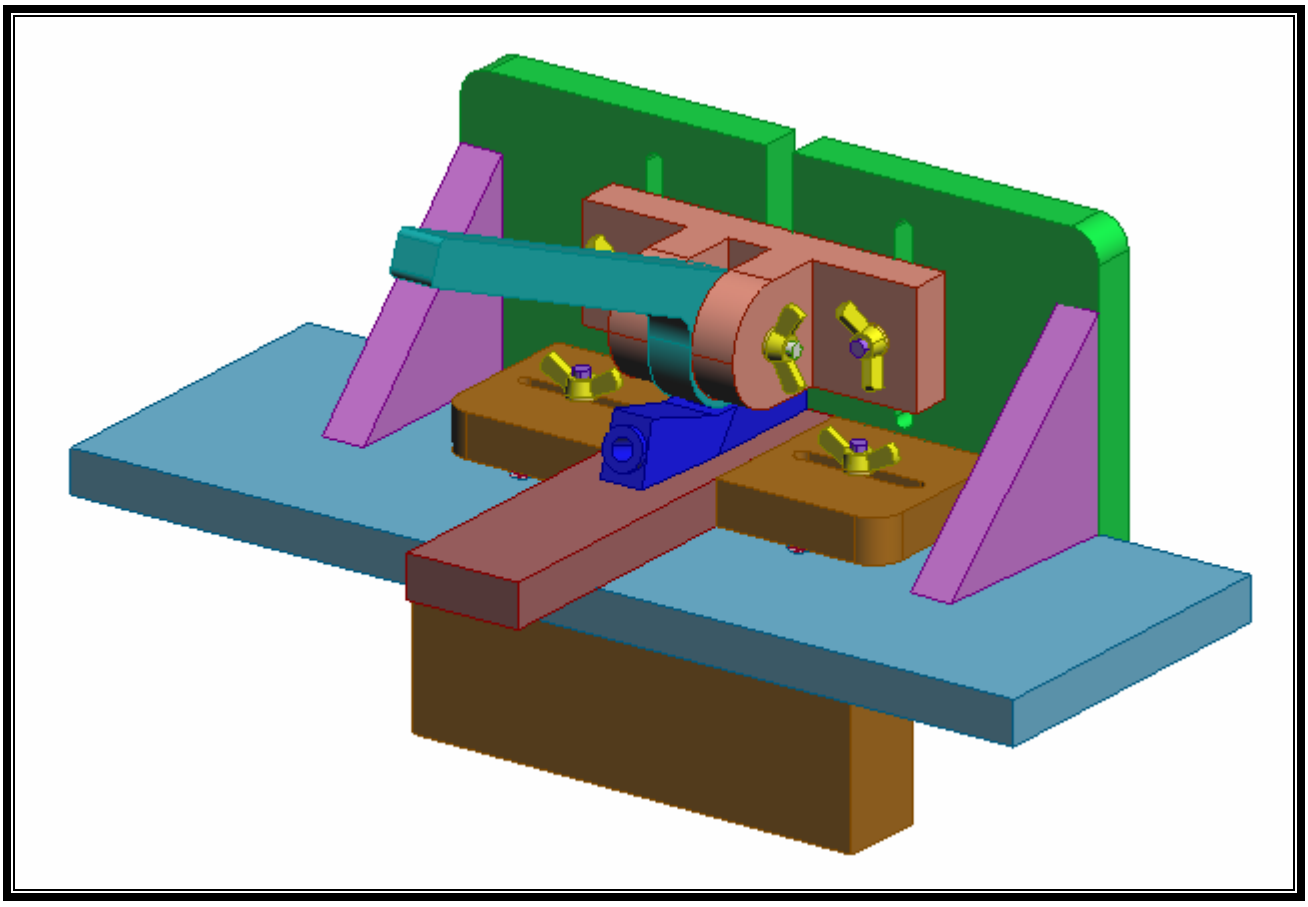
Pocket Hole Jig

Prepared by: Harry Hawkins



This project is the design of a Pocket Hole Jig. This device can be used to quickly use a low-priced commercial drill guide and special drill to make pocket holes in wood members. The resultant screwed and glued joint is relatively simple and very strong.

You will use the normal Pro/DESKTOP functions of creating workplanes and sketches as well as other functions. A complete set of dimensioned drawings are included at the end of the tutorial. These drawings should be consulted to get accurate dimensions as you generate designs.

A number of separate pieces will be designed such as the base, fence, cam lever, screw heads and others then these parts will be assembled into a completed unit. In the end you will make a photo album of the design.

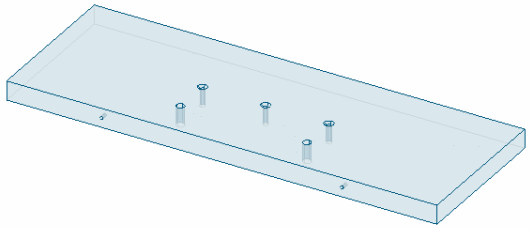
This tutorial is presented in serial order and you should complete it by following this sequence. The numbers of the steps will continue through each part so that the entire tutorial will precede from number 1 to the last number rather than separately number the steps for each part.

When you open Pro/DESKTOP be sure to go to Tools then Options then select the Units tab and set the units in both fields to Inches.

As the tutorial progresses directions will be less specific since you should by then be relatively familiar with how to extrude etc.

Knowledge or information you should know prior to beginning this tutorial:	What you should learn from completing this tutorial:
★ Navigate in Windows. ★ Create, save, rename and delete folders. ★ Mouse commands – right and left click and mouse wheel. ★ Basic Pro/DESKTOP sketching tools – lines, circles etc. ★ Pro/DESKTOP Workplanes and sketches.	★ Use the Duplicate function. ★ Use the Mirror function. ★ Create an Assembly. ★ Create an Album. ★ Use constraints to assemble parts. ★ Suppress or delete constraints when necessary to create additional part placement. ★ Mirror Solids

Base



1. Start Pro/DESKTOP and open a new design. Be sure to set Units to Inches.

2. Press CTRL - W to go to ortho display.

3. Draw a rectangle 17" x 6" according to figure 1.

4. Extrude this rectangle to a distance of ¾" (0.75") above the workplane. It should look like figure 2.

5. Select the top face (it will turn red) then right click and create a new sketch. Name it Mounting Holes. Use the dimensions given in figure 3 to place a 0.2D hole at the center or orgin.

6. Draw a horizontal line from the origin to the right 2-1/2" (hold the shift down to keep it horizontal).

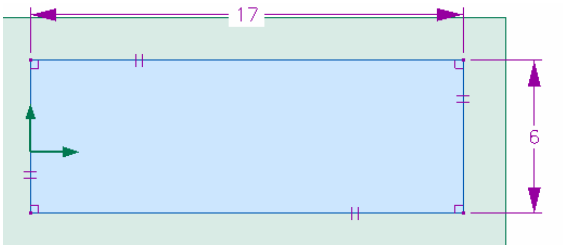


Figure 1. Base outline dimensions.

