

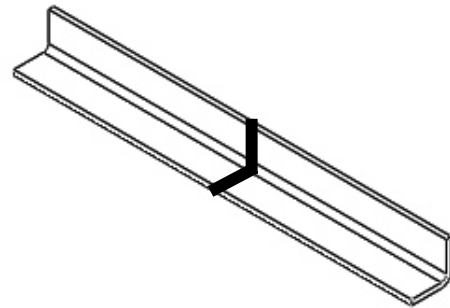
ANGLE-13HOLE Part

The ANGLE-13HOLE part is an L-shaped support bracket. The ANGLE-13HOLE part is manufactured from 0.090"[2.3] aluminum.

The ANGLE-13HOLE part contains fillets, holes and slot cuts.

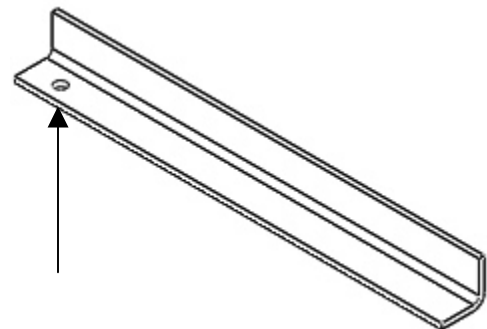
Simplify the overall design into seven features. Utilize symmetry and Linear Patterns.

The open L-Shaped profile is sketched on the Right plane.

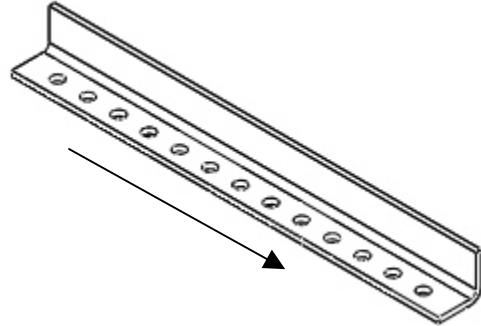


Utilize an Extruded Thin feature with the Mid Plane option to locate the part symmetrical to the Right plane.

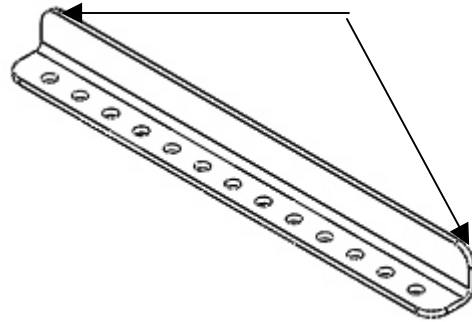
Insert the first Extruded Cut feature for the first hole. The first hole is sketched on the top face of the Extruded Thin feature.



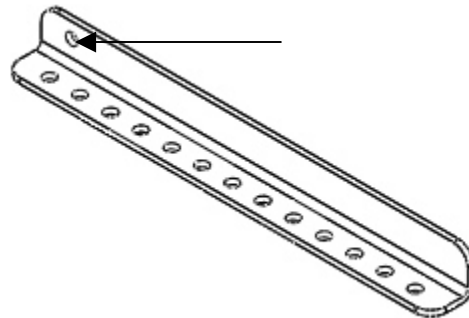
Insert a Linear Pattern to create an array of 13 holes along the bottom horizontal edge.



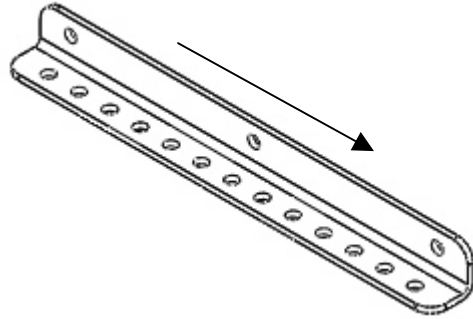
Insert a Fillet feature to round the four corners.



Insert the second Extruded Cut feature on the front face of the Extruded Thin feature.

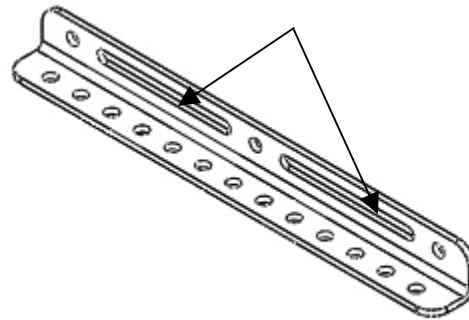


Insert a Linear Pattern to create an array of 3 holes along the top horizontal edge.




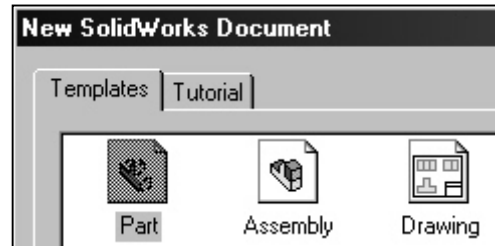
Utilize the Sketch Mirror tool to create the slot profile.

Insert the third Extruded Cut feature to create the two slots.

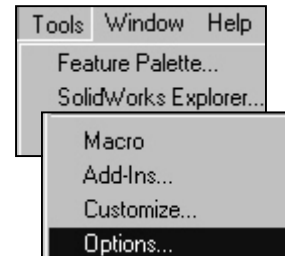


Create a New Part.

- 1) Click **New**  from the Standard toolbar. The Templates tab is the default tab. Part is the default template from the New SolidWorks Document dialog box.
- 2) Click **OK**.

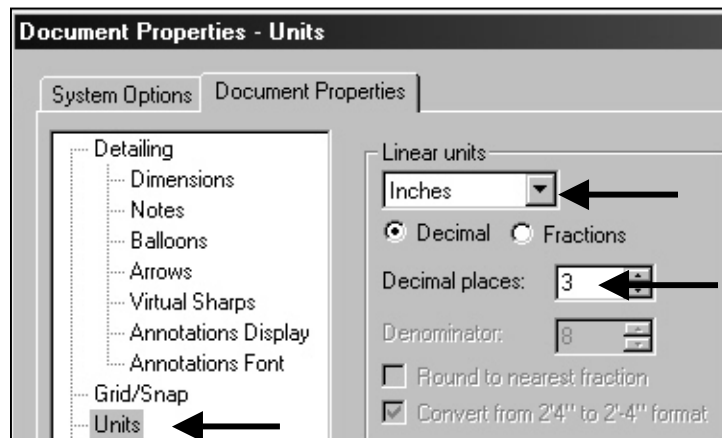


- 3) Set the Dimensioning Standard and Part Units. Click **Tools** from the Main menu. Click **Options**.



