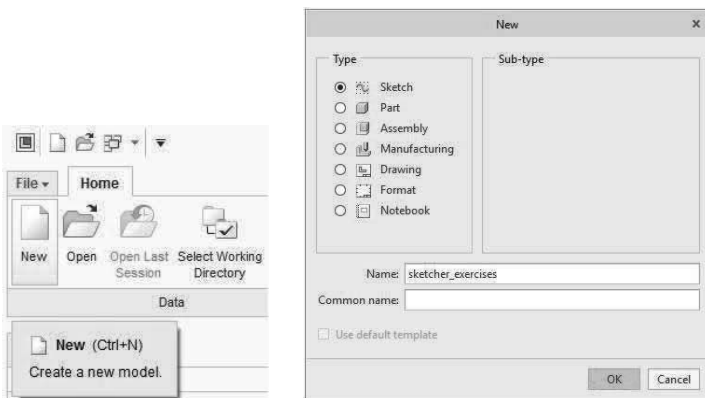


Figure 3-17 New Sketcher File

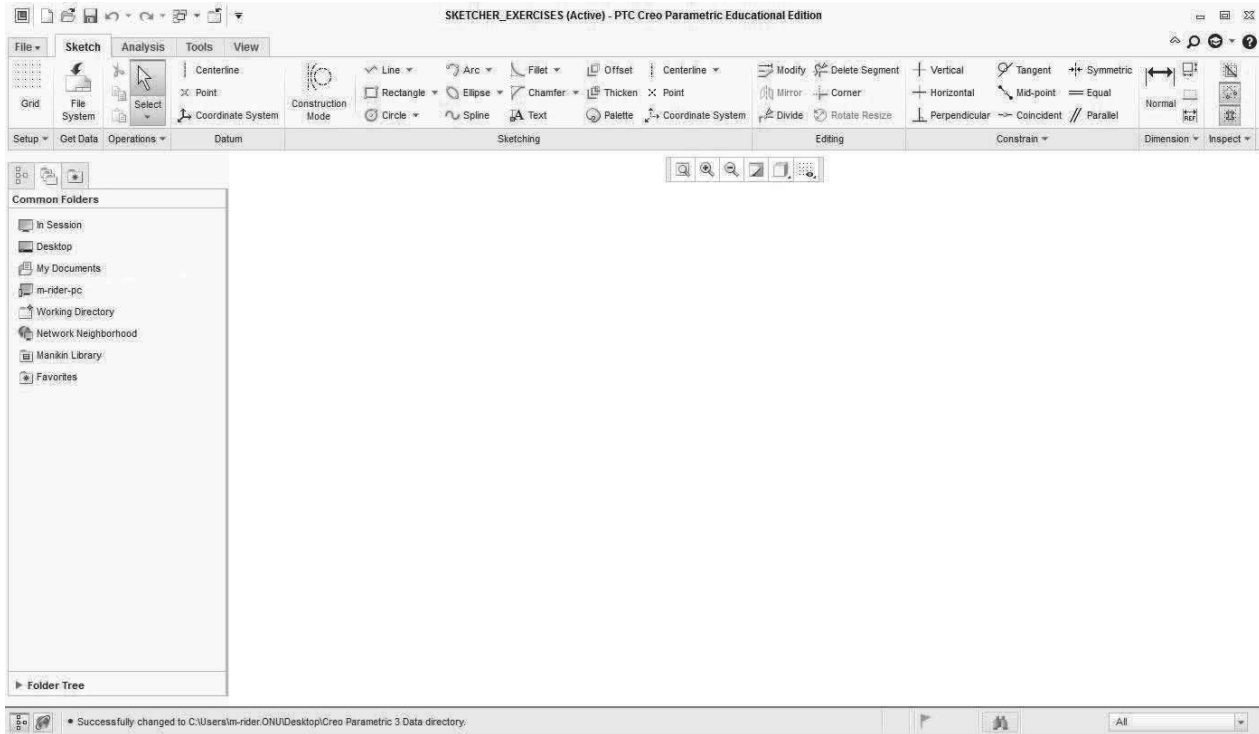


Figure 3-18 Sketcher Window

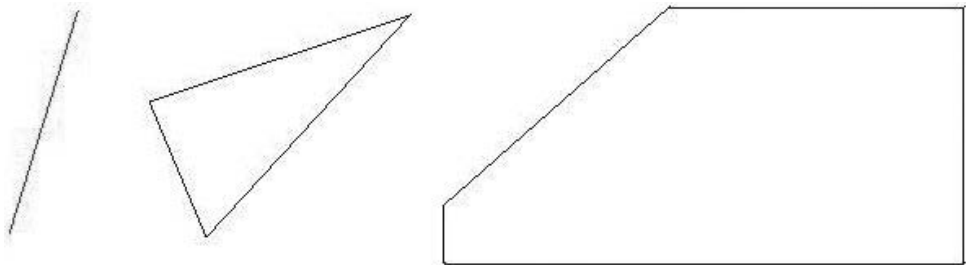



Figure 3-19 Sketch Lines

Your sketcher window’s drawing area may be colored. For purposes of this book, the sketcher window will be white. (This can be done by *selecting File>Options. Select System Appearances. Set System Colors to "Black on White."*) See Figure 3-18. Sketcher commands and frequently used icons are located at the top of the window. For now, let’s concentrate on the sketcher tools located in the sketch tab.

Step 4: Use  to draw Figure 3-19. Do not concern yourself with the size or dimensions of the sketch.

Did you use the MMB to end one figure before drawing the next figure? If not, repeat the exercise. Draw the figure on the right, press MMB, draw the triangle in the middle, press MMB, then draw the line on the left, and press MMB twice to terminate this tool.

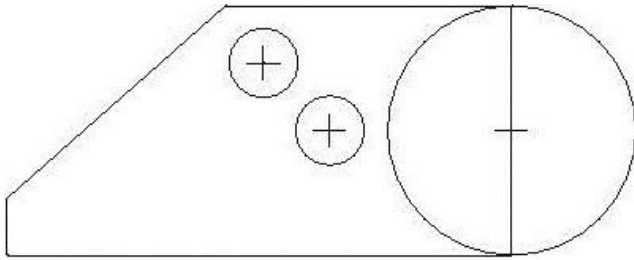


Figure 3-20 Delete Lines and Add Circles

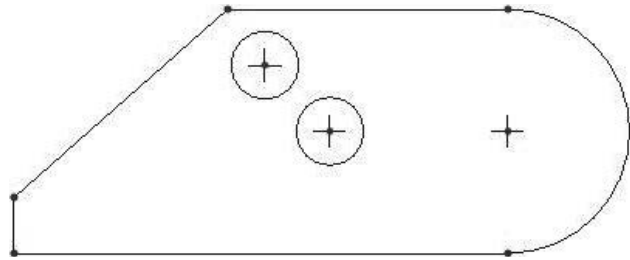






Figure 3-21 Delete Lines

Step 5: Use  to erase the line on the left and the middle triangle. With the cursor near the line, *press and hold* the LMB. *Drag* the cursor through the line and the three sides of the triangle. They should turn red. *Release* the LMB, and they are erased.

Step 6: Use  to draw a circle on the right end as shown in Figure 3-20. Also, draw two circles in the interior of the figure. Draw the first circle at the same horizontal position as the big circle, then draw a second circle the same size as the previous circle centered up and to the left of it as shown. Press MMB to terminate this command.

Step 7: Use  to erase the line on the right end of the original figure and the left half of the circle. See Figure 3-21.

Step 8: Use  to dimension the sketch according to design intent. The first step is to dimension the figure per design intent as shown in Figure 3-22. Don't worry about the value of the dimensions yet.

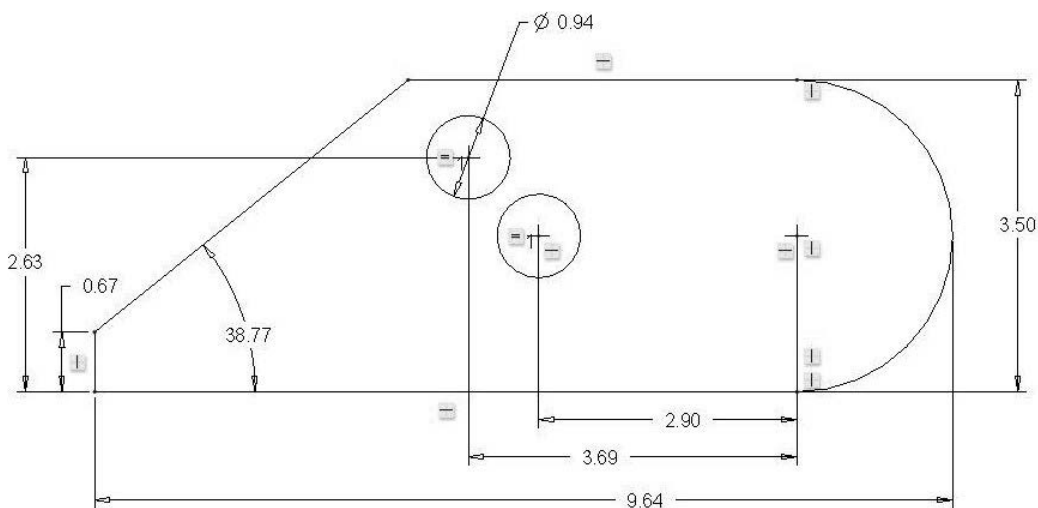



Figure 3-22 Add Dimensions

Design Intent

The height of the figure must be 3.00 inches. The overall length must be 7.50 inches. The slanted side must be at a 40-degree angle. The two small holes must be 0.75 inches in diameter. The center of the first small hole must be 3.00 inches to the left of the big radius's center. The center of the second small hole must be 1.00 inches from the bottom of the figure and 4.00 inches from the center of the big radius. The short vertical leg on the left side must be 0.50 inches long.

Now modify each dimension to reflect the design intent. See Figure 3-23.

Step 9: Use  to add a reference dimension showing the diameter of the other small hole, the value of the big radius, and length of the slanted line. These dimensions are not needed to define the figure, thus they are for reference only (Figure 3-24). Normally you would not do this, but we are practicing here.