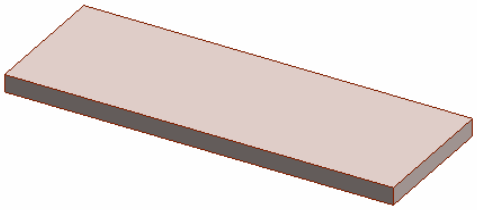


Figure 2. Base extruded ¾"

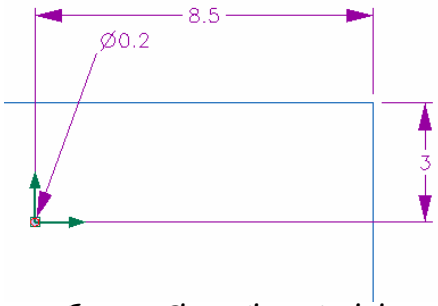


Figure 3. Placing the center hole.

7. Place a 0.2" Diameter hole at the right end of this line as shown in figure 4.

8. Draw a straight line from the origin down a short distance. This will be the axis used for mirroring the right hole. It is shown by an arrow in figure 4.

9. With the right hole selected, select Line then Mirror from the drop down menu. Select the Axis tab (it will have 0 in it) then double click the vertical line (in red) shown in figure 5. If the preview box is checked you will see the hole at the left. Click OK to execute the command.

10. Use the delete tool to delete all unnecessary lines until a valid profile is obtained. (the circles will fill with color)

11. Shift-T to see the design in 3D. as in figure 6.

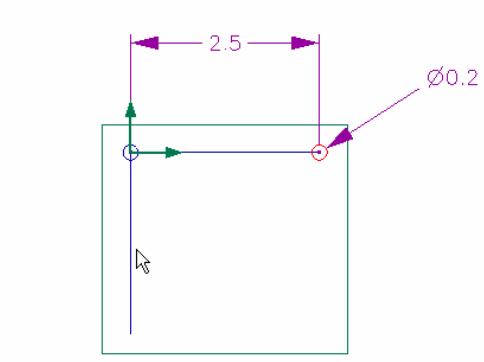


Figure 4. Placing right hole.

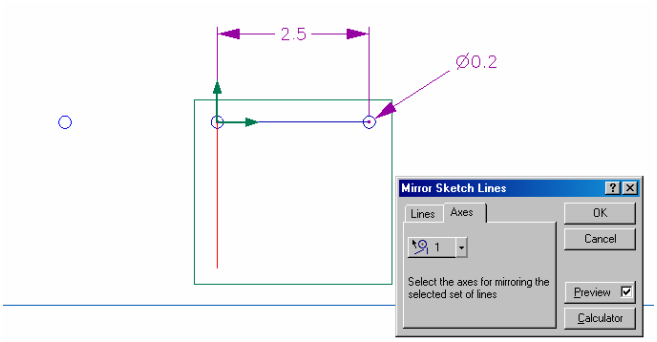


Figure 5. Duplicating the left hole.

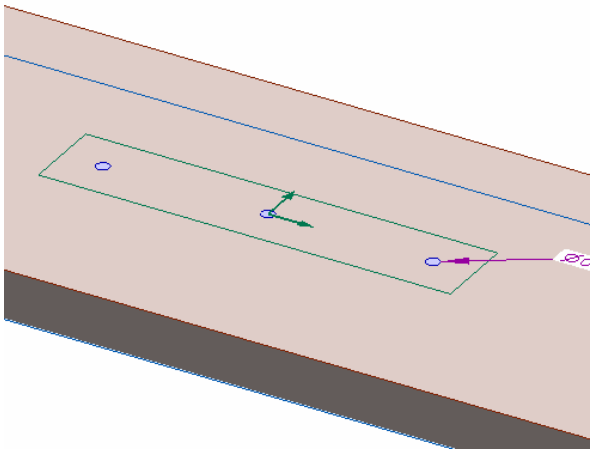


Figure 6. Mounting Holes ready to be extruded.

12. Select the three holes (they will turn red) then select Insert Holes (under Features). Set the dialog boxes according to figure 7.

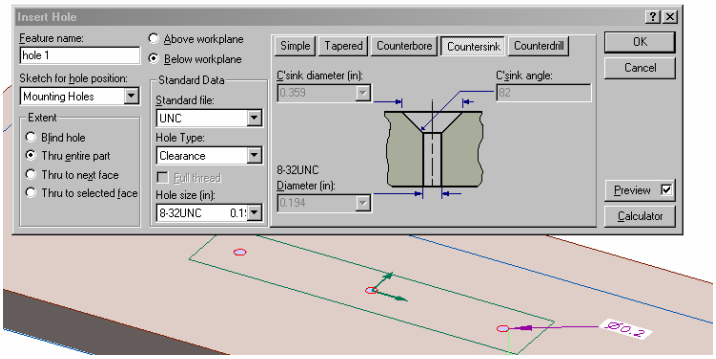


Figure 7. Inserting holes settings:
Countersink
Below Workplane
UNC
Clearance
8-32UNC

13. Press OK to execute. The holes will be inserted and should look like figure 8.

14. Rotate the design to see the back face (red arrow in figure 8).

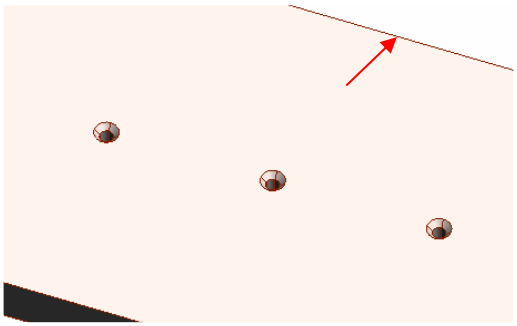


Figure 8. Countersunk holes installed.

15. Use face selection to select the back face then right click and create a new sketch. Name the sketch fence mounting holes. Use Shift-W to see the sketch in ortho as in figure 9.

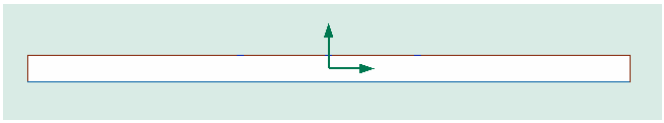


Figure 9. Fence Mounting Holes Sketch.

16. Layout and place a 1/8" D (0.125) hole on the right as in figure 10.

17. Delete the construction line used to locate the hole so that the hole fills with color.

18. Draw a vertical line from the origin upwards. This is temporary and will be the axis for mirroring.

19. Select the previously drawn hole and use mirror and the axis line to create the hole at the left as in figure 11.

20. Use extrude to extrude the holes. Subtract Material below the workplane ¼". Figure 12 shows the result.

21. Two more holes are needed in the base. Select the top face and create a new sketch. Name it Sliding Clamp holes.

22. Place two 5/16" (0.3125) holes 1-1/2" above the existing outside holes as in figure 13. Delete unnecessary lines for a valid profile.

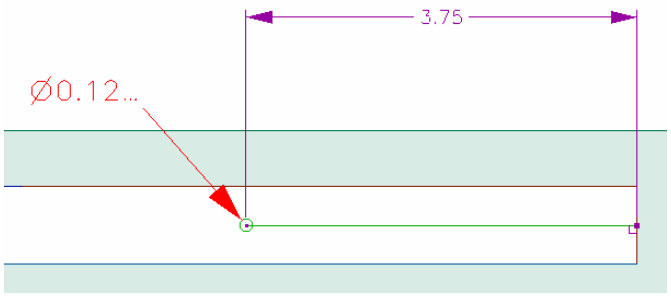


Figure 10. Placing right mounting hole.