- 4) Click the **Document Properties Tab**.
- 5) Select **ANSI** from the Dimensioning Standard list box.

- 6) Click **Units**.
- 7) Select **Inches** [Millimeters] for Linear units.
- 8) Select **3** [2] for Decimal places.
- 9) Click **OK** to set Document units.



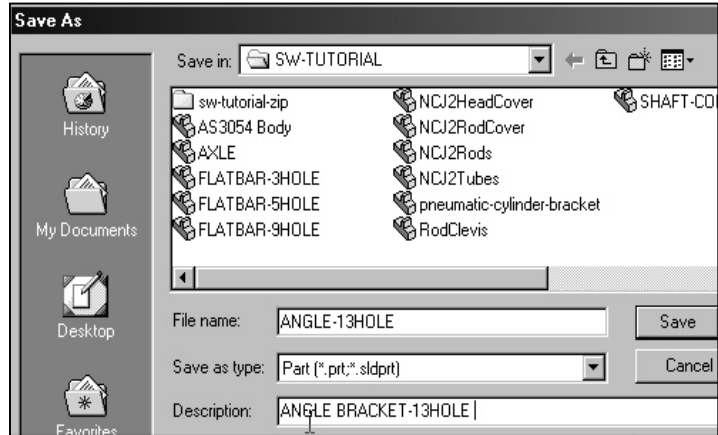
10) Save the Part.
Click **File, Save As**.

11) Select **SW-TUTORIAL** for the Save in file folder.

12) Enter **ANGLE-13HOLE** for File name.

13) Enter **ANGLE BRACKET-13 HOLE** for Description.

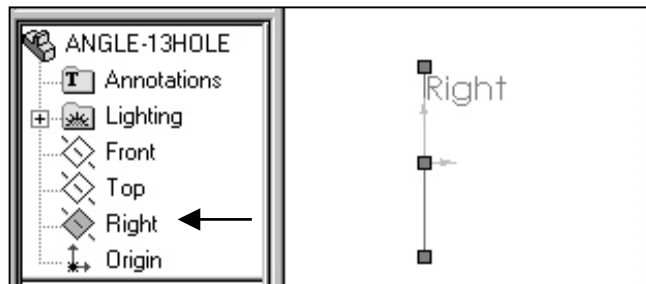
14) Click the **Save** button.



Insert an Extruded Thin feature sketched on the Right plane.

Select the Sketch plane.


15) Click **Right** from the Feature Manager.

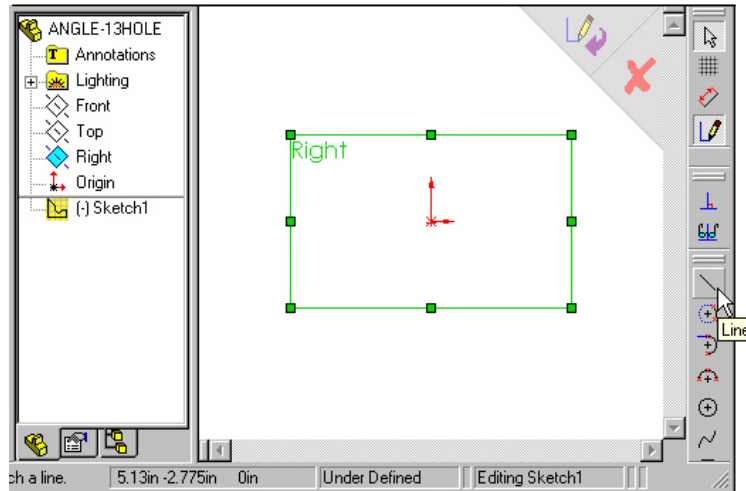



The Right plane is displayed as a vertical line in the Front view. The Right plane name icon in the FeatureManager is displayed in blue.

Insert a new sketch.

16) Click Sketch  to insert a sketch on the Right plane.

17) Click Right  from the Standards view toolbar. The Right plane green boundary is displayed in the Right view.



18) Click Line  from the Sketch Tools toolbar. The cursor displays the Line feedback symbol

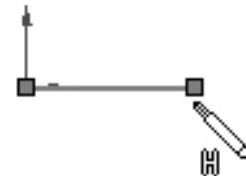


19) Sketch a horizontal line. Click the **Origin**



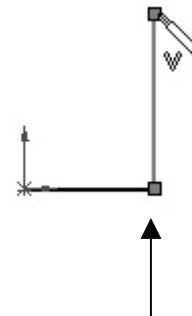
Click a **position** to the right of the Origin. The

cursor displays the Horizontal feedback symbol



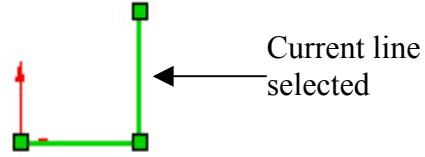
20) Sketch a vertical line. Click a **position** directly above the right end point. The cursor displays the Line feedback

symbol



Add Relations.