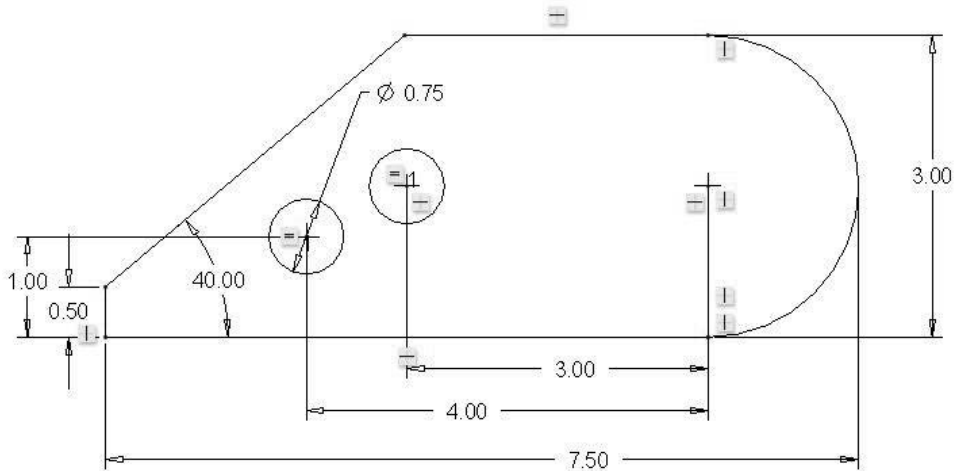


Figure 3-23 Design Intent Dimensions

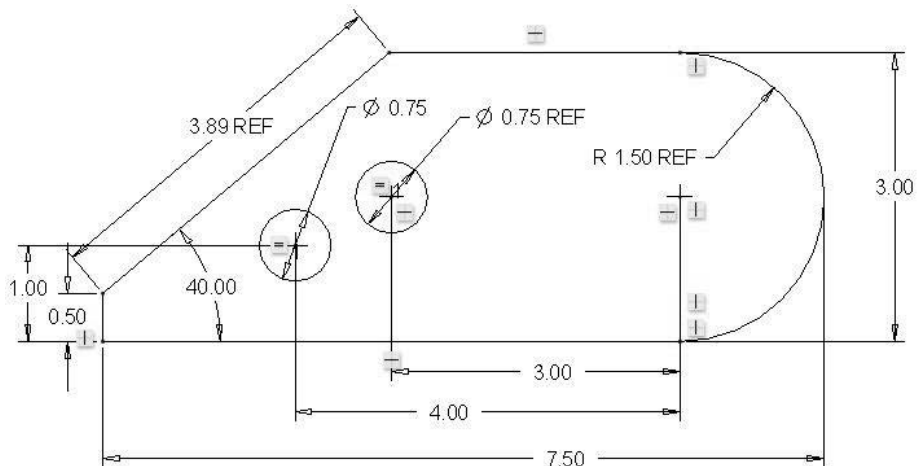


Figure 3-24 Design Intent Dimensions

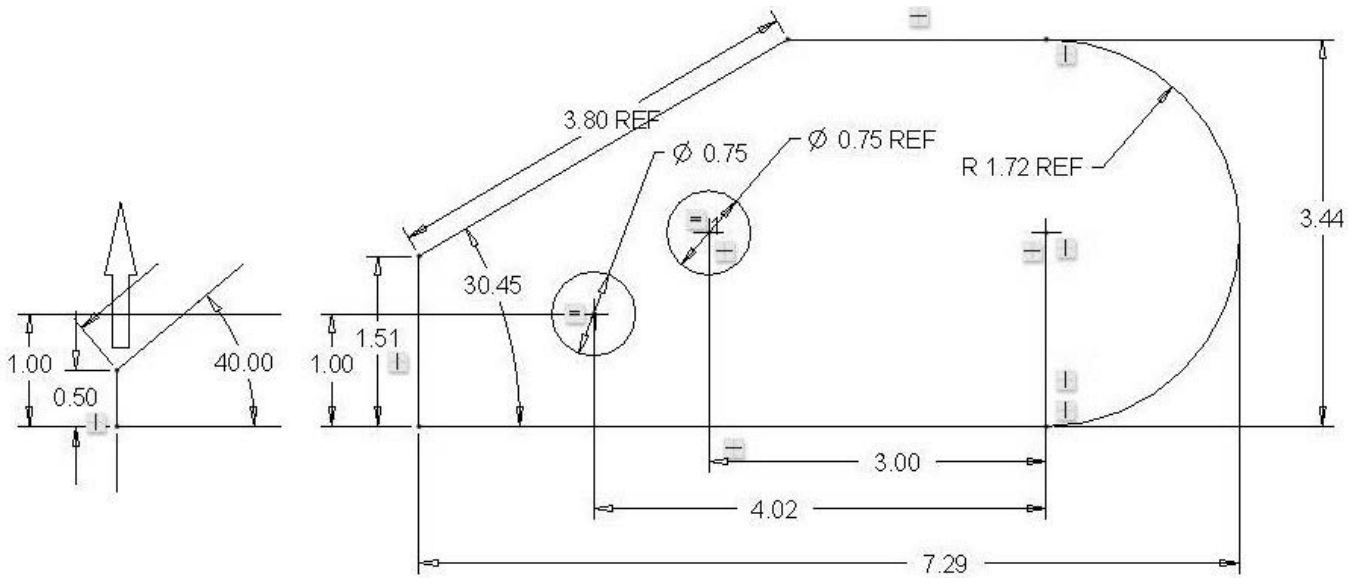



Figure 3-25 Design Intent, 0.50 Changed to 1.51 and diameter changed to 3.44

Step 10: Move the cursor onto the point where the left vertical line meets the slanted line, then *press* the LMB. The point will turn red. *Press down* the LMB, then move the cursor upward. The point will follow the cursor. Move the point to the left and right. Pick the point where the top of the arc meets the horizontal line. Move it upward as well. As you move the point observe the dimensions on the figure. Note that the left small hole and its size do not change since it references the horizontal distance to the center of the big radius and the vertical distance from the bottom of the figure. See Figure 3-25. What dimensions change or do not change, and why?

Step 11: Draw an imaginary box around the entire figure, then press the <Delete> key. What happens? (The screen should be completely empty again.)

Step 12: Use  to draw three circles somewhere on the screen as shown in Figure 3-26. Do not let the circles intersect. Do not make the circle the same diameter.

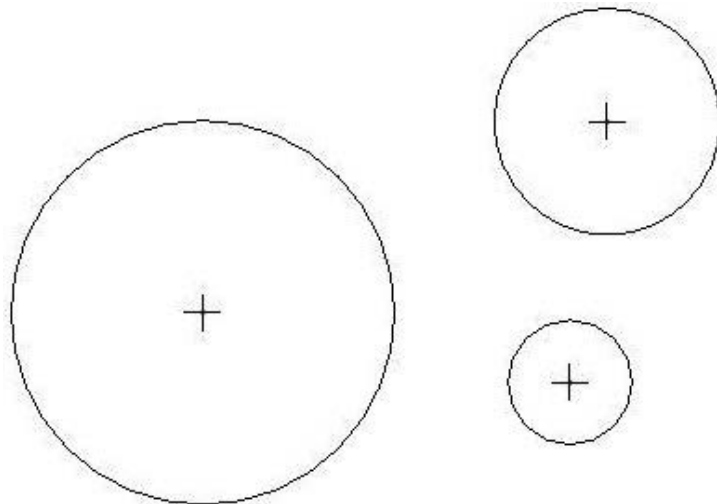





Figure 3-26 Three Circles

Step 13: Use  to draw three tangent lines as shown in Figure 3-27.

Step 14: Use the LMB to select any of the tangent points or the center of any circle. *Press down and hold* the LMB while moving the cursor.

Design Intent
<p>Note that the lines remain tangent to the circles because you specified a design intent of tangent lines. See Figure 3-28.</p>

Step 15: Use  to draw a vertical centerline to the left of the figure and a horizontal centerline above the figure.

Step 16: Use  to create a mirror image of the entire figure from step 13 or 14 above the centerline. Then mirror both figures about the vertical centerline as seen in Figure 3-29.

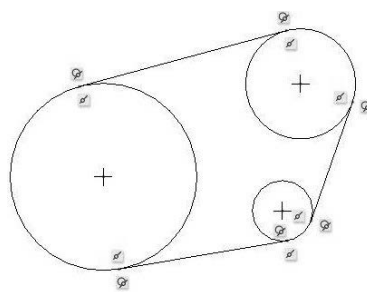


Figure 3-27 Lines Tangent to Circles

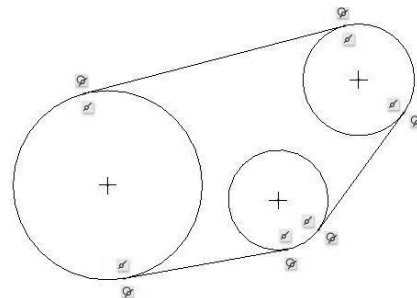


Figure 3-28 Design Intent with Tangents

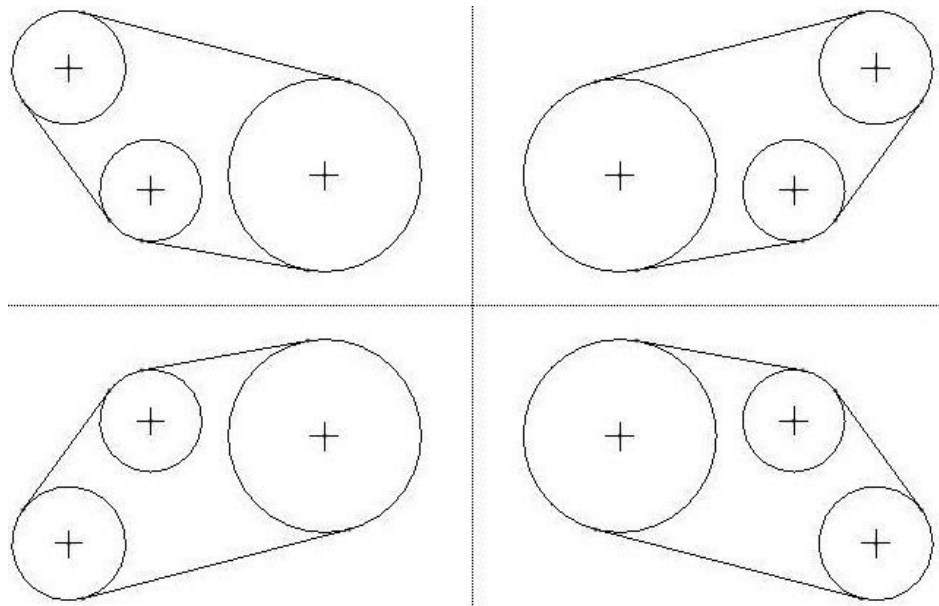






Figure 3-29 Mirror Images


Step 17: Draw an imaginary box around all the figures including the two centerlines, then press the <Delete> key to clear the screen. If the centerlines remain, *select* them individually with the LMB, then press the <Delete> key.


Step 18: Use  to draw two rectangles as shown in Figure 3-30. (R)

Step 19: Use  to draw two angled rectangles as shown in Figure 3-30. (AR)


Step 20: Use  to draw two center-located rectangles as shown in Figure 3-30. (CR)

Step 21: Use  to draw two parallelepipeds as shown in Figure 3-30. (PP)

Step 22: Use  to remove the bottom five figures or draw an imaginary box around the bottom five figures, then press the <Delete> key to clear the screen.

Step 23: Use  to draw a circle that goes through three points as seen in Figure 3-31.

Step 24: Draw an imaginary box around all of the figures, then press the <Delete> key to clear the screen.

Step 25: Draw the figure shown below on the right. Use  to create a circle that is tangent to the three circles and a circle that is tangent to the two bigger circles and the straight line segment. See Figure 3-32.

Step 26: Draw an imaginary box around all the figures, then press the <Delete> key to clear the screen.

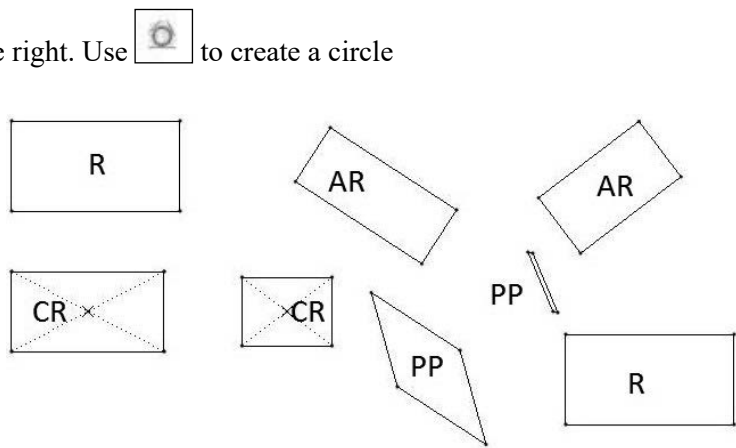


Figure 3-30 Drawing Rectangles

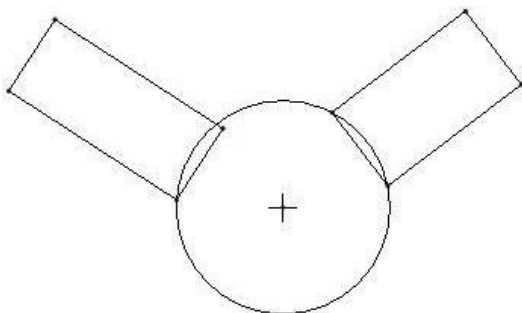


Figure 3-31 Circle through 3 Points

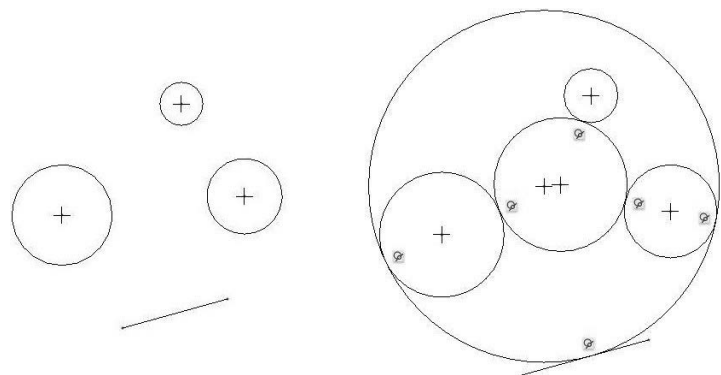






Figure 3-32 Circles Tangent to Others

Step 27: Use  to draw a circle near the middle of the screen.

Step 28: Use  to draw four concentric circles. Note that you must first select the desired circle, then create the new concentric circles. See Figure 3-33.

Step 29: Use  and  to create two ellipses similar to the ones shown in Figure 3-34.