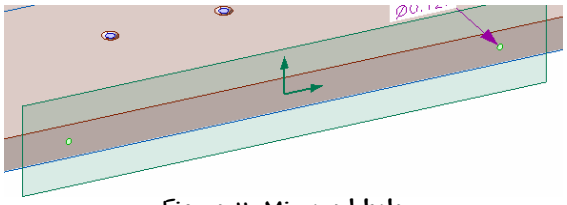


Figure 11. Mirrored hole

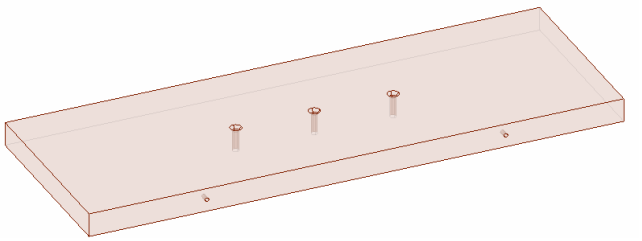


Figure 12. Fence mounting holes completed.

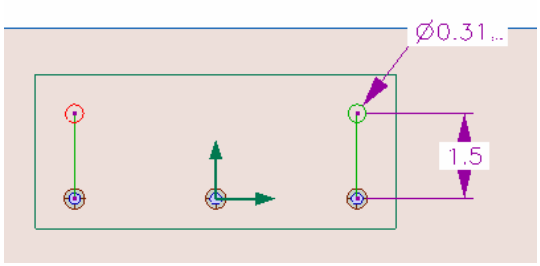


Figure 13. Inserting 5/16" holes.

23. Use Project Profile to subtract material below the workplane through the entire part as in figure 14.

24. This completes the base unit. It should look like figure 15. This figure is in transparent mode so that you can see all the features.

25. Save the design and name it Base if you have not already done so.

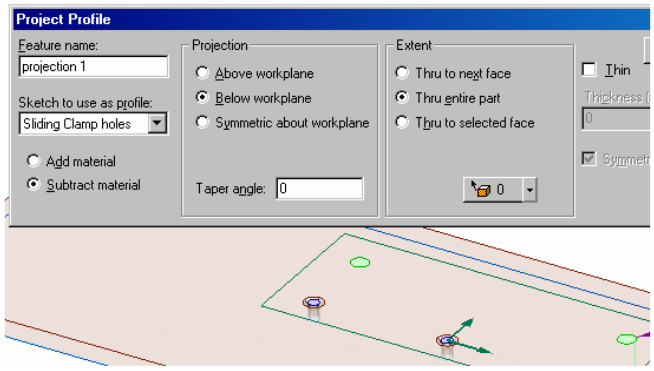


Figure 14. Project hole profile through base.

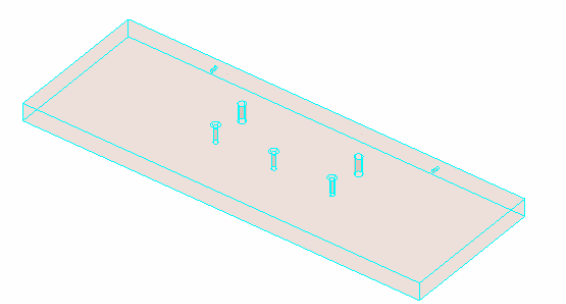
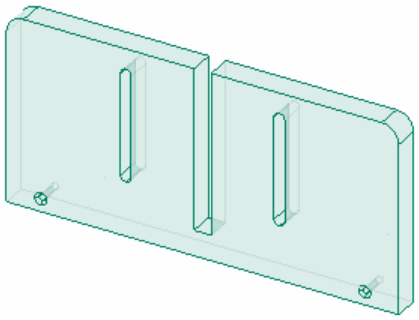


Figure 15. Completed Base.

Fence



26. Create a new design.

27. Select the Frontal workplane and create a new sketch. Name it Fence.

28. Draw a rectangle 5-1/2" x 11-1/2".

29. Draw the center slot according to the dimensions in figure 16.

30. Use delete to delete unwanted lines. The sketch should fill with color.

31. Extrude the profile above the workplane a distance of ¾". The result should look like figure 17.

32. Select the front face and right click. Create a new sketch. Name it base mounting holes.

33. Shift-W to see this in orthographic format as in figure 18.

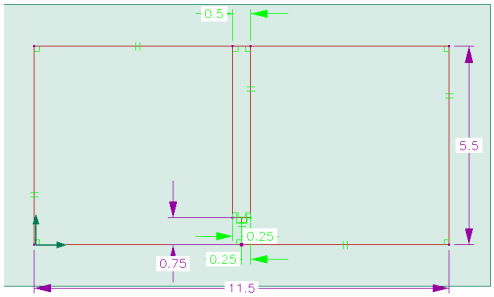


Figure 16. Drawing the center slot

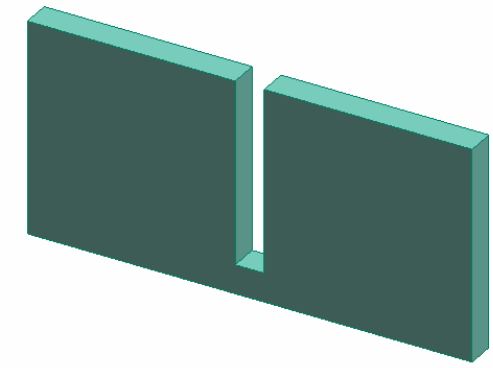


Figure 17. Extruded fence.

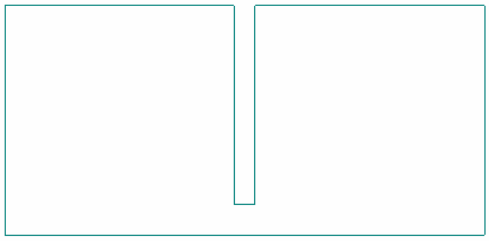


Figure 18. Sketch for base mounting holes

34. Locate and create a circle at the right of the design according to figure 19. This hole should be 1/8" diameter but it could be any size since it will be used for alignment purposes during assembly. The hole should be 3/8" (0.375) up from center and 4-3/4" to the right as shown in figure 19.

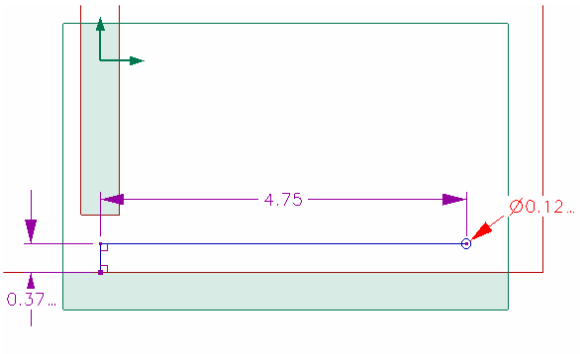


Figure 19. Layout for right fence mounting hole.

35. Delete all construction lines until the hole fills with color.

36. Draw a temporary axis line from the center or origin up or down.

37. Select the hole just drawn then select mirror and select the temporary axis line to create the hole on the left. Delete the temporary axis line.