21) Click **Add Relations**

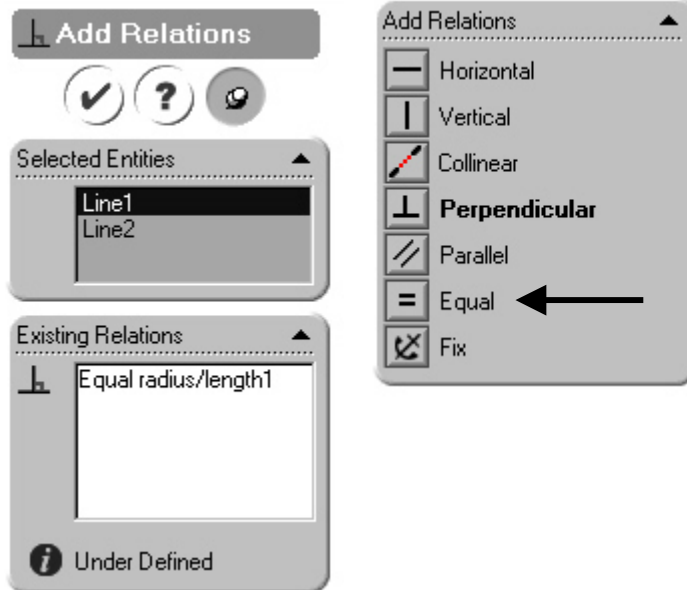


The vertical line, Line2 is displayed in the Select Entities box.

22) Click the **horizontal line**, Line1.

23) Click the **Equal** button.

24) Click **OK** to add an Equal relation.



Add Dimensions.

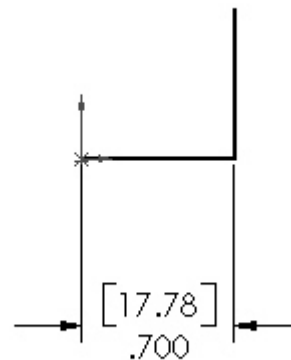
25) Click **Dimension**

26) Click the **horizontal line**.

27) Click a **position** below the profile.


28) Enter **.700 [17.78]** in the Modify dialog box.

29) Click the **Check Mark** . The black sketch is fully defined.



Extrude the sketch.

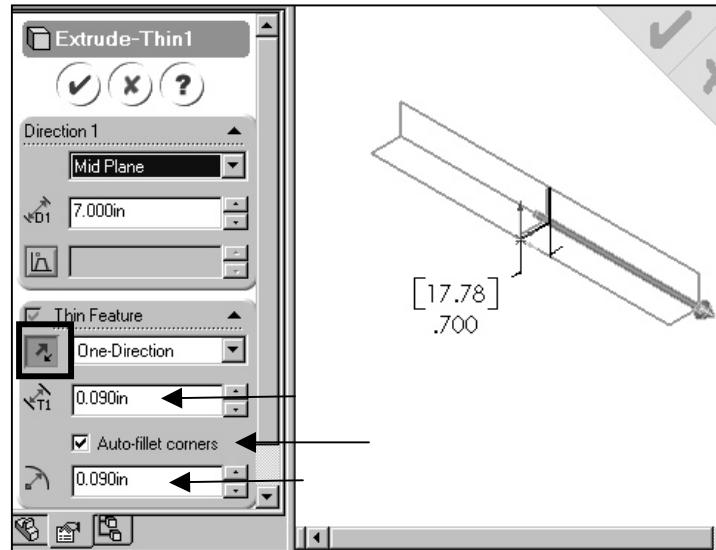
30) Click **Extruded**

Boss/Base 
from the Features
toolbar.

31) Select **Mid Plane**
for Direction1 End
Condition.

32) Enter **7.000**
[177.80] for
Depth. The Thin
Feature box is
displayed.


33) Click the **Reverse
Direction Arrow**
button for One-
Direction.
Material thickness
is created above
the Origin.



34) Enter **.090** [2.3]
for Thickness.

35) Check **Auto-fillet corners**.

36) Enter **.090** [2.3] for Fillet Radius.

37) Click **OK**  to insert an Extrude-Thin feature.

38) Fit the model to the Graphics window. Press the **f** key.

39) The Extrude-Thin1 feature is displayed.

Modify feature dimensions.

40) Double-click **Extrude-Thin1** in the Graphics window.

41) Double-click **7.000 [177.80]** dimension.

42) Enter **6.500 [165.10]** in the Modify dialog box.

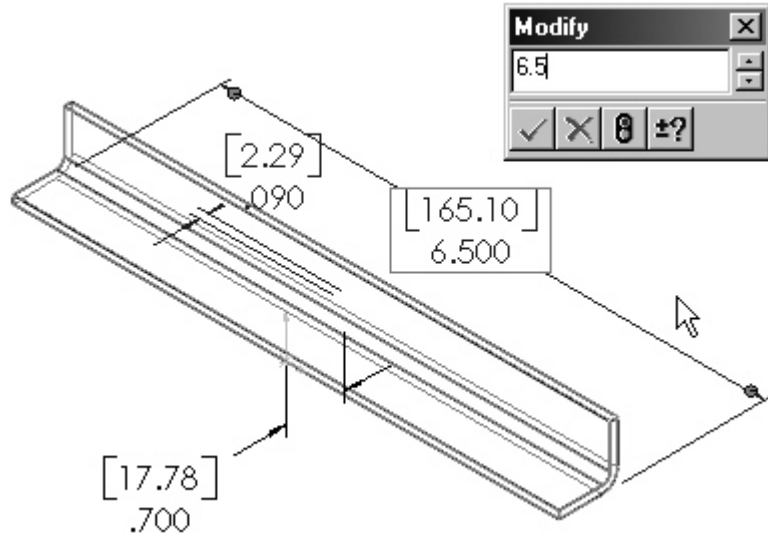
43) Click the **Check Mark**



44) Click **Rebuild**



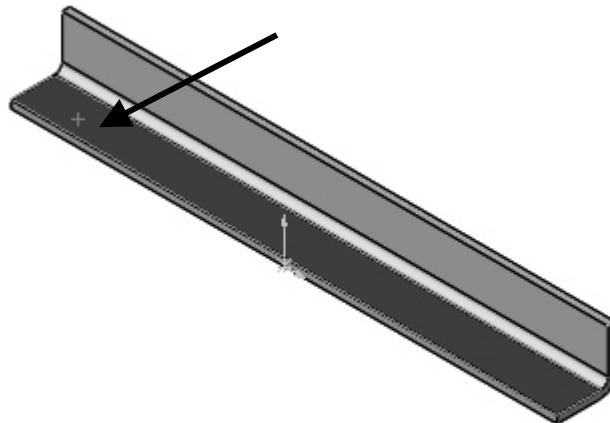
from the Main toolbar to update the feature dimensions.



Insert a new sketch for the Extruded Cut.