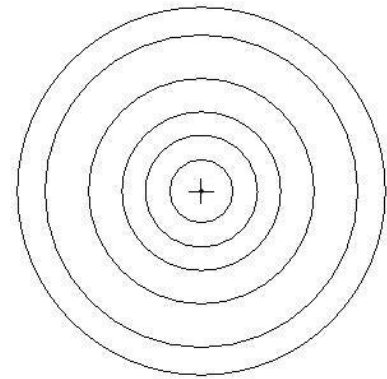


Figure 3-33 Concentric Circles

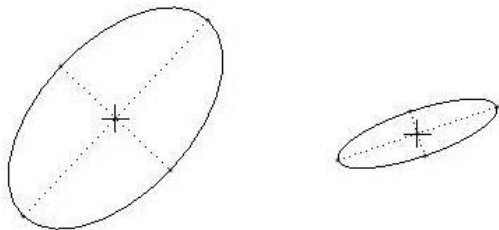






Figure 3-34 Ellipses


Step 30: Draw an imaginary box around the concentric circles and the two ellipses, then press the <Delete> key to clear the screen.

Step 31: Use  to create an arc, then use  to create four concentric arcs as shown on the left in Figure 3-35.

Step 32: Use  to create the upper and lower right most arcs in Figure 3-35.

Step 33: Use  to draw the arc that is tangent to three different arcs as shown in Figure 3-35.

Step 34: Draw an imaginary box around the arcs, then press the <Delete> key to clear the screen.

Step 35: Draw an angled rectangle. Use  to draw a conic curve on the upper left side of the rectangle. See the left side of Figure 3-36.

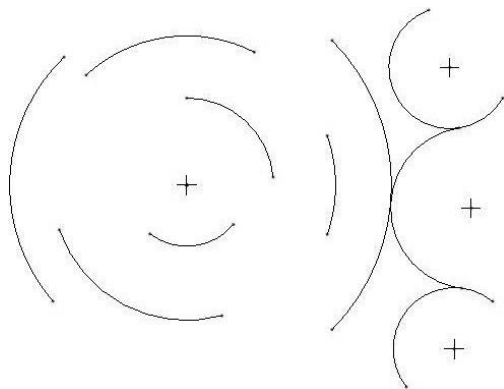


Figure 3-35 Arcs

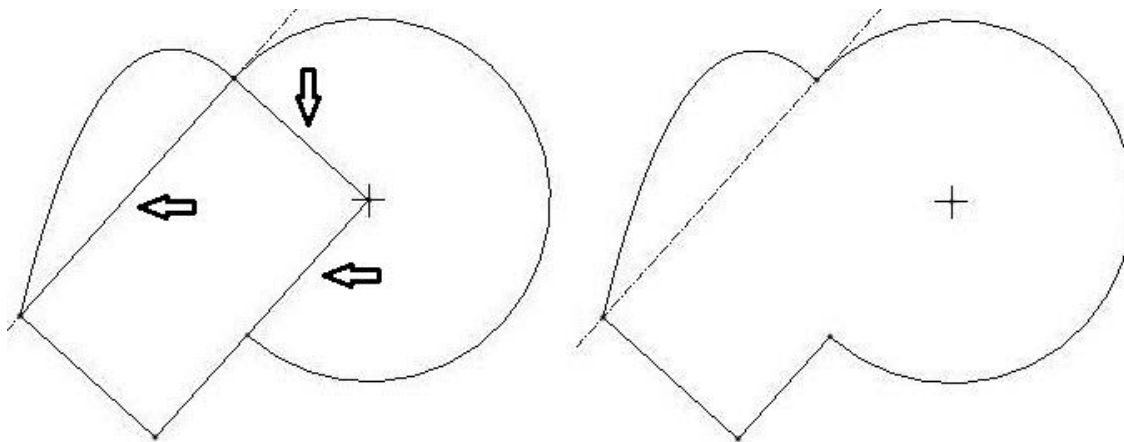







Figure 3-36 Review

Step 36: Use  to locate the center of an arc at the corner of the angled rectangle, then draw the arc shown.






Step 37: Use  to remove the interior lines. See the right side of Figure 3-36. How did you do? Review the tools if necessary.


Step 38: Add a concentric circle just inside the big arc. Use  to create an arc tangent to the concentric circle and the two straight lines. Draw a line between the inner circle and the big arc where the construction line crosses both. The dashed lines represent the object's shape before the tangent arc was added.

Step 39: Use  to divide the big arc at the point where you drew the previous short line. Use this tool to break the line where the small arc intersects the bottom line.

Step 40: Remove the unwanted lines as shown in Figure 3-37. Use the *undo* tool, , if you accidentally erase the wrong line segment.

Step 41: Figure 3-37 is a bit weird looking, but it does demonstrate the use of some of sketcher's tools. Draw an imaginary box around the figures, then press the <Delete> key to clear the screen.

Step 42: Create a rectangle, then use  to create four rounded corners. Use  to make all four rounded corners the same radius. Adjust the dimensions so the rectangle is 4.00 inches long and 2.00 inches high with 0.50-inch radius corners. There should be only three dimensions present on the sketch. See Figure 3-38. Remember that    stand for horizontal, vertical, and tangent.

Step 43: Use  to draw a vertical construction centerline near the middle of the rectangle. Add three lines that represent the cutout, then remove the horizontal line at the top of the cutout.

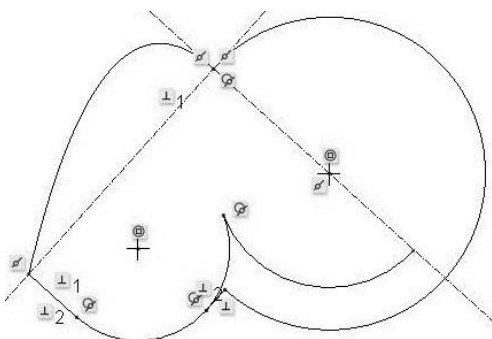


Figure 3-37 Tangent and Divide

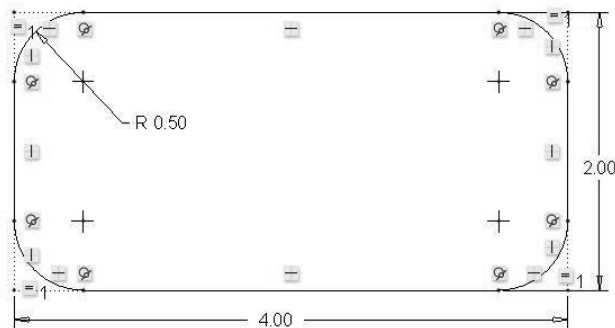


Figure 3-38 Rounded Corners

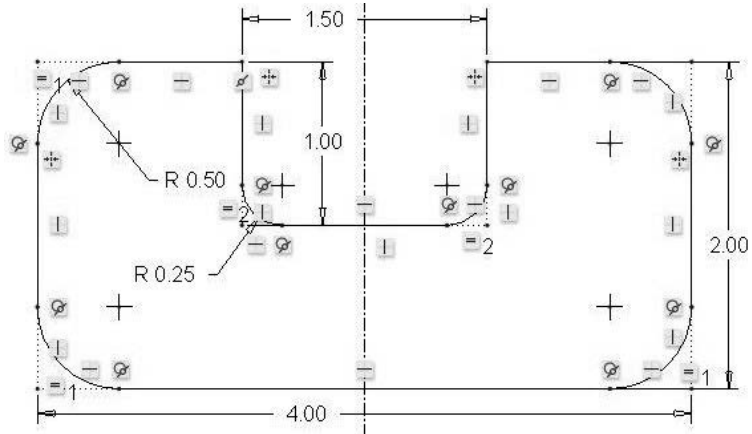






Figure 3-39 Symmetry and Rounds

Step 44: Use  to make the cutout symmetric about the vertical centerline. Use it again to make the rectangle symmetric about the centerline. Make the cutout 1.50 inches wide and 1.00 inches deep. The  symbols are positioned on the sketch to show two points that are symmetric about the centerline.

Step 45: Use  to round the inside corners of the cutout to a radius of 0.25 inches (Figure 3-39). There should be only three additional dimensions on the sketch, 1.50, 1.00, and 0.25. Note that this tool leaves a dot where the corner used to be.

Step 46: Determine the circumference of this figure using . Remember to *pick* on **line segment** with the LMB, then hold down <CTRL> and pick the rest of the segments around the perimeter of the figure. *Pick OK.* *Pick* the 4.00-inch dimension to be the adjustable dimension. You should get a perimeter of 12.93 inches. What happens to the overall length of the figure if the perimeter is changed to 15 inches? Overall length becomes 5.03 inches. Can you verify this? See Figure 3-40.

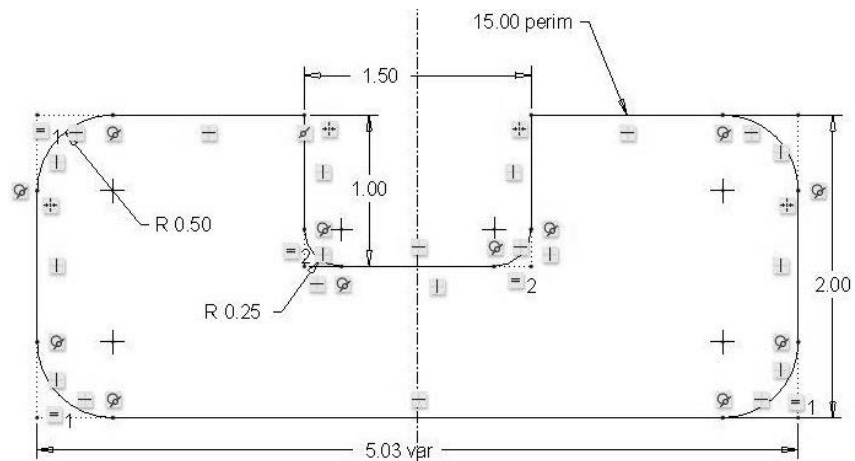



Figure 3-40 Perimeter Dimension Added

Step 47: Draw an imaginary box around the entire figure, then select . Note that you cannot modify the overall length of the figure since its value is driven by the perimeter dimension. Pick the red X to cancel the request without making any changes.

Step 48: At this point we have covered most of the commands found in sketcher. Use your knowledge to draw and dimension the following figures, Figures 3-41, Figure 3-42, and Figure 3-44.

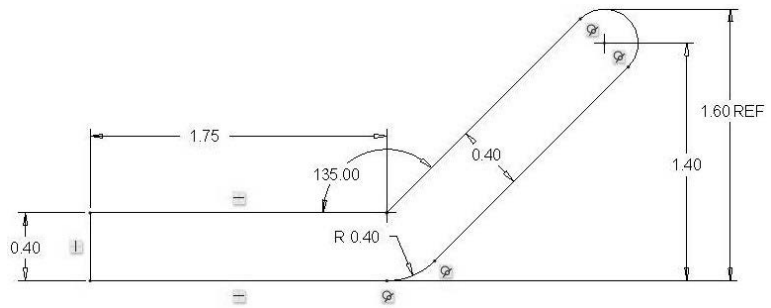


Figure 3-41 Control Bracket

Now, practice on your own without detailed instructions to see how you do. Refer back to the needed instruction if you forgot how to implement it.

Step 49: Draw control bracket side view (Figure 3-41).

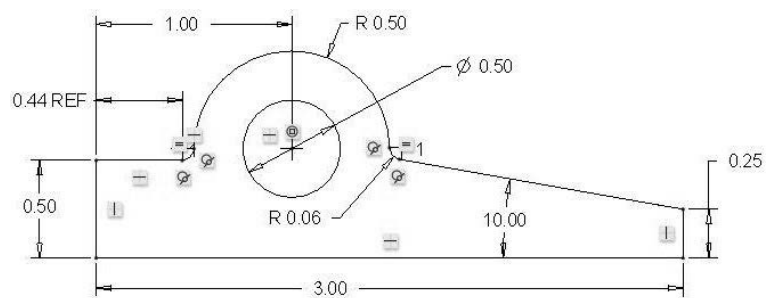


Figure 3-42 Angle Bearing

Step 50: Draw angle bearing side view (Figure 3-42).

Step 51: Draw sheet metal pattern (Figure 3-43). Note that there are only five dimensions on this sketch. If you fold the sheet metal pattern at the edges, can you guess what it forms? (Answer: Right prism.)

The phantom lines shown below at the folds were created using the LINE tool, then *selecting* the line, holding down RMB until **Menu** pops up, *selecting Properties* from the pop-up menu, changing SOLIDFONT to PHANTOMFONT, *picking Apply*, then *picking Close*. Is it easier to determine what shape is created with the phantom lines added? See Figure 3-36.