38. Figure 20 shows the completed holes.

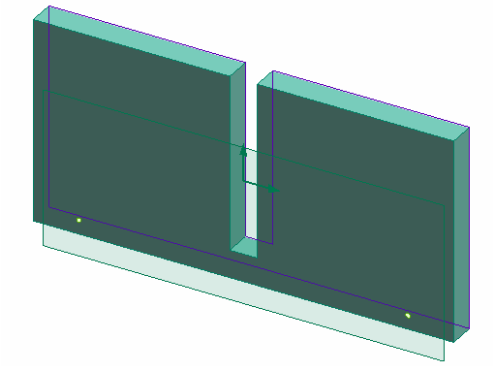


Figure 20. Completed base mounting holes.

39. Select Project Profile and project the holes (remove material, below workplane, through the entire piece).

40. Select these two holes and use Insert Holes to make them countersink, UNC 8-32, clearance and through the entire part. The fence should now look like figure 21

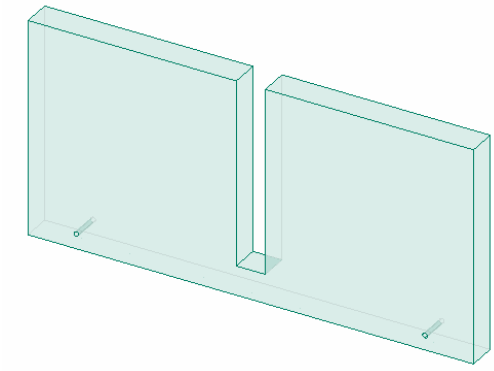


Figure 21. Completed base mounting holes.

41. Select the front face of the fence and create a new sketch. Name it Fence slots.

42. Use Shift-W to view the orthographic.

43. Draw the right slot according to the dimensions given in figure 22. Draw two 5/16" holes then draw horizontal diameter lines in each hole. Connect the ends of these lines to draw the sides of the 3" slot.

44. Delete unneeded lines. The slot will fill with color when all unnecessary lines are deleted as in figure 22.

45. Create a temporary axis line from the Origin center up or down.

46. Select the right slot lines and use mirror to duplicate the left slot.

47. Delete the temporary axis line.

48. Figure 23 shows what the fence should look like at this point.

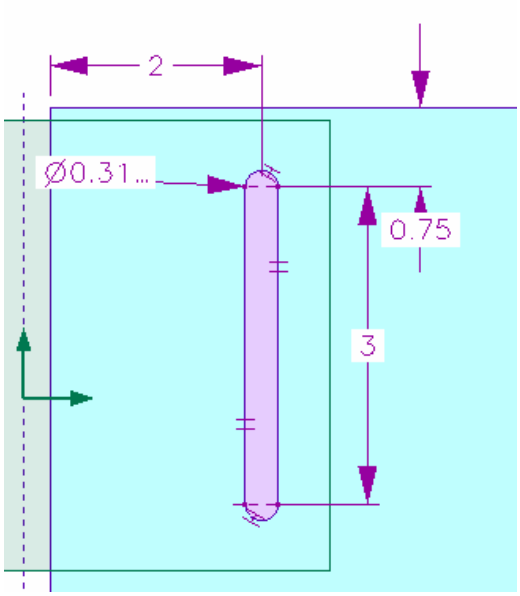


Figure 22. Layout for right slot.

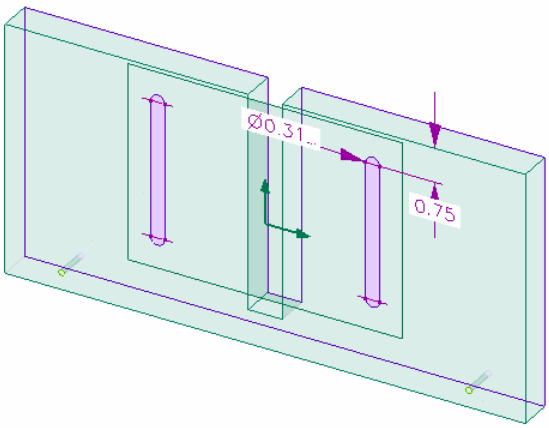


Figure 23. slots ready to be projected.

49. Use Project Profile to project the slots through the entire part (remove material, below the workplane).

50. Figure 24 shows the fence with the slots completed.

51. Use edge selection to select the two top outside edges of the design (right edge shown at red arrow in figure 24).

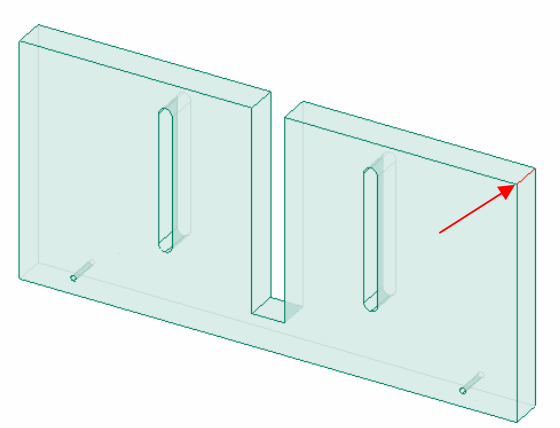


Figure 24. Completed Slots.

52. Right click and select Round Edges...

53. In the Round dialog box, set the radius to ½" as shown in figure 25. Notice that a trial view in yellow will show on the design.

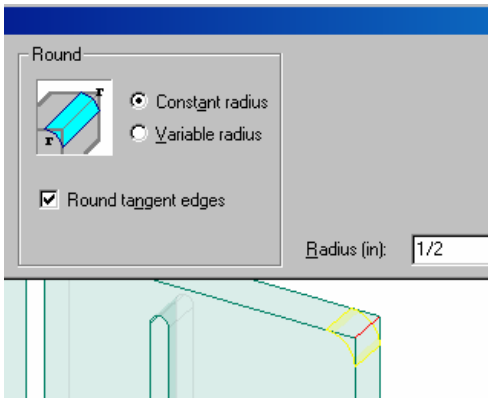


Figure 25. Rounding the top corners.

54. Click OK to have the corners rounded. Figure 26 shows the top corners rounded.

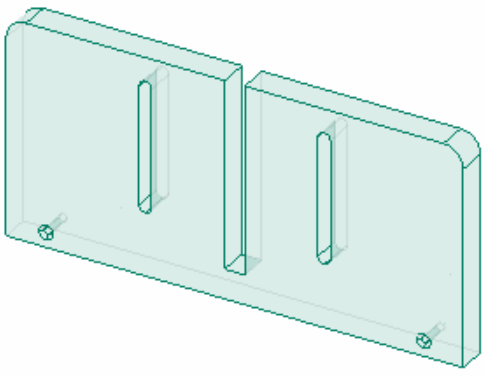
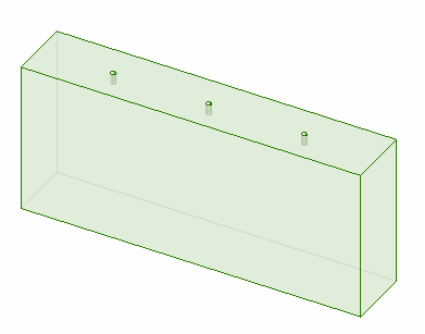


Figure 26. Completed Fence.