45) Select the sketch plane. Click the **top face** of the Extrude Thin1. The cursor displays the Face

feedback symbol.



46) Click **Sketch**



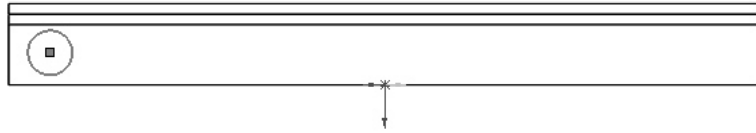
47) Click **Top**



from the Standards view toolbar.

48) Click Circle

 from the Sketch Tools toolbar.



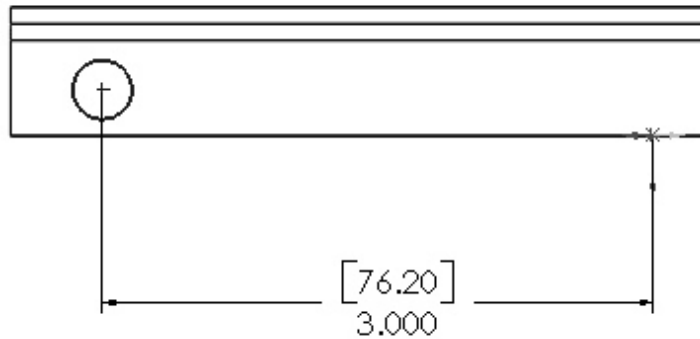
49) Sketch a circle on the left side of the Origin.

Add dimensions.
Create a horizontal dimension.

50) Click the Origin.

51) Click the center point of the circle.

52) Click a position below the horizontal profile line.



53) Enter 3.000[76.20].

54) Create a vertical dimension.
Click the **bottom horizontal line**.

55) Click the center point.

56) Click a position to the left of the profile.

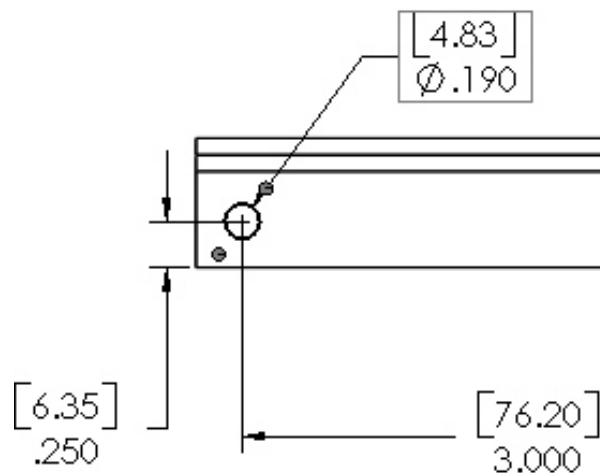
57) Enter .250 [6.35].

58) Create a diameter dimension. No number

59) Click the diameter of the circle.

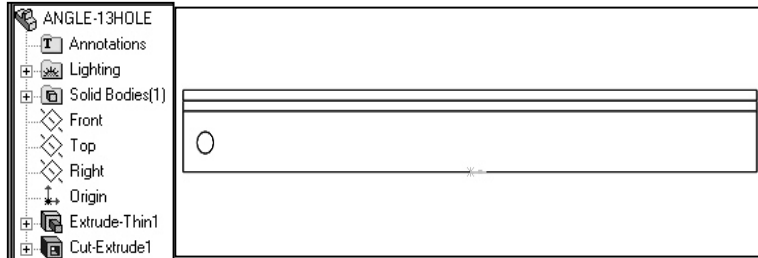
60) Click a position above the profile.

61) Enter .190 [4.83].



Insert an Extruded Cut.


- 62)** Click **Extruded Cut**  from the Features toolbar.



- 63)** Select **Through All** for Direction1 End Condition.

- 64)** Click **OK**  to insert the Extruded-Cut feature.

Create a Linear Pattern.


- 65)** Click **Linear Pattern**  from the Features toolbar. The Cut-Extrude1 is displayed in the Features to Pattern box.



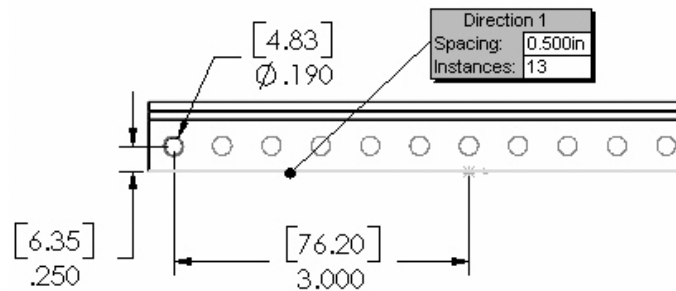
- 66)** Select the **bottom horizontal edge** of the Extrude Thin1 feature for Direction1. Edge<1> is displayed in the Pattern Direction box.

- 67)** Enter **0.5 [12.70]** for Spacing.

- 68)** Enter **13** for Number of Instances.

- 69)** The Direction arrow points to the right. Click the **Reverse Direction**  button if required.

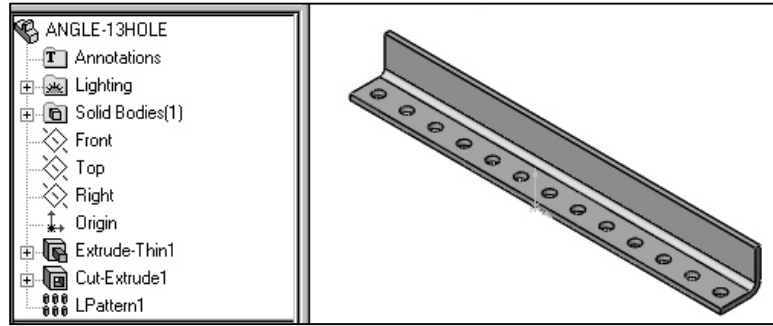
- 70)** Click **OK**  to insert a Linear Pattern.



71) Click **Isometric**



72) Save the ANGLE-13HOLE part. Click **Save**



Insert Fillet feature.