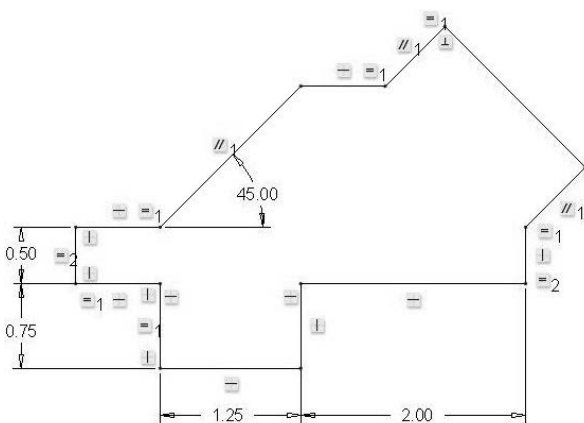


Figure 3-43 Sheet Metal Pattern

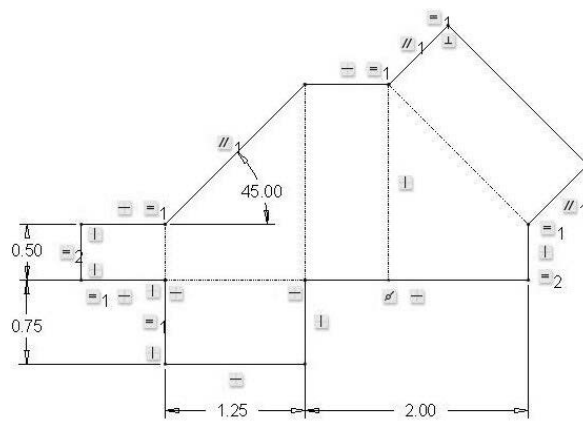


Figure 3-44 Pattern with Phantom Lines

Sketcher Exercise

Design Intent

A pipe needs to be pushed up and over to the right using a simple, inexpensive jig. There is a small protruding tab mounted to a base plate just to the left of the pipe. After brainstorming and analysis, it is decided that a simple tapered block will solve the problem. The slanted face of the block will push the pipe over and up as dictated by the problem statement. If the block is sized correctly, then it can be held in place by the protruding tab. See the layout below, Figure 3-45.

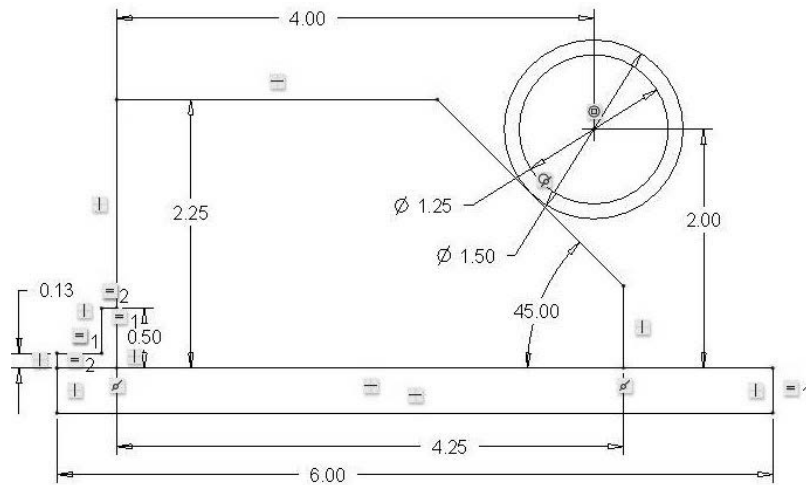


Figure 3-45 Layout 1

Design the tapered block (shown above) by properly sizing it according to the data given in Table 3-1.

Dimension	Design_1a Size (inches)	Design_1b Size (inches)	Design_1c Size (mm)
Pipe_Locator	4.00	5.00	100
Pipe_Height	2.00	2.50	50
OD	1.50	2.00	42
ID	1.25	1.50	32
Push_Angle	45	60	45
Height	2.25	2.50	60
Base	?	?	?
Top	?	?	?

Let's use Creo Parametric's sketcher to draw the sketch above to scale, and then determine the appropriate sizes for the tapered block.

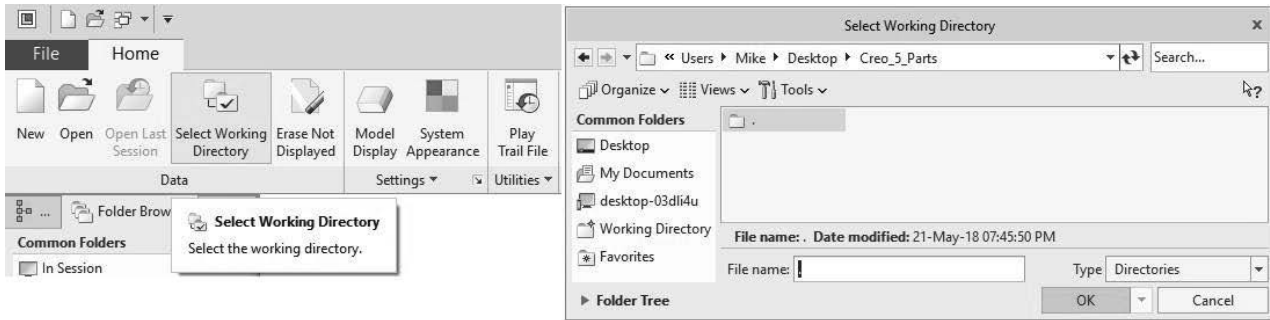


Figure 3-46 Set Working Directory

Using Creo Parametric to Sketch the Jig, Design_1a

- Step 1:** Start Creo Parametric by *double-clicking* with the LMB on the **Creo Parametric** icon on the desktop, or from the Program list: Creo Parametric.
- Step 2:** Set your working directory by *selecting* the **Select Working Directory** icon in the Home ribbon. Locate your working directory, and then *pick* **OK**. See Figure 3-46.
- Step 3:** Create a new sketch by *picking* the **New** icon at the top of the screen (Figure 3-47), or by *selecting* **File>New**.
- Step 4:** *Select* **Sketch** and name the sketch, “exercise1a”. *Pick* **OK**. See Figure 3-48.

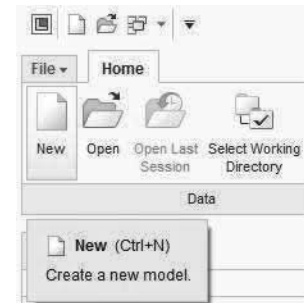





Figure 3-47 New Sketcher File

- Step 5:**  *Select* the **Rectangle** tool, and then draw a rectangle as shown below by *selecting* the location of two of its opposite diagonal corners using the LMB. Don’t worry about its size right now. See Figure 3-49. The  icon lets you know this line is horizontal. The  indicates that the line is vertical.

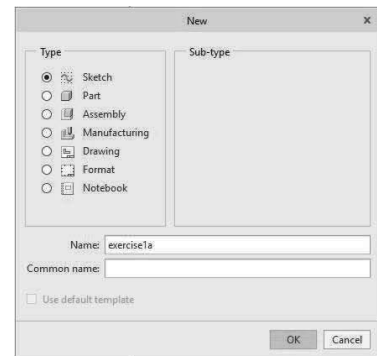





Figure 3-48 Name Sketcher File

- Step 6:**  *Select* the **Line** tool and draw the “L-shaped” tab on the left end of the base. Place the cursor near the upper left corner of the base. Creo Parametric will use the same point as previously defined when an X and a circle appear over the top of the point as shown in Figure 3-50. Draw a short vertical line (left end of L shape). When drawing the upper horizontal line and the left vertical line of the tab, extend the lines until the  symbol appears. When drawing the top horizontal line of the tab, again extend the line until the  symbol appears.

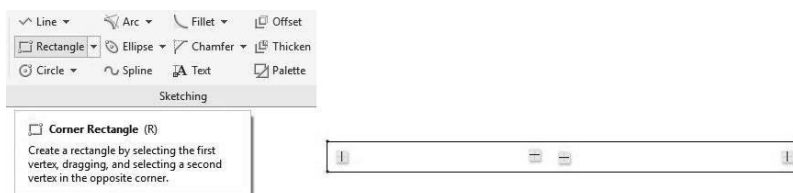


Figure 3-49 Rectangle



Figure 3-50 Same Point

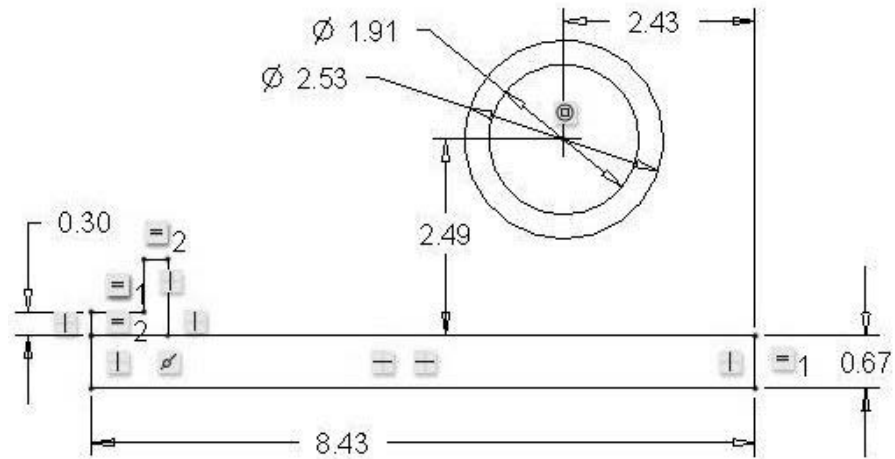



Figure 3-51 Initial Layout

- Step 7:**  Select the **Center and Point Circle** tool. Move the cursor to the approximate center of the circles, then *push* the LMB. Move the cursor away from this point, thus increasing the size of the circle. When the circle appears approximately the correct size, *push* the LMB again. Draw two concentric circles near the right end of the base rectangle as shown in Figure 3-51. Don't worry about the circles' diameters.
- Step 8:** Your sketch should look similar to the one pictured in Figure 3-51, although the dimension locations and values may be different. The two short lines should be labeled with =_2 . The two lines of the angle tab and the thickness of the plate should be labeled with =_1 . The = subscripts, 1 and 2, may be different.
- Step 9:** Because the dimensions shown above do not match our design intent, we will use the **Normal dimension** tool to add the appropriate dimensions. Note that as you add dimensions, others disappear. Creo Parametric will show only enough dimensions to completely define the shape. Dim dimensions are referred to as weak dimensions. Normal text dimensions are referred to as strong dimensions. All dimensions should be strong before you complete any drawing or sketch.



To place a dimension between two points, *pick* the first point with the LMB, *pick* the second point with the LMB, then move the cursor to where you want the dimension to show up and press the MMB. After placing the dimension, type the desired value in the box and press the <Enter> key.

To dimension a line length, *pick* the line with the LMB, then move the cursor to where you want the dimension to show up and press the MMB. After placing the dimension, type the desired value in the box and press the <Enter> key.

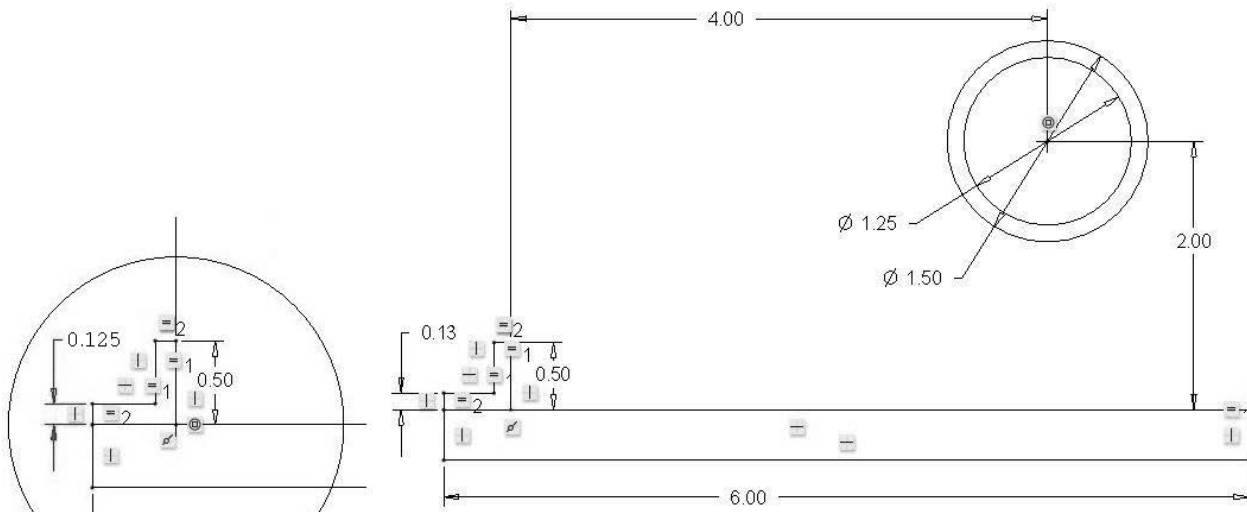


Figure 3-52 Reflects Design Intent

Modify your sketch so that it matches the one shown in Figure 3-52. Note that 0.125 rounds to 0.13 for 2-place decimals. To change a dimension's value, *double-click* using LMB, then type its new value followed by the <Enter> key. To move a dimension, *select* it with the LMB, then *press and hold* the LMB while moving the cursor. The dimension will follow. *Release* the LMB when the dimension has been properly placed.