55. This completes the fence. Save it and name it Fence if you have not yet done so.

Clamp Block



56. Open a new design and create a rectangle on the Frontal workplane. This should be 8" x 3-1/2". Name it Block

57. Extrude this design above the workplane a distance of 1-1/2".

58. Figure 27 shows the completed extrusion.

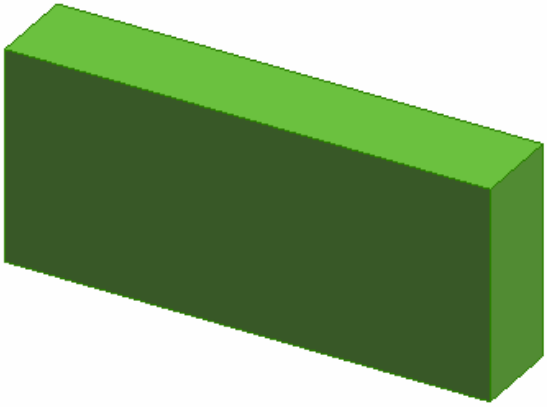


Figure 27. Extruded Clamp Block.

59. Select the top face and create a new sketch. Name it mounting holes.

60. Layout three 1/8" (0.125") holes as shown in figure 28.

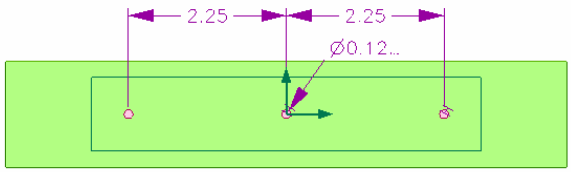


Figure 28. Hole layout for clamp block.

61. Delete all unneeded lines so that you have a valid profile and the holes fill with color.

62. Extrude these holes a distance of ¼" below the workplane.

63. Figure 29 shows the completed clamp block. Save it and name it Clamp Block if you have not already done so.

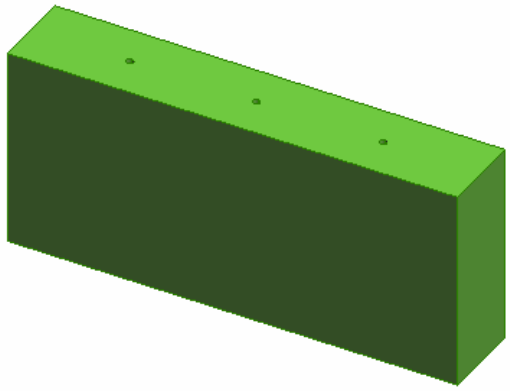
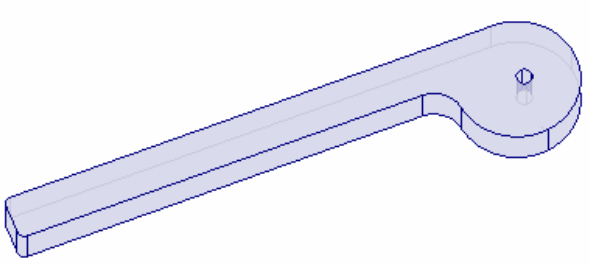


Figure 29. Completed Clamp Block.

Cam Lever



64. Open a new design.

65. It is easier to draw circles then vertical and horizontal lines from the origin as shown in figure 30 then construct the handle.

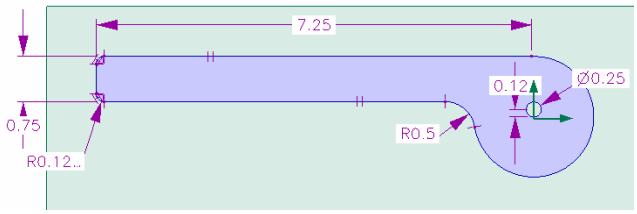


Figure 30. Detail to draw handle from circle.

66. Draw the profile of the cam lever according to the dimensions shown in figure 30 and 31. Be careful to offset the center hole by 1/8" otherwise the cam will not function properly.

67. Delete all unnecessary lines so that you have a filled profile.

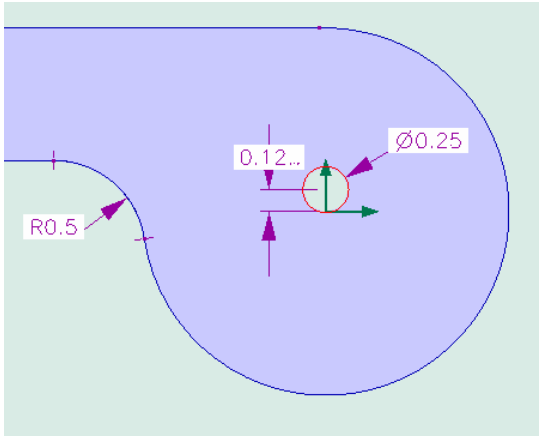


Figure 31. Layout for offset center hole.

68. Use the Arc or Fillet tool to stretch the lower intersection of the handle and the head to a ½" radius as shown in figure 32. You may need to use constraint dimensioning to set the radius accurately.

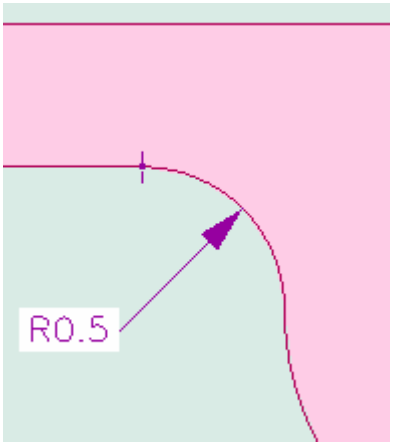


Figure 32. Handle to Head Radius.

69. Extrude the profile a distance of ¾" above the workplane. Your design should look like figure 33

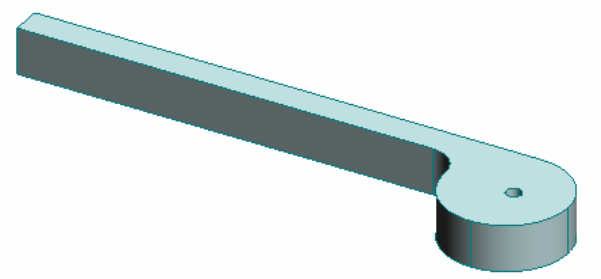


Figure 33. Completed extrusion.

70. Select the top and bottom edges of the end of the handle and round them to a radius of ¼". This is shown in figure 34.

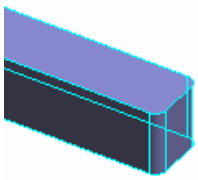


Figure 34. Rounded edges on handle end.

71. This completes the cam handle. Save it and name it Cam Handle if you have not already done so. It should look like figure 35.

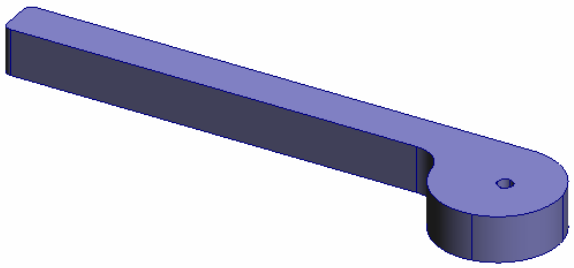
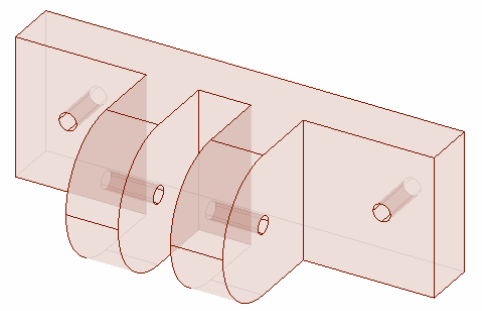


Figure 35. Completed Cam Handle.