73) Click **Zoom to Area**



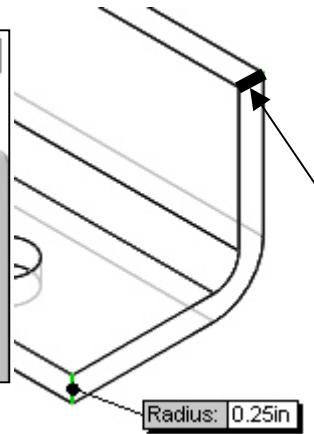
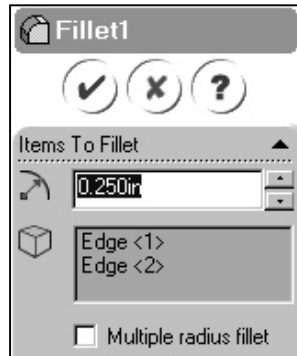
Select the right side of the **Extrude Thin** feature.

74) Click **Fillet** from the Features toolbar.



75) Enter **.250 [6.35]** for Fillet Radius.

76) Select the **two small edges**. Edge<1> and Edge<2> are displayed in Fillet box.

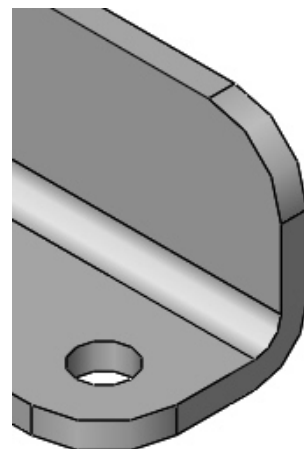


77) Click **OK** to create the Fillet.




Combine Fillets of the same size.

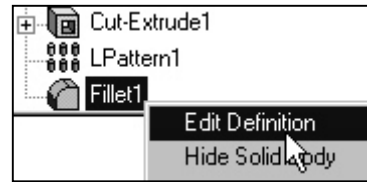
Edit the Fillet feature and add two left edges.



Edit the Fillet feature.

78) Click **Zoom to Area** . Select the left side of the **Extrude Thin** feature.

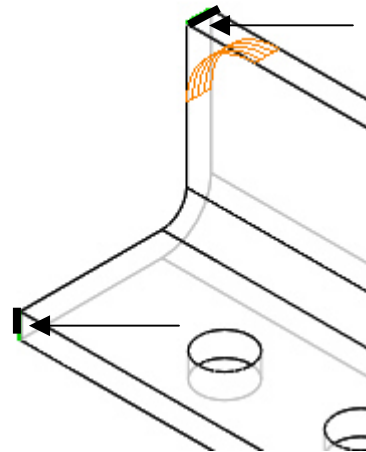
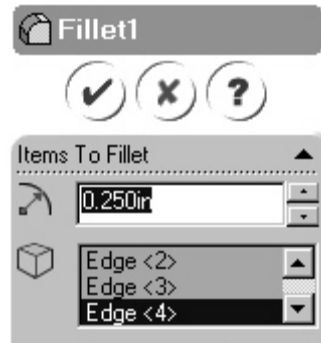
79) Right-click **Fillet1** in the FeatureManager.



80) Click **Edit Definition**.


81) Select the **two left small edges**. Edge<3> and Edge <4> are displayed in the Fillet box.

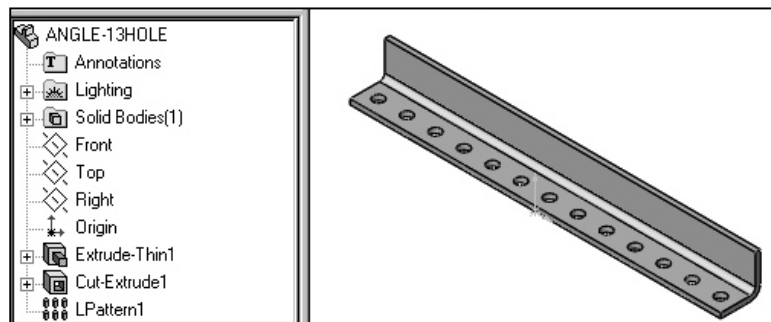
82) Click **OK**  to update the Fillet.



Display the Isometric view.

83) Click **Isometric** .

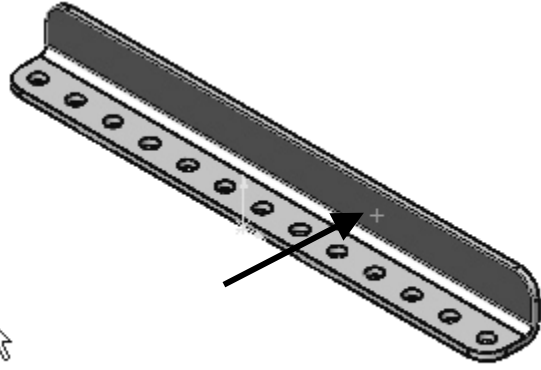
84) Save the ANGLE-13HOLE part. Click **Save** .



Insert a new sketch for the second Extruded Cut.

- 85)** Select the sketch plane. Click the **front face** of the Extrude Thin1. The cursor displays the


Face  feedback symbol.



- 86)** Click **Sketch** .

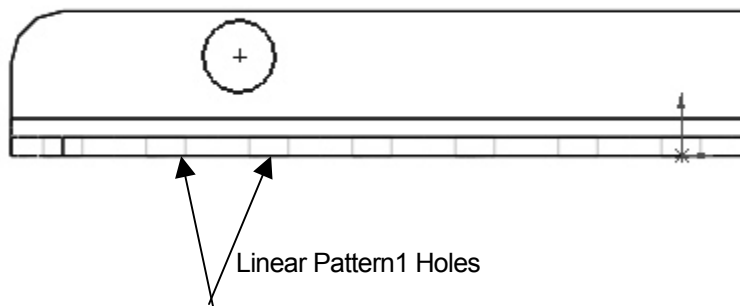
- 87)** Click **Front**  from the Standards view toolbar.

- 88)** Click **Hidden Lines Visible**

 from the View toolbar to display the Linear Pattern1 feature. Do not align the center point of the circle with the center point of the Linear Patter1 holes. The center point is position is controlled with dimensions.