Step 10: If any 's are missing from your sketch, they can be added using the **constraint** tools found on top of the screen (Figure 3-53). If there are extra L's or other symbols, they can be erased by *selecting* them with the LMB, then pressing the <Delete> key.

Step 11: Use the **Line** tool to sketch in the slanted-surface jig. Start at the base of the tab, move upward a couple of inches, then horizontally toward the two circles. Do not draw horizontally through the center of the circles. Stop before reaching the circles. Draw a slanted line that touches the outer circle tangent (Figure 3-54), then draw a vertical line back down to the base.

Step 12: Use the **dimension** tool if necessary to dimension the height and base of the jig. Set the height to 2.25 inches.

Step 13: Use the **dimension** tool to define the angle between the base of the jig and the slanted edge. Set this value to 45 degrees. To dimension an angle, *pick* the slanted line with the LMB, *pick* the base line with the LMB, then move the cursor between the two and press the MMB to locate the dimension. Type 45 <Enter> to set the angle to 45 degrees.

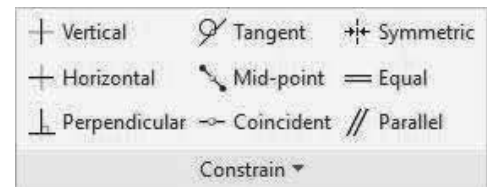


Figure 3-53 Constraint Tools

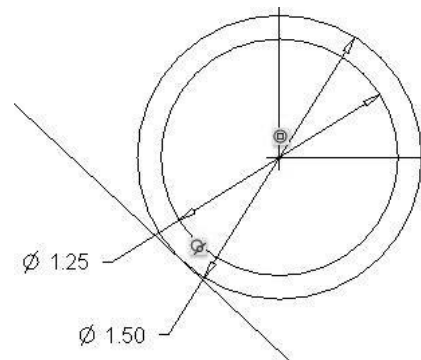


Figure 3-54 Slanted Surface Touches Outer Circle

Step 14: The base dimension can be any value; however, a value near 4.25 inches looks reasonable. See Figure 3-55. To change a dimension, *double-click* LMB on it, then type in the new value followed by the <Enter> key. Will a base size of 4.5, 4.0, or 3.25 work?

Step 15: It may appear that all necessary dimensions are shown for the jig; however, this is not true. The slanted side of the jig is controlled by the line being tangent to the outer diameter of the pipe. Without the pipe present this slanted line could move. We need to add another dimension in this sketch to completely define the jig. *Select* the **Dimension** tool again.

Step 16: *Select* the horizontal line at the top of the jig with the LMB. Move upward a bit, then press the MMB to locate the dimension. Note that a “Resolve Sketch” window appears. See Figure 3-56. This appears because the sketch as defined is completely defined and we are trying to add another dimension. One constraint and 4 dimensions must be modified. A reference dimension is a dimension shown on a sketch that is not necessary to define the sketch. When the window appears, be sure the new dimension is highlighted, then *pick* the **Dim>Ref** button. *Pick* **OK**.

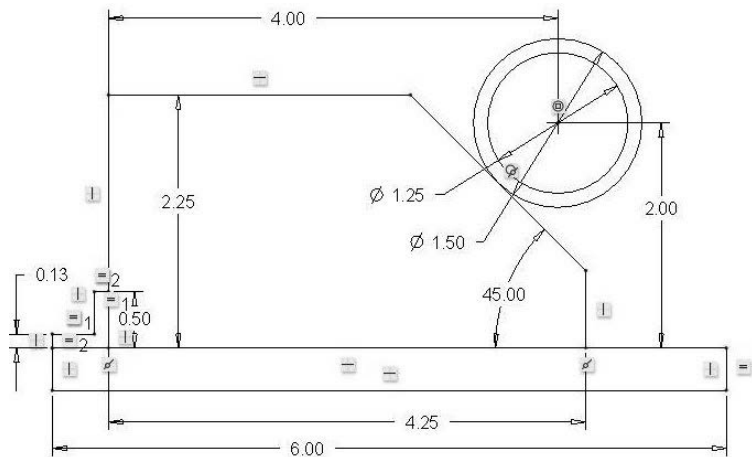


Figure 3-55 Sizing the Block

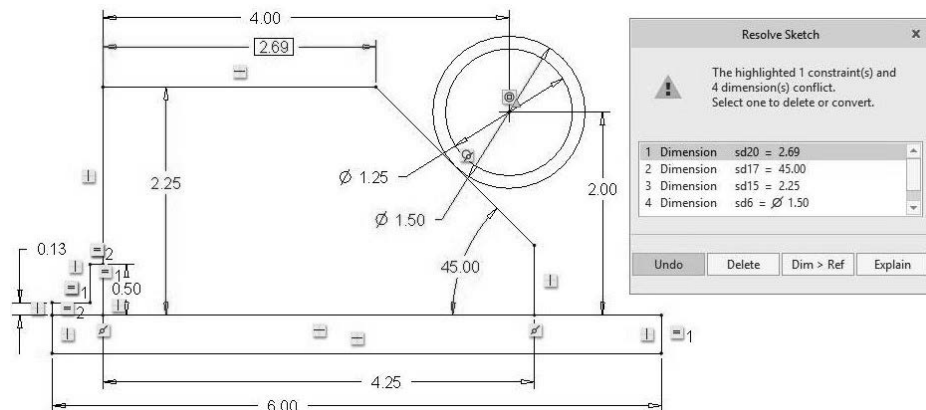


Figure 3-56 Resolve Sketch

Step 17: The letters REF will appear after the reference dimension. At this point your sketch should look similar to the one shown in Figure 3-57. The top horizontal line of the jig needs to be 2.69 inches. This will cause the slanted side of the jig to just touch the outside diameter of the pipe.

Step 18: Save your sketch by *picking* the **Save** icon at the very top of the screen, or by *selecting* **File>Save** from the menu. *Pick OK.*

Step 19: What if the pipe had an outside diameter of 1.75 inches, then the horizontal top surface of the jig would be _____ inches. (Answer: Top = 2.51 inches.)

Step 20: Set the pipe’s outside diameter back to 1.50 inches. What if the height of the jig was 2.00 inches, then the horizontal top surface of the jig would be _____ inches. (Answer: Top = 2.94 inches.)

Step 21: With the jig height at 2.00 inches, the base of the jig could be reduced to _____ inches so that the pipe hits near the middle of the slanted surface. (Answer: Base = 4.00 inches.)

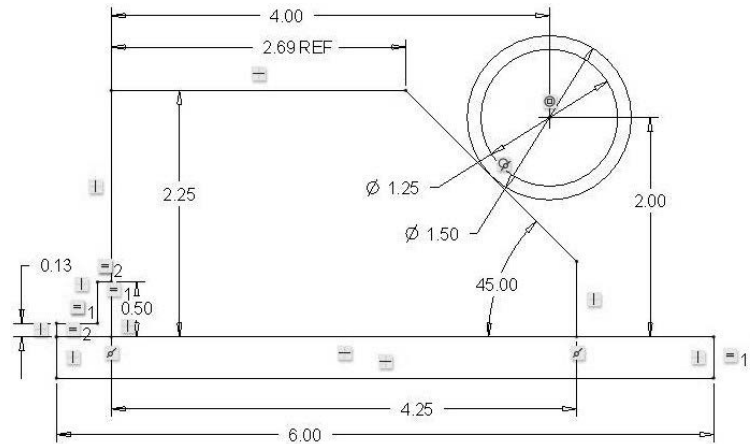


Figure 3-57 Reference Dimension Added

Do you see some of the power of parametric dimensioning? You can quickly see the effects of changing a dimension or parameter after the initial design is set up.

Using Creo Parametric to Design a Jig, Design_1b

Step 22: With sketch “exercise1a” visible on the screen, *select* **File>Save As>Save a Copy**. See Figure 3-58.

Step 23: In the New Name area, type “exercise1b”, then *pick* **OK**. Do not change the model name. See Figure 3-59.

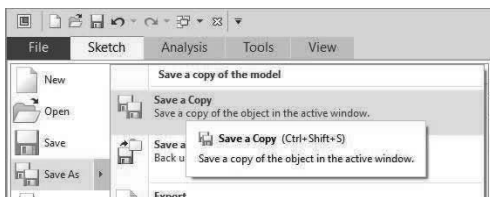


Figure 3-58 Save a Copy

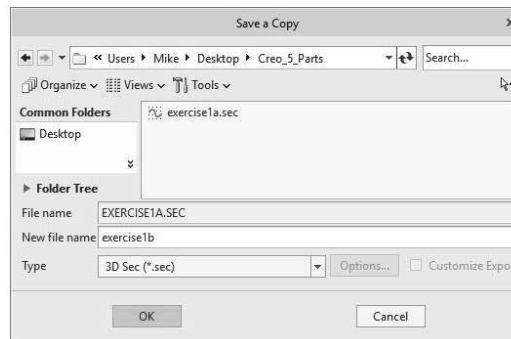


Figure 3-59 Save As “exercise1b (*.sec)”

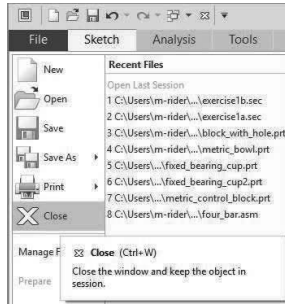


Figure 3-60 Close Window

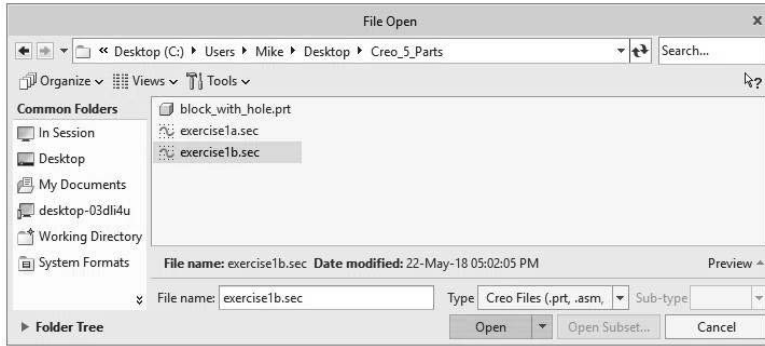


Figure 3-61 Open “exercise1b.sec”

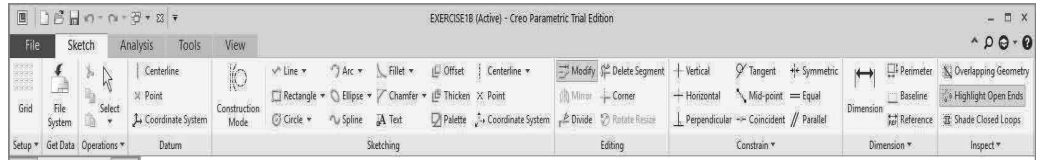


Figure 3-62 Filename at Top of Screen

Step 24: **File>Close** to close the window and clear the screen (Figure 3-60).

Step 25: **File>Open** or pick the *Open* icon.

Step 26: Locate “exercise1b.sec” in your working directory. *Pick* the **Open** button as shown in Figure 3-61.

Note that “EXERCISE1B” appears at the top of the Creo Parametric window (Figure 3-62).

Step 27: Modify the dimensions according to Design_1b’s parameters shown in Table 3-2 and Figure 3-63. To modify a dimension *double-click* on its value using the LMB, and then type a new value followed by the <Enter> key.

Table 3-2 Design 1b’s Data	
Dimension	Design_1b Size (inches)
Pipe_Locator	5.00
Pipe_Height	2.50
OD	2.00
ID	1.50
Push_Angle	60
Height	2.50
Base	?
Top	?