Cam Holder



72. Open a new design and create a new sketch on the profile workplane. Name it Cam Holder.

73. Draw a 2" x 6" rectangle.

74. Extrude this rectangle a distance of ¾" below the workplane.

75. This design should look like figure 36.

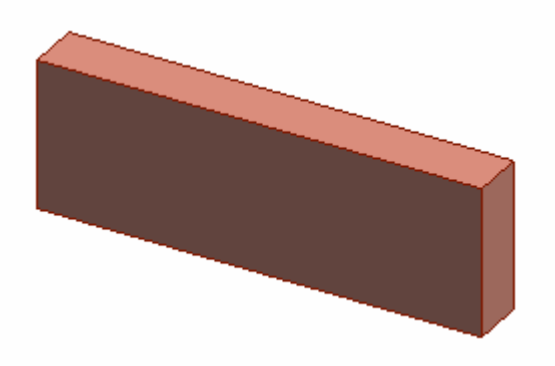


Figure 36. Extruded Cam Holder Back.

76. Create another sketch on the frontal workplane. Name it mounting holes.

77. Layout and draw two ¼"D holes according to the dimensions in figure 37.

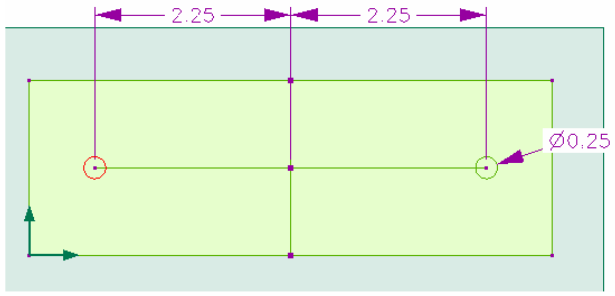


Figure 37. Layout for mounting holes.

78. Delete all unnecessary lines so that you have a valid profile.

79. Project Profile these holes below the workplane, through the entire piece.

80. Figure 38 shows the design at this point.

81. Select the front face and create a new sketch. Name it Cam Arms.

82. Shift-W to see the orthographic of the design.

83. Draw two rectangles according to the dimensions in figure 39. Start at the center and draw a line 3/8" to one side then draw the rectangle from there.

84. Delete all unnecessary lines so that you have a valid profile.

85. Extrude the profile above the workplane and add material to a distance of 2"

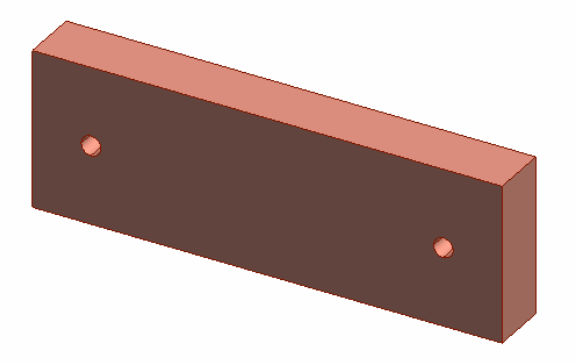


Figure 38. Cam Holder Back with holes installed.

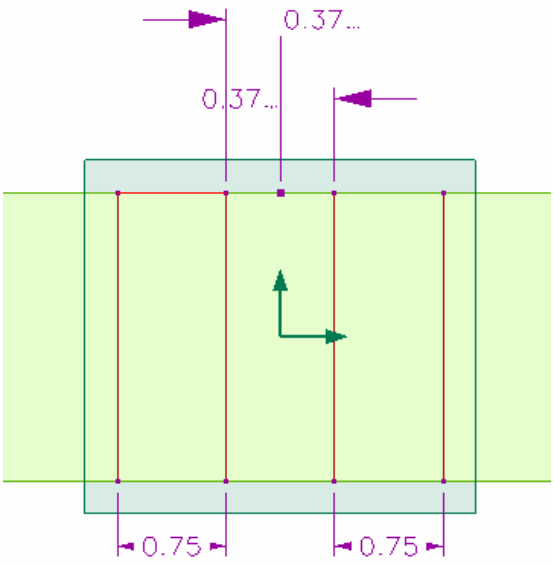


Figure 39. Layout for Cam Arms.

86. Figure 40 shows the extruded arms.

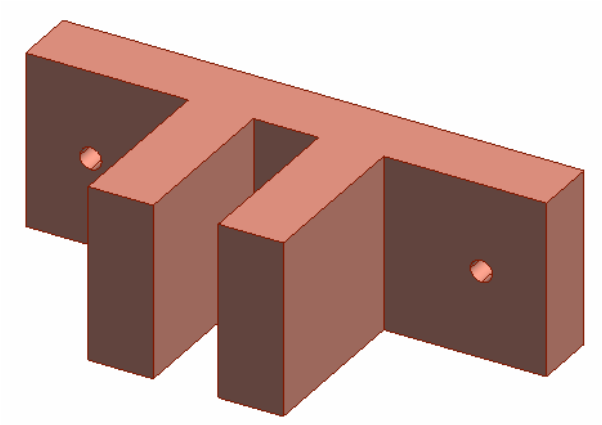


Figure 40. Extruded Arms.

87. Select the outside face of the right arm. Right click and create a new sketch. Name it Arm Round and Hole.

88. Shift-W to see the orthographic of this sketch.

89. Create the profile of the round and the ¼" center hole as shown in figure 41. First draw vertical lines from the origin up and down beyond the arm profile. Now draw concentric circles of 2D and 1/4D at the origin. You must also draw straight lines over the top left and bottom left lines. Draw the left vertical line away from the edge to the left in order to create the round profile.

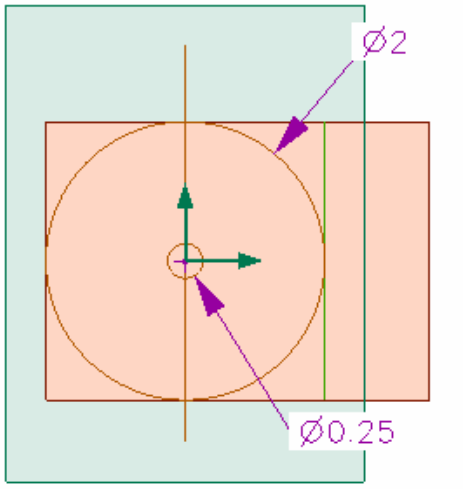


Figure 41. Layout for round and center hole.

90. Delete unnecessary lines so that you have a valid, filled profile for the ¼" hole and the two corners. This should look like figure 42. Notice that the filled area extends over the end of the arm.