- 89)** Click **Circle**  from the Sketch Tools toolbar.

- 90)** Sketch a **circle** on the left side of the Origin between two Linear Pattern1 holes.



Add dimensions.

91) Create a horizontal dimension. Click the **Origin**.

92) Click the **center point** of the circle.

93) Click a **position** below the horizontal profile line.

94) Enter **3.000**[76.20].

95) Create a vertical dimension. Click the **top horizontal line**.

96) Click the **center point**.

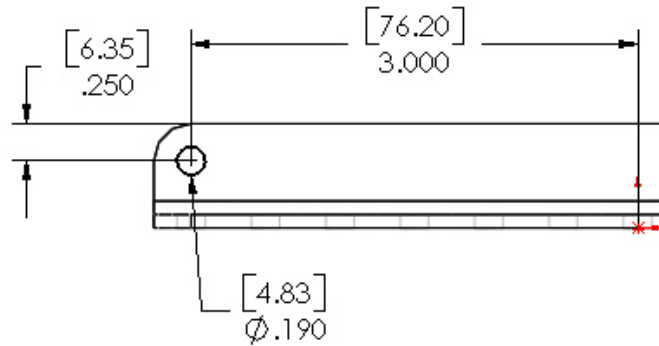
97) Click a **position** to the left of the profile.

98) Enter **.250** [6.35].

99) Create a diameter dimension. Click the **diameter** of the circle.

100) Click a **position** above the profile.

101) Enter **.190** [4.83].




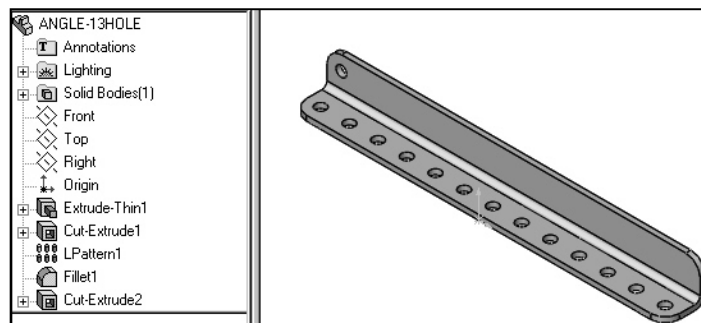
Insert an Extruded Cut.

102) Click **Extruded**

Cut  from the Features toolbar.


103) Select **Through All** for Direction1 End Condition.

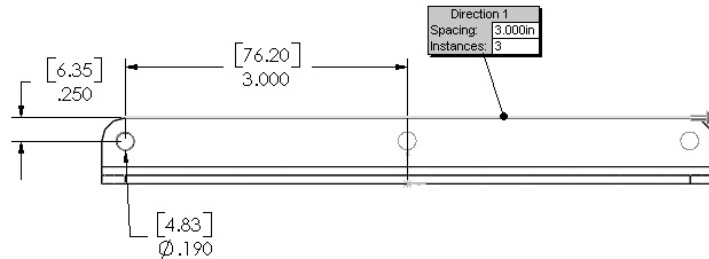
104) Click **OK**  to insert the Extruded-Cut feature.



Create the second
Linear Pattern.

105) Click **Top**  from the Standards view toolbar.


106) Click **Linear Pattern**  from the Features toolbar. The Cut-Extrude2 is displayed in the Features to Pattern box.



107) Select the **top horizontal edge** of the Extrude Thin1 feature for Direction1. Edge<1> is displayed in the Pattern Direction box.

108) Enter **3.000** [76.20] for Spacing.

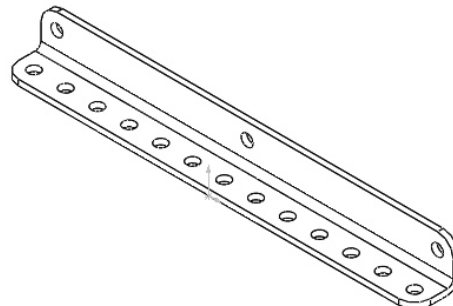
109) Enter **3** for Number of Instances.

110) The Direction arrow points to the right. Click the **Reverse Direction**  button if required.

111) Click **OK**  to create the Linear Pattern.

112) Click **Isometric** .

113) Save the ANGLE-13HOLE part. Click **Save** .

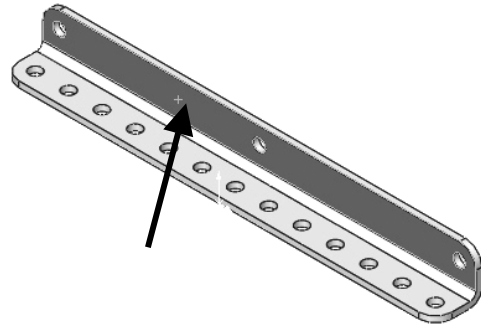



Insert a new sketch for the third Extruded Cut.

114) Select the sketch plane. Click the **front face** of Extrude Thin1.

115) Click **Sketch** .


116) Click **Front**  from the Standards view toolbar.

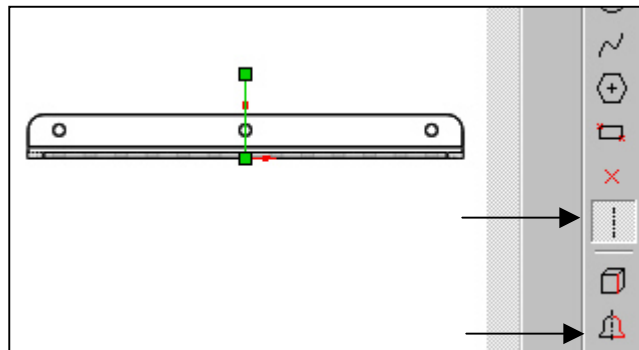


117) Sketch a vertical centerline. Click **Centerline**  from the Sketch Tools toolbar.

118) Click the **Origin**.

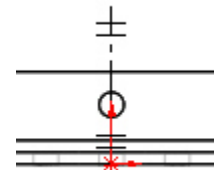
119) Click a vertical **position** above the top horizontal line.

120) Click **Sketch Mirror**  from the Sketch Tools toolbar.