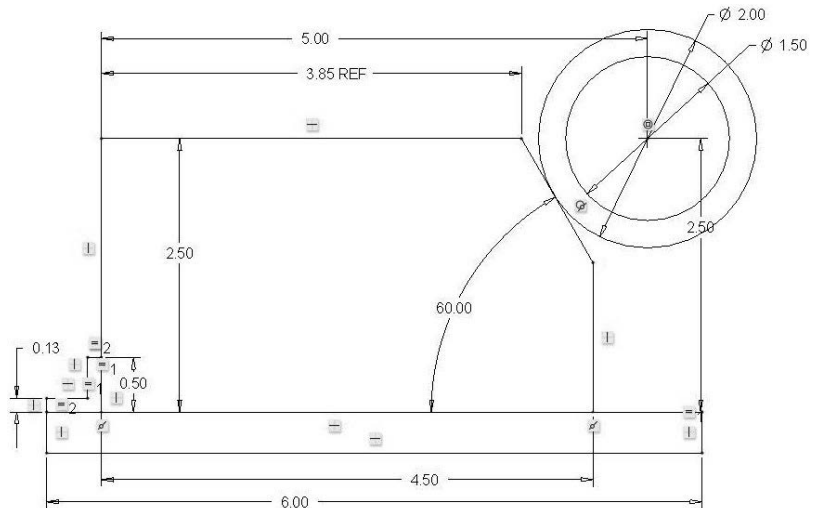


Figure 3-63 Jig Design 1b

Step 28: Within a few minutes the six defined parameters should be changed. Setting the jig's base to 4.50 inches allows the pipe to touch near the center of the slanted surface. The jig's new top dimension is _____ inches. (Answer: Top = 3.85 inches.)

The power of parametric dimensioning is the ability to change dimensional values without redrawing the object.

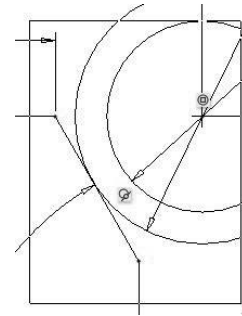




Figure 3-64 Zoom In

Step 29:  Select the **Zoom In** icon from the graphics tool bar. Draw an imaginary box around the area where the outer diameter of the pipe touches the slanted surface using the LMB. See Figure 3-64. Press the MMB to exit from this mode.

Step 30:  Select the **centerline** tool. Using the LMB pick the center of the pipe, then the point where the line touches the jig's slanted surface. The normal force produced by the pipe pushing on the slanted surface will lie along this line. If this line crosses the left edge of the jig above the fixed tab, then the jig will not stay in place by itself. In this case, this line crosses below the fixed tab, thus this jig will stay in place when released. See Figure 3-65.

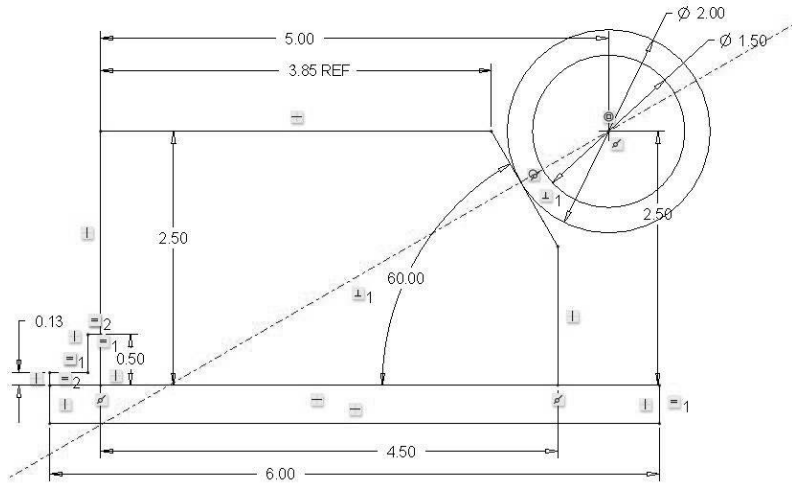


Figure 3-65 Line of Action for Pipe Force

Step 31: Pick this centerline with the LMB, and then press the <Delete> key to remove it from the sketch.

Step 32: File>Save or 

Step 33: File>Manage File>Delete Old Versions. See Figure 3-66.

Step 34: Pick **Yes** to remove all previously saved versions of this file in your working directory (Figure 3-67).

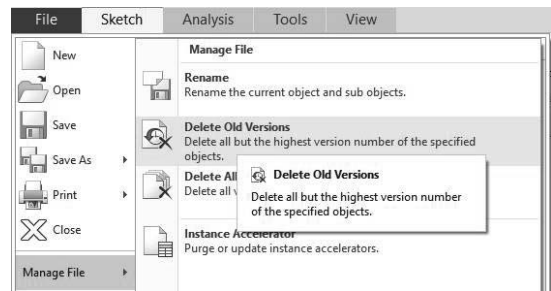


Figure 3-66 Delete Old Versions

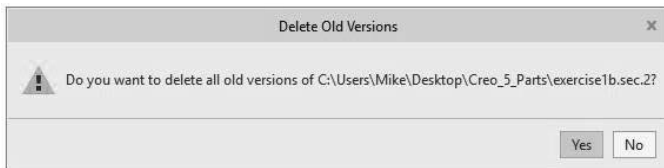



Figure 3-67 Accept Command Actions

Using Creo Parametric to Design a Jig, Design_1c

Step 35: With sketch “exercise1b” visible on the screen, *select File>Save As> Save a Copy.*

Step 36: In the New Name area, type “exercise1c”, then *pick OK.* Do not change the model name.

Step 37: **File>Close** to close the window and clear the screen or *pick* 

Step 38: **File>Open** or *pick* the *Open* icon.

Step 39: Locate “exercise1c.sec” in your working directory. *Pick* the **Open** button. Note that “EXERCISE1C” appears at the top of the Creo Parametric window.

Step 40: Draw an imaginary box around the entire sketched jig including all dimensions. *Pick* the first corner with the LMB, then *press and hold* the LMB while moving to the opposite corner of the figure. *Release* the LMB.

Step 41:  *Select* the **Modify** tool.

Step 42: Check the “Lock Scale” box, then type a value of 100 where there was previously a value of 5.00. See Figure 3-68. *Pick OK* to update the entire sketch. (Be sure to check the Lock Scale box before changing the dimension to 100.)

Step 43: Change the tab’s height dimension from 10 to 12, and the tab thickness dimension from 2.5 to 3. Change the rectangular base’s length from 120 to 140. See Figure 3-69.

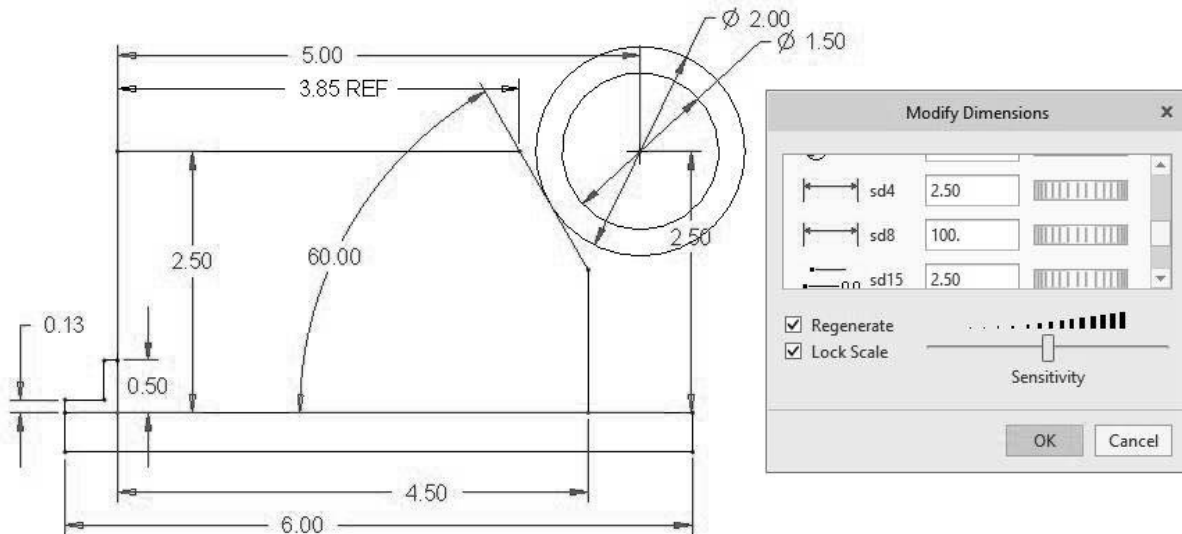
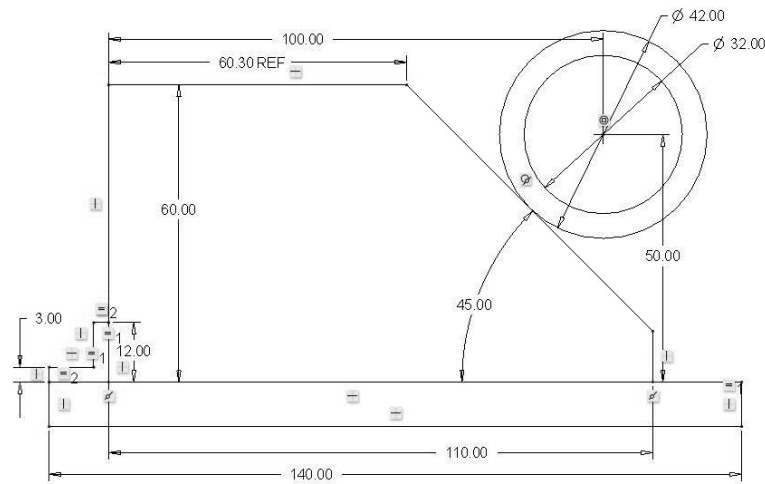


Figure 3-68 Scale All Dimensions



Dimension	Design_1c Size (mm)
Pipe_Locator	100
Pipe_Height	50
OD	42
ID	32
Push_Angle	45
Height	60
Base	?
Top	?

Figure 3-69 Jig Design 1c

Step 44: Continue to modify the dimensions according to Design_1c's parameters shown in Table 3-3. To modify a dimension *double-click* on its value using the LMB, and then type a new value followed by the <Enter> key.



Step 45: Within a few minutes the six defined parameters should be changed. Setting the jig's base to 110 millimeters allows the pipe to touch near the center of the slanted surface. The jig's new top dimension is _____ mm. (Answer: Top = 60.3 mm.) See Figure 3-69.

The power of parametric dimensioning is the ability to change dimensional values without redrawing the object.

Step 46: File>Save.

Step 47: File>Manage File>Delete Old Versions. *Pick Yes* to remove all previously saved versions of this file in your working directory.




Step 48: File>Close.

Step 49: If you are done at this time, then exit from Creo Parametric by picking the  in the upper right corner of the window. When you move the cursor over the icon, it changes to . *Pick Yes* to exit.



End of Sketcher Exercise.

Review Questions

1. What is the main function of the left mouse button (LMB) in sketcher?
2. What are the main functions of the middle mouse button (MMB) in sketcher?
3. What are the main functions of the right mouse button (RMB) in sketcher?
4. What might happen to your sketch if you delete a sketcher constraint such as    ?
5. What is the difference between a light blue dimension and a blue dimension?
6. How do you create an explicit dimension for a line?
7. How do you convert a weak dimension into a strong dimension?
8. How do you create a diameter dimension?
9. How can you verify that a given section is closed?
10. Can you trim a construction line?
11. What is a reference dimension?
12. Why do weak dimensions show up on your sketch after creating a series of constructions?
13. How can you find where two lines overlap?
14. How do you scale all dimensions in a sketch at one time?
15. How do you change the length of a line?
16. How do you change the diameter of a circle?
17. How do you force a line to be tangent to a circle?
18. How do you find the name of a sketcher variable?
19. How do you get rid of a weak dimension placed on the sketch by the intent manager?
20. How do you zoom in on a selected area of a sketch?
21. How do you zoom out?
22. If you are sketching on the FRONT datum plane and the sketch gets rotated in 3D, how do you reposition the sketch so the FRONT datum plane is parallel with the screen again?
23. Should you set your sketcher constraints or your sketcher dimensions up first? Why?
24. How do you create a symmetrical sketch?
25. What is the main advantage of parametric design?

Sketcher Problems

3.1 Draw "shear_plate" (Figure 3-70). **Design Intent**—The left hole must be directly $15/16$ inches above the lower left corner at point A. The right hole must be 2.875 inches from the left hole. The slot must be oriented 30 degrees from the indicated centerline. What is the length of line KA, height at point L, and the angle at K? (>File>Options>Sketcher, change to 3 decimal places. Pick OK.)