91. Use Project Profile to remove material, below the workplane and through the entire piece.

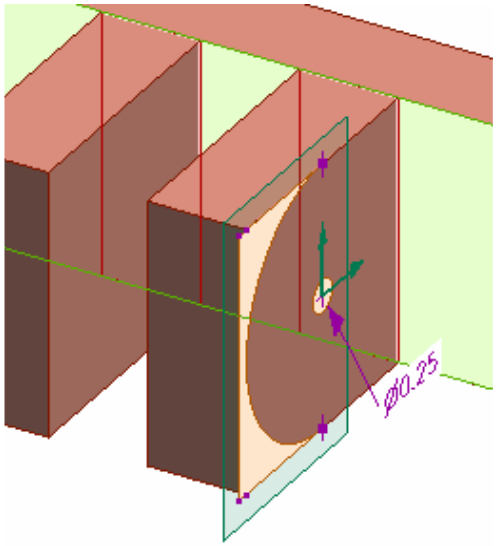


Figure 42. Profile for hole and round.

92. Figure 43 shows the resultant removal of material.

93. This completes the Cam Holder. Save it and name it Cam Holder if you have not already done so.

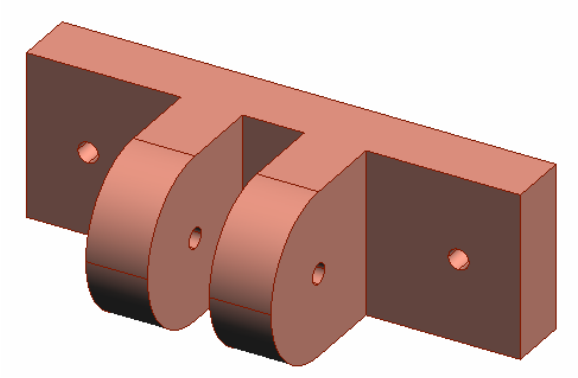
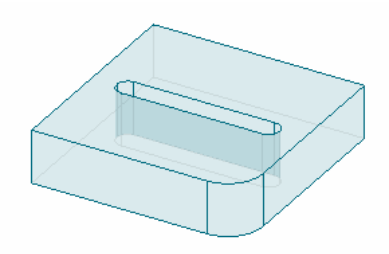
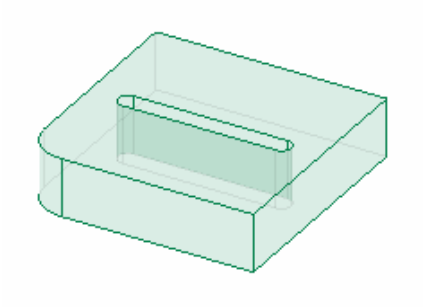


Figure 43. Completed Cam Holder





Right and Left Guide Blocks

94. Open a new design.

95. On the base workplane, create a 3" x 3" square. Use the rectangle tool and hold down the shift to make a square.

96. Extrude this design to a distance of ¾". It should look like figure 44.

97. Select the top face, right click and create a new sketch. Name it slot. First locate the ¼" holes, then draw horizontal lines to the sides of the each hole and finally draw the 2" sides of the slot.

98. Figure 45 shows the slot drawn before deleting unnecessary lines.

99. Delete all unnecessary lines so that you have a filled, valid profile of the slot.

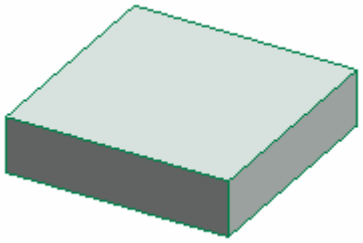


Figure 44. Extruded Right guide block.

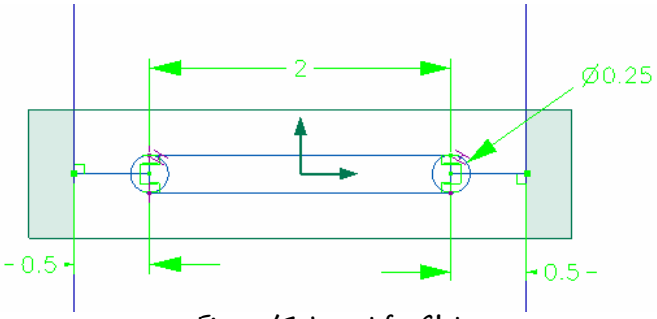


Figure 45. Layout for Slot.

100. Use Project Profile to project the slot by removing material, below the workplane and through the entire part.

101. Figure 46 shows what the guide block will look like at this point.

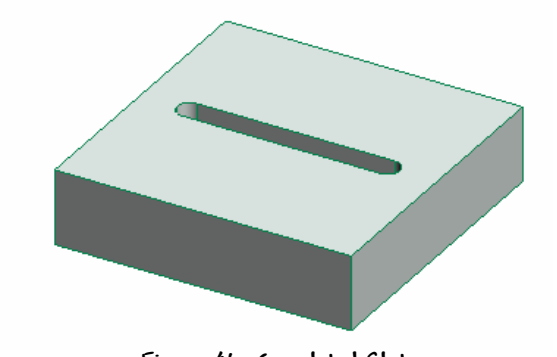


Figure 46. Completed Slot.

102. Select the right-front vertical edge of the solid, right click and select round edges. Set the radius to ½" then click OK to execute.

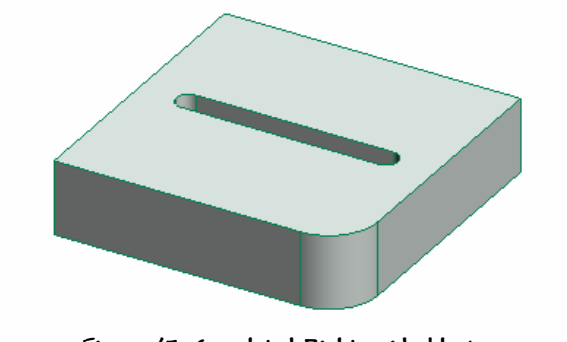


Figure 47. Completed Right guide block.

103. Figure 47 shows the completed Right Guide Block. Save it and name it Right guide Block.

104. To create the left block we will mirror the solid right block just made.

105. Shift-W to see the right block in orthographic.

106. Use part select to select the solid. It will turn red as in figure 48.

Figure 48. Selected right guide block.

107. From the Feature drop down menu, select Modify Solids... then select Mirror Solids...

108. The Mirror Solids dialog box will appear as in figure 49.

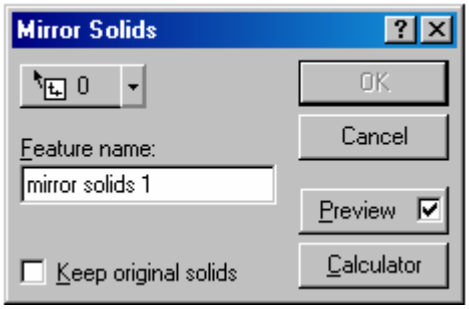


Figure 49. Mirror Solids dialog box.

109. Be sure to uncheck the Keep original solids box since we only want the new mirror image and not the original.

110. Click on the Selection button at the top left (it has a 0 in it). This will display several choices. Click on Workplanes.

111. Now click or double click on the vertical line as shown in red in figure 50. You will see a yellow mirror image of the duplicate at the left. This is shown in figure 50.