Two sets of parallel lines are displayed on the centerline.

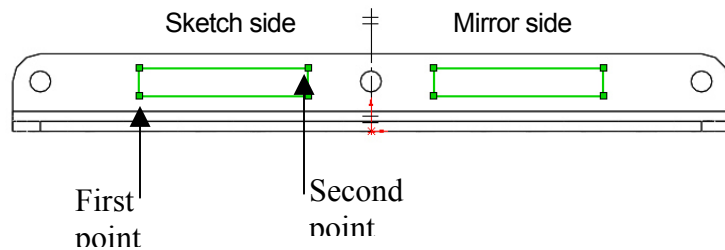
The centerline is a line of symmetry.



121) Click

Rectangle 
from the Sketch
Tools toolbar.

122) Do not align
the rectangle
first point and
second point to
the center
points of
Lpattern1.
Click the **first
point** of the
rectangle to the
left of the
Origin.

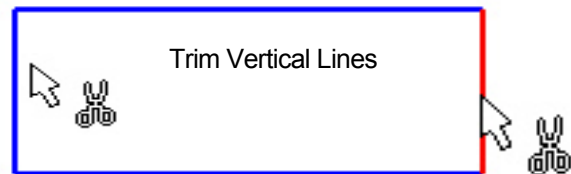


123) Click the
second point
of the rectangle
diagonally from
the first point,
to the left of the
Origin. A
second sketch
is created on
the Mirror side
of the centerline.

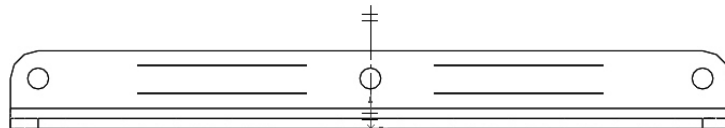
124) Trim the vertical lines. Click


Sketch Trim  from the
Sketch Tools toolbar.

125) Click the **two vertical lines** of
the left rectangle.




126) Click the **two
vertical line** of
the right
rectangle.

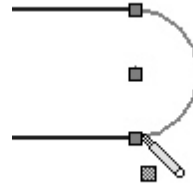


127) Sketch the right arc on the left side of the center line. Click **Tangent Arc**  from the Sketch Tools toolbar.




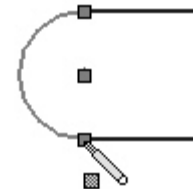
128) Drag the **mouse pointer** along the top horizontal line to the right. A dotted horizontal line is displayed.

129) Click the **top right endpoint**. Drag the **mouse pointer** to the right and downward. Click the **bottom right endpoint**. The cursor displays the Coincident to point feedback symbol  at each endpoint.

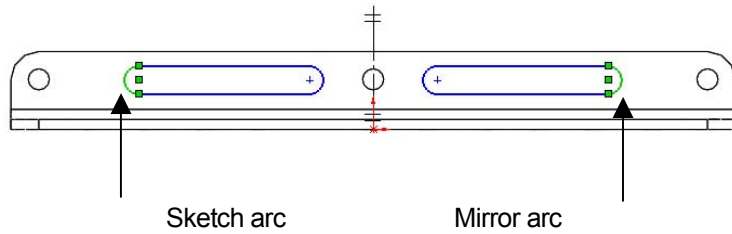


130) Sketch the left arc on the left side of the centerline. Drag the **mouse pointer** along the top horizontal line to the left. A dotted horizontal line is displayed.

131) Click the **top left endpoint**. Drag the mouse pointer to the right and downward. Click the **bottom left endpoint**. The cursor displays the Coincident to point feedback symbol  at each endpoint.




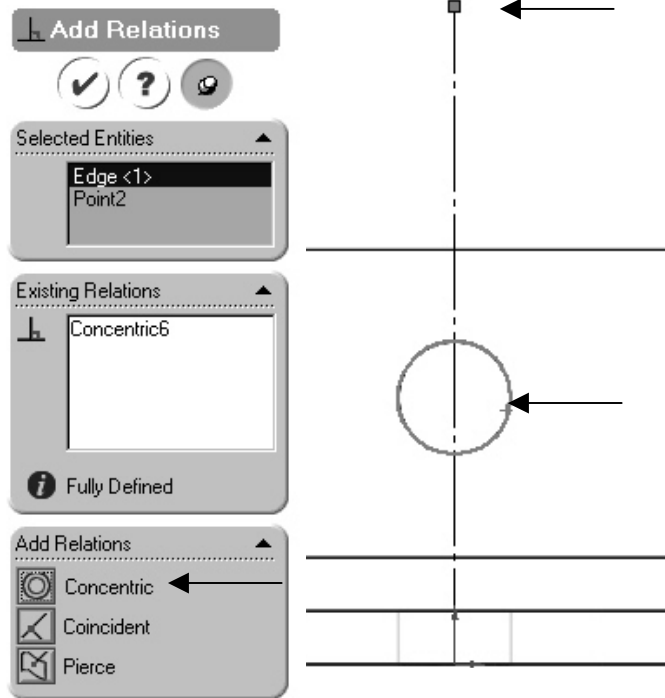
The two sketched arcs are displayed on the Mirror side.



- 132)** Deactivate Sketch Mirror. Click **Sketch Mirror** .

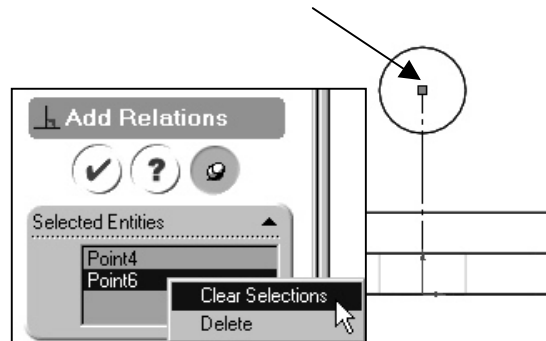
Add Relations.

- 133)** Zoom in on the centerline.
- 134)** Press the **Shift z** key, approximately 3 times.
- 135)** Click **Add Relation** .
- 136)** Select the **top endpoint** of the centerline.
- 137)** Select the **circle**.
- 138)** Click **Concentric** from the Add Relations box.