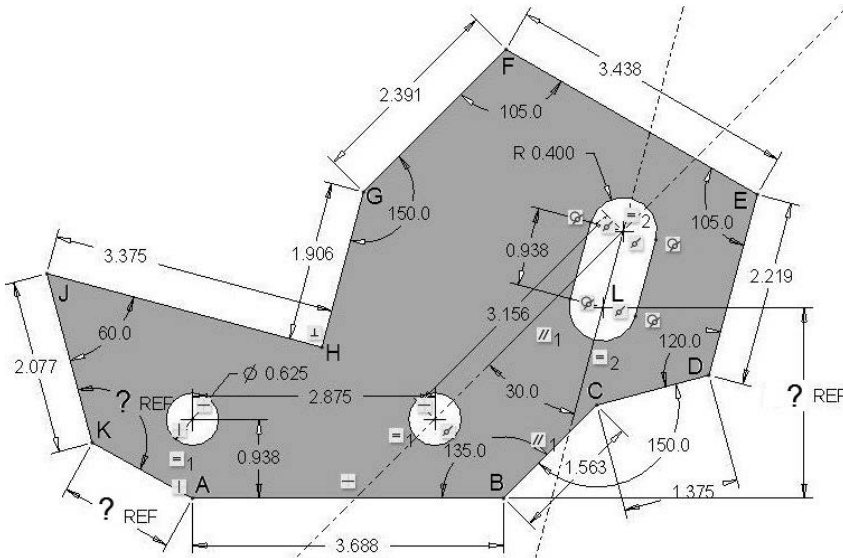


Figure 3-70 Problem 1—Shear Plate

3.2 Draw "latch_plate" (Figure 3-71). **Design Intent**—The plate must be symmetrical about a horizontal centerline (draw 1st). The four holes must be at the corners of a 3.000 inch by 3.500 inch imaginary box. Note that the beginning of the 15° line is directly below or above the center of the circles of radius R1. What is the length of the far right vertical line L_2 and the horizontal line L_1 ? Note that 2.093 must be a reference dimension if the sketch is drawn correctly. (>File>Options>Sketcher, change to 3 decimal places before sketching. Pick OK.)

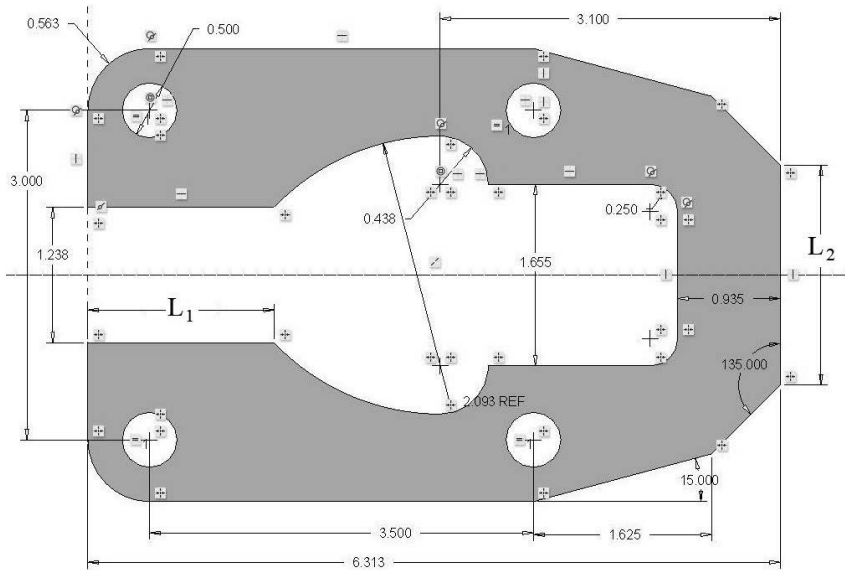


Figure 3-71 Problem 2—Latch Plate

3.3 Draw the front view of "special_cam" (Figure 3-72). **Design Intent**—The cam must be a symmetrical ellipse (draw centerlines). The hexagonal shaped hole must be 1.500 inches from the geometric center of the cam. The cutout and small hole are for mass reduction only. Mating arcs are tangent. What is the maximum height of the circular arc cutout? The 3.625 radius arcs are centered at the top and bottom of the ellipse.

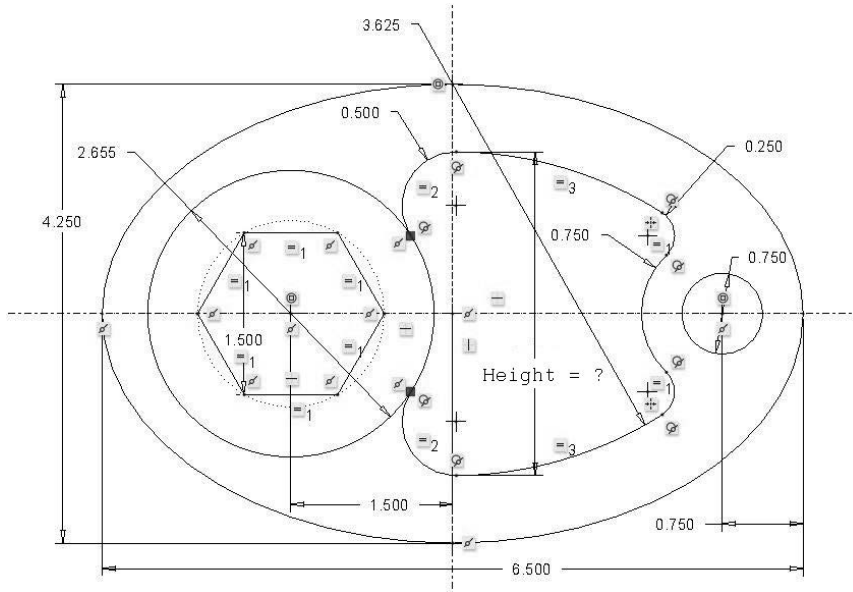


Figure 3-72 Problem 3—Special Cam

3.4 Draw the front view of "rocker_arm" (Figure 3-73). **Design Intent**—The hexagonal shaped hole of the rocker must be offset by 15 degrees. The right 0.65-inch diameter hole must be 3.00 inches from the center hex hole. The left 0.65-inch diameter hole must be 5.62 inches from the right hole. What is the overall size of rocker arm?

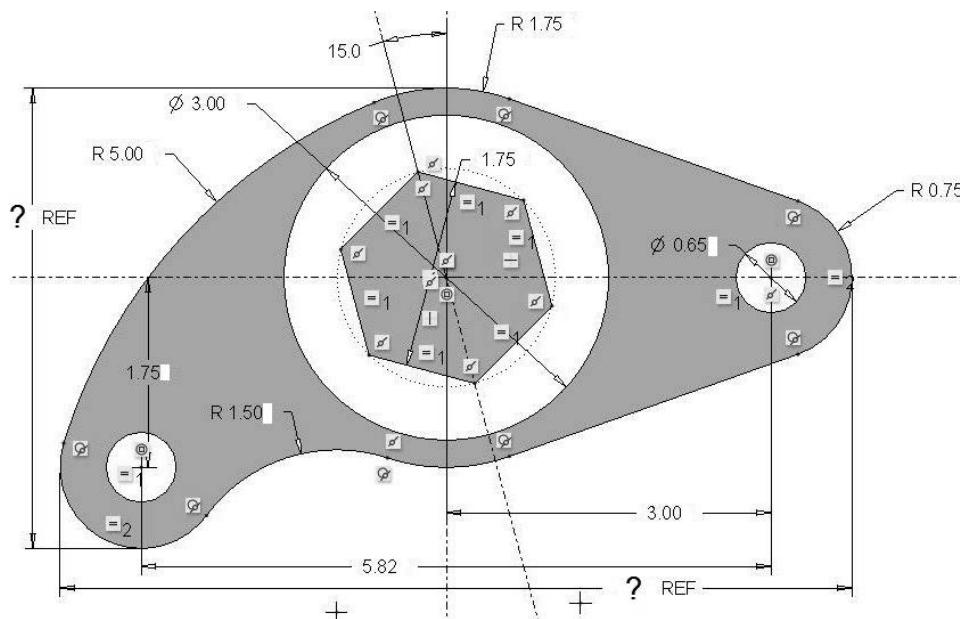


Figure 3-73 Problem 4—Rocker Arm

- 3.5 Draw the front view of the "rocker" (Figure 3-74). **Design Intent**—The rocker is symmetric about the center hole. The right 1.00-inch diameter hole must be 4.00 inches right of the center hole and 1.25 inches below it. The left 1.00-inch diameter hole must be 4.00 inches left of the center hole and 1.25 inches above it. The arcs that form the perimeter are tangent to each other. **Hint:** draw circles tangent to each other, then trim them to form the arcs.

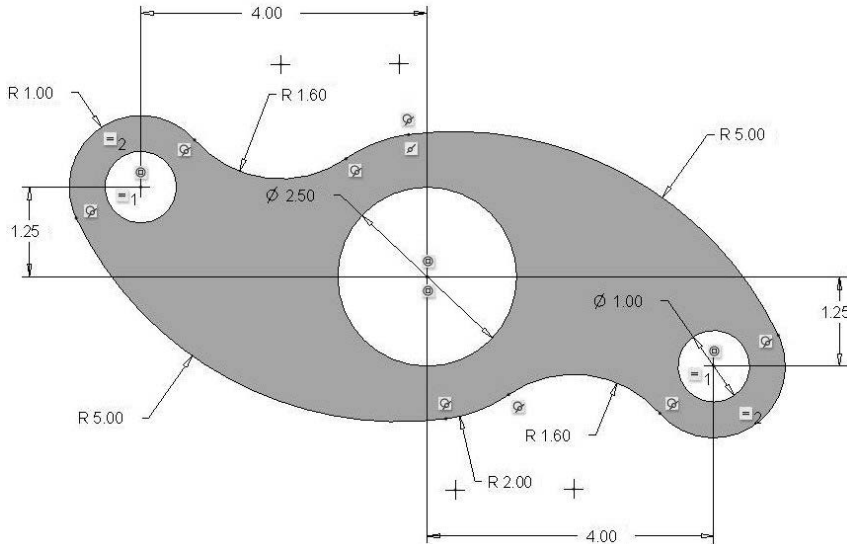


Figure 3-74 Problem 5—Rocker

- 3.6 Draw the front view of "cover_plate" (Figure 3-75). **Design Intent**—The plate is symmetric about its center in both directions. Why does the 3.00 dimension have the letters REF after it?

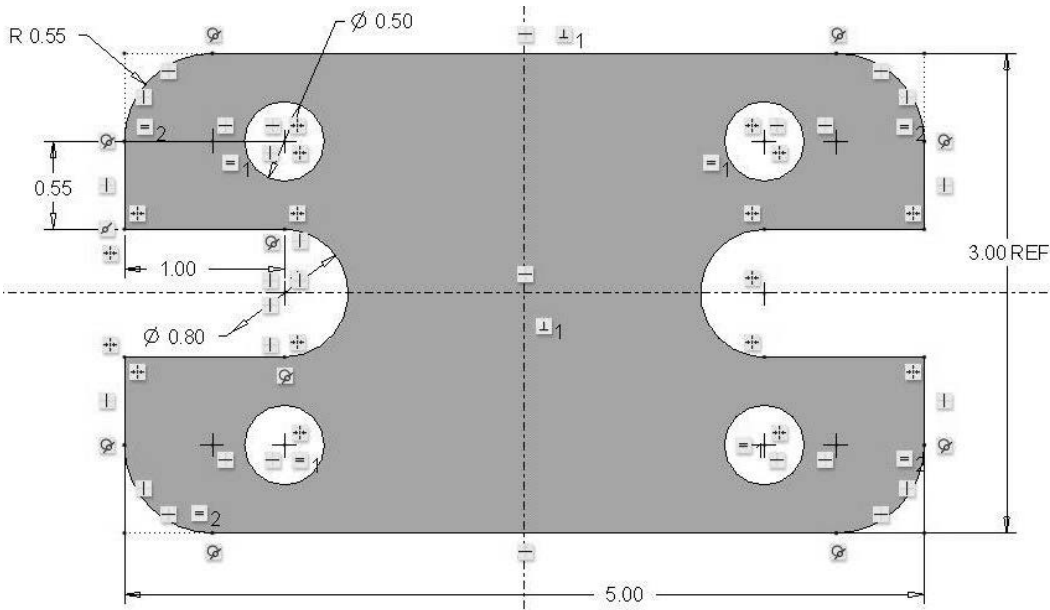


Figure 3-75 Problem 6—Cover Plate

3.7 Draw the "front_cover_plate" (Figure 3-76). **Design Intent**—All features are located from the outside edge of the cover plate, and not relative to each other.

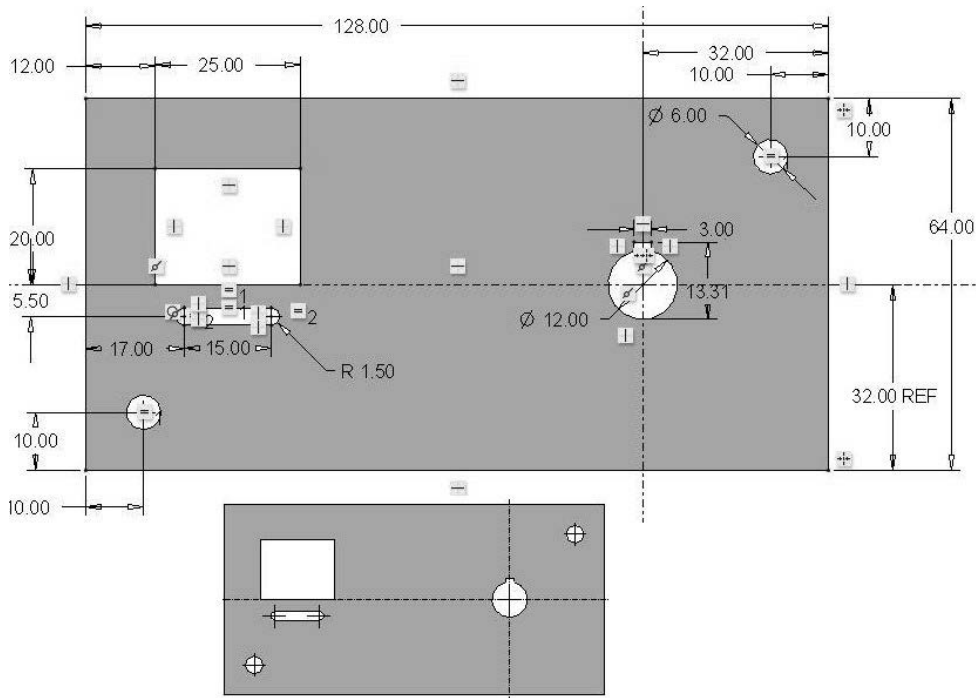


Figure 3-76 Problem 7—Front Cover Plate

3.8 Draw the "support_frame" (Figure 3-77). **Design Intent**—The thickness of the frame is $\frac{3}{4}$ of an inch. What is the volume of the shaded area if the support frame was 10 inches deep? Can you determine the volume underneath the support frame (white space under) if it is 10 inches deep?

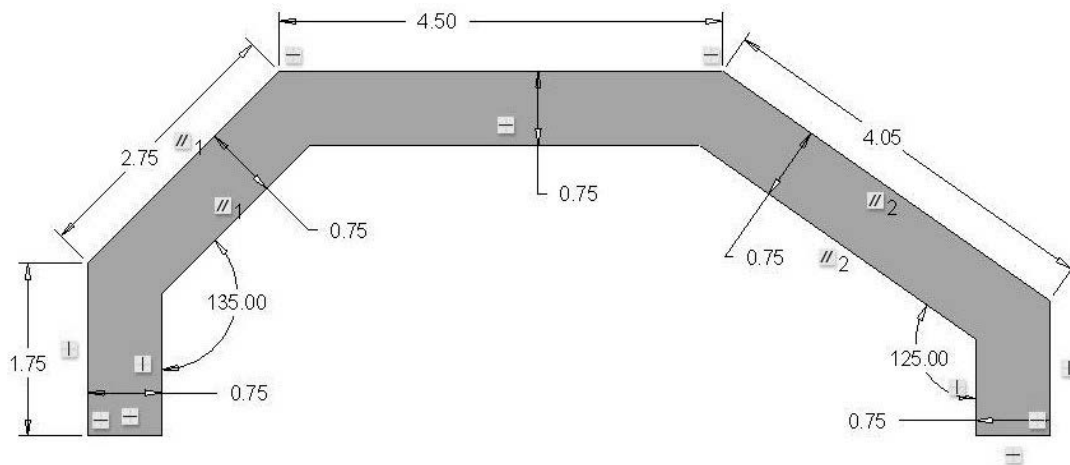


Figure 3-77 Problem 8—Support Frame

3.9 Draw the "adjustable_sector" (Figure 3-78). **Design Intent**—The two holes need to be 4.00 inches apart. The angular slot needs to be 25 degrees clockwise from vertical to 10 degrees counterclockwise from horizontal. To reduce stresses round all sharp corners using a 1/4-inch round or fillet. (Filletlets and rounds are not shown in the figure.)

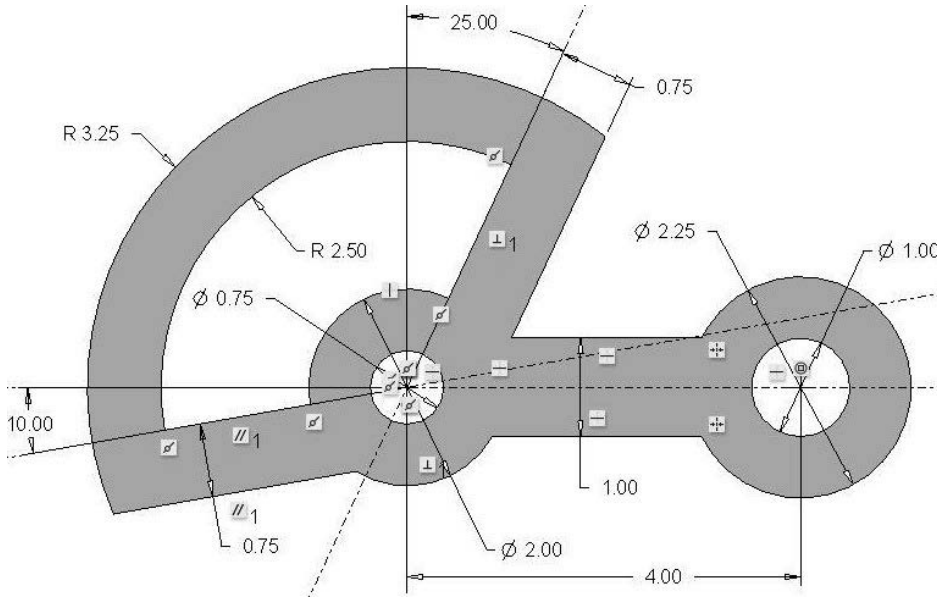


Figure 3-78 Problem 9—Adjustable Sector

3.10 Draw the "cork_gasket" (Figure 3-79). **Design Intent**—The three holes need to be on a 5.50-inch bolt circle spaced 120° apart. All boundary arcs are tangent to each other. (Note (=3) on numerous arcs equal 2.75 inches, or half of the 5.50 bolt circle.)

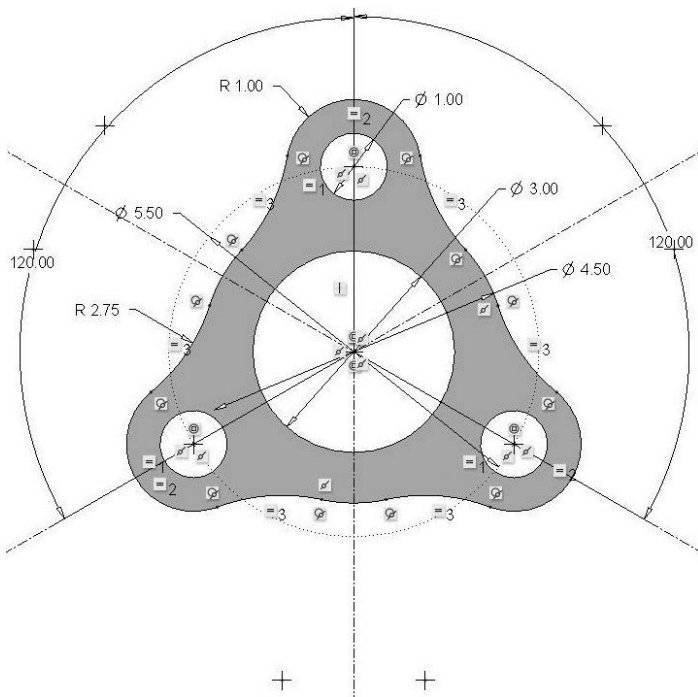


Figure 3-79 Problem 10—Cork Gasket

3.11 Challenge problem - Draw the "Special_S-wrench" (Figure 3-80). **Design Intent** — All arcs are tangent to each other and the edges of the two end shapes. The diamond shape is 1.30 inches per side. The outer pentagon fits inside a circle of 1.75 inches in diameter. The inner pentagon uses a 1.20-inch diameter circle. The 3-inch radius centerline is tangent at both ends and at its midpoint. The tangent points for the outer arcs are directly above and below the tangent point for the centerline arc.

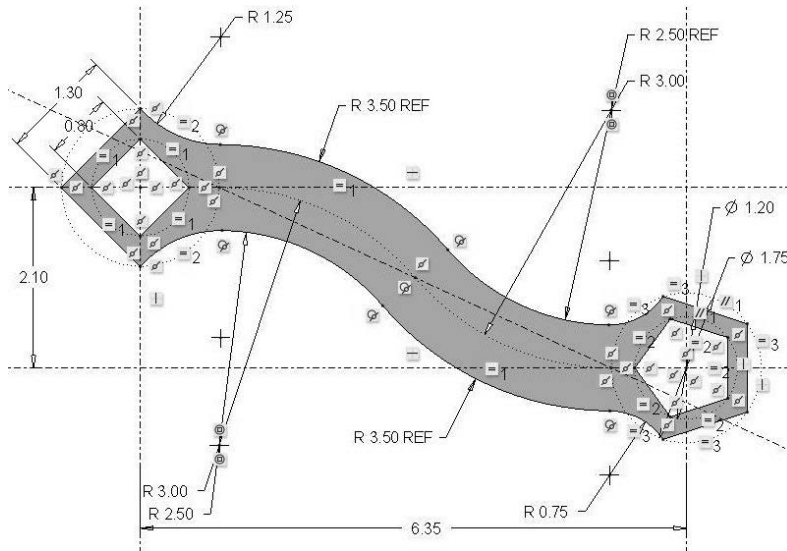


Figure 3-80 Problem 11—Special S Wrench

3.12 Challenge problem - Draw the "Racket_Pawl" (Figure 3-81). **Design Intent**—All arcs are tangent to each other or the straight edges. The R 12.00 arcs are centered along the horizontal lines.

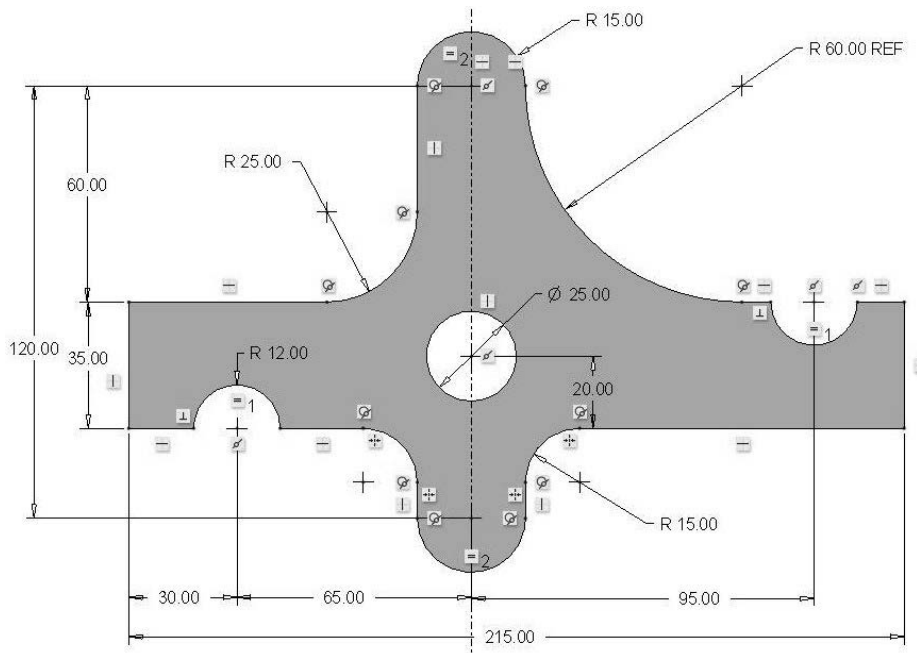


Figure 3-81 Problem 12—Racket Pawl