The endpoint of the centerline is positioned in the center of the circle.

Note: Right-click Clear Selections to remove selected entities from the Add Relations box.



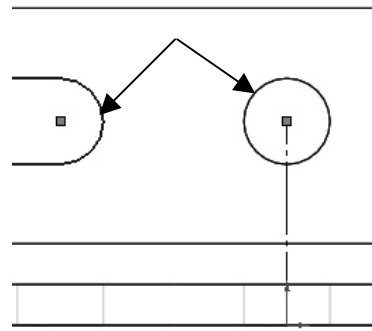
139) Click **Add Relation** .

140) Select the **circle**.

141) Select the **left arc**.

142) Click **Equal** from the Add Relations box.


The arc radius is equal to the circle radius.

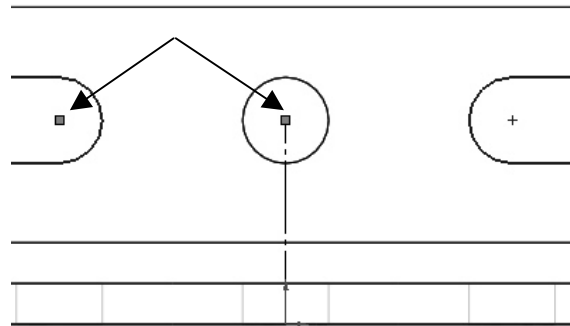


143) Select the **top endpoint** of the centerline.

144) Select the **left center point** of the left arc.

145) Click **Horizontal** from the Add Relations box.

146) Click **OK**  to add a Horizontal relation.



The right arc is horizontally aligned to the left arc due to symmetry from the Sketch Mirror tool.

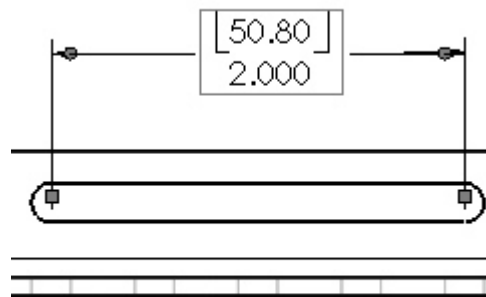
Dimension the slots.

147) Click **Dimension**



148) Dimension the left slot. Click the **left center point** of the left arc. Click the **right center point** of the right arc.

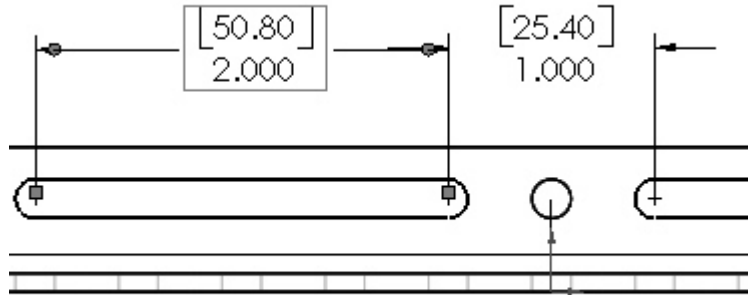
149) Click a **position** above the top horizontal line.



150) Enter **2.0 [50.80]** in the Modify dialog box.

151) Click the **Check Mark** .


152) Dimension the distance between the two slots. Click the **right arc** of the left slot.



153) Click the **left arc** of the right slot.

154) Click a **position** above the top horizontal line.

155) Enter **1.000 [25.40]** in the Modify dialog box.

156) Click the **Check Mark** . The black sketch is fully defined.

Insert an Extruded Cut.

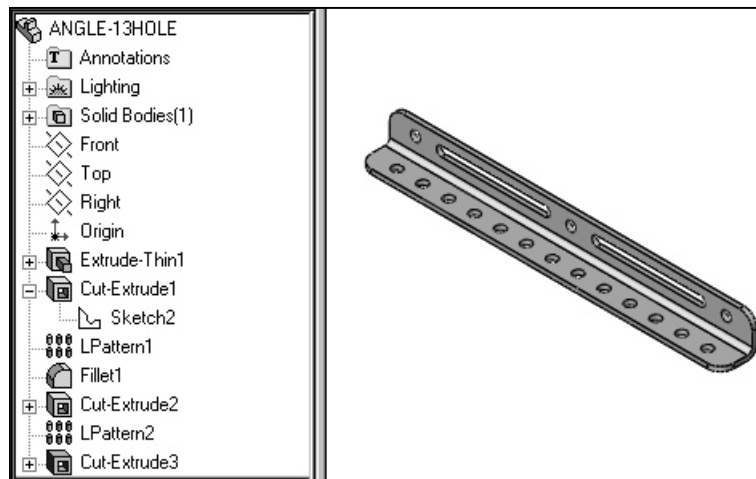
157) Click **Extruded**


Cut  from the Features toolbar.

158) Select **Through All** for Direction1 End Condition.

159) Click **OK**  to create the Extruded-Cut feature.

160) Click **Isometric** .



161) Save the ANGLE-13HOLE part. Click **Save** . The ANGLE-13HOLE is complete.

Note: The dimension between the two slots is over-defined if the arc center points are aligned to the center points of Lpattern1.

Review the ANGLE-13HOLE Part.

The ANGLE-13HOLE utilized an open L-Shaped profile sketched on the Right plane. The Extruded Thin feature with the Mid Plane option located the part symmetrical to the Right plane.

The first Extruded Cut feature created the first hole sketched on the top face of the Extruded Thin feature.

The first Linear Pattern created an array of 13 holes along the bottom horizontal edge.

The Fillet feature rounded the four corners.

The second Extruded Cut feature created on the second hole sketched on the front face of the Extruded Thin feature.

The second Linear Pattern created an array of 3 holes along the top horizontal edge.

The third Extruded Cut feature created two slot cuts sketched with the Sketch Mirror.

Exercise

Create the ANGLE BRACKET Part. The Base Extrude feature is sketched with an L-Shaped profile on the Right Plane. The ANGLE BRACKET Part is machined from 0.060 [1.5mm] Stainless Steel flat stock. The default units are inches.