112. Click OK to execute and you will now have the left guide block as shown in figure 51.

113. Save this design and name it Left Guide Block if you have not already done so.

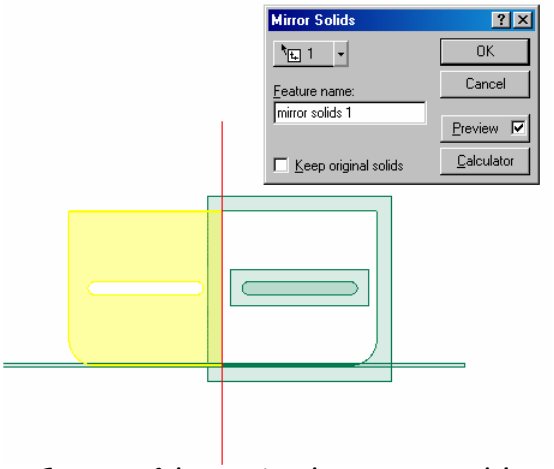


Figure 50. Selecting Workplane to mirror solid.

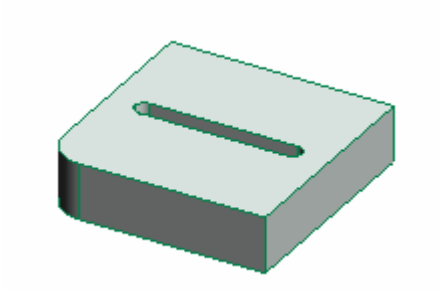
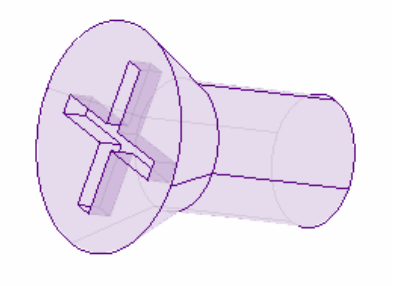


Figure 51. Completed Left Guide Block.



Philips Head Screw

114. Open a new design.

115. Draw a 0.164D circle at the origin as shown in figure 52.

116. Extrude this circle a distance of ¼" above the workplane.

117. Figure 53 shows the resulting solid.

118. Select the top circular surface of the solid, right click and create a new sketch. Name it Head.

119. Shift-W to see the new sketch in orthographic.

120. Draw a 1/4D circle centered on the origin.

121. Figure 54 shows the new sketch and circle on top of the solid shank.

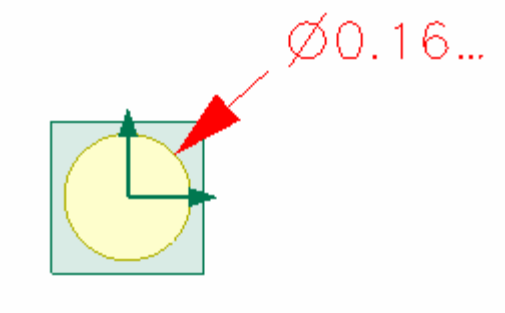


Figure 52. Screw shank diameter.

Figure 53. Extruded shank.

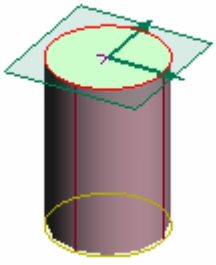
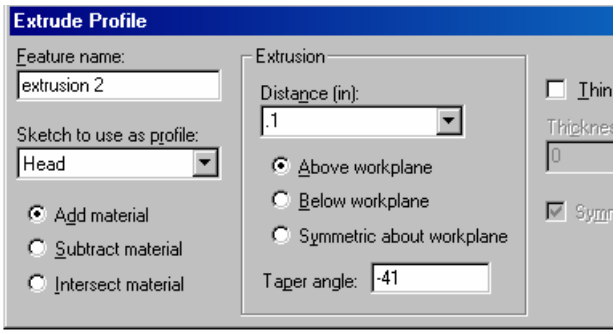


Figure 54. Head profile ready for extrusion.

122. Extrude this profile above the workplane a distance of 0.1". Also set the Taper Angle to -41° as shown in figure 55

123. Press OK to execute the extrusion.



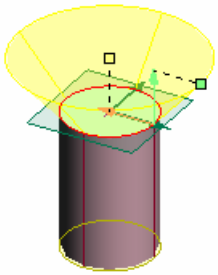


Figure 55. Extruding the tapered head.

124. Figure 56 shows the extrusion after it is completed.

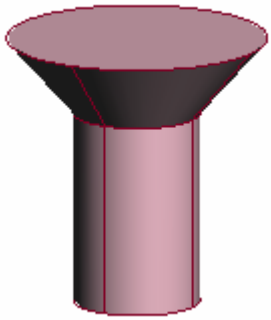


Figure 56. Completed head.

125. Select the top surface of the head, right click and create a new sketch. Name it Philips Cross.

126. Use the dimensions in figure 57 to create a vertical rectangle 0.03" x 0.115") from the origin upwards. This will be the first leg of the Philips Cross.

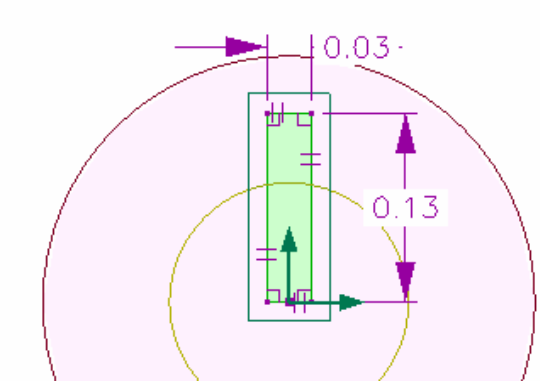


Figure 57. First leg of Philips Cross.

127. Select the rectangle then under Edit, select Duplicate.

128. Figure 58 shows the Duplicate dialog box. Complete it by selecting the circular button, and then make sure that 4 is entered in the Number box. Also the Angle should be total and 360. This will create 4 duplicates spaced rectangles evenly around the 360 degree circle.

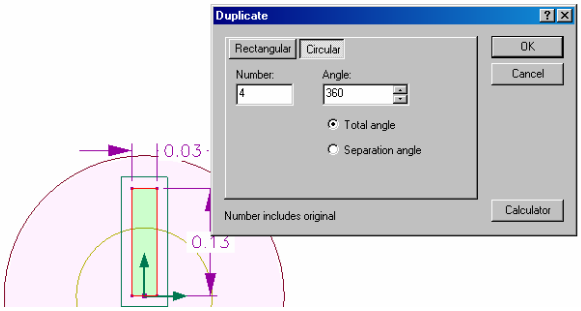


Figure 58. Duplicate dialog box set for circular.

129. Press the OK button to execute the duplicate. It should look like figure 59.

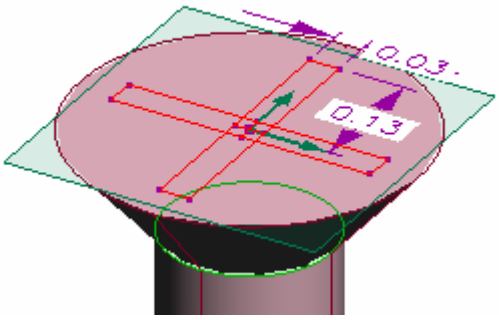


Figure 59. Duplicated legs form cross.

130. Delete unwanted lines so that the cross will fill and become a valid profile.

131. With the cross lines selected, use Extrude Profile to remove material below the workplane a distance of 0.03".

132. Press OK to execute the extrusion. The solid should now look like figure 60. This is the head and shank of the screw. Since threads will not show in the final assembly there is no need to construct them.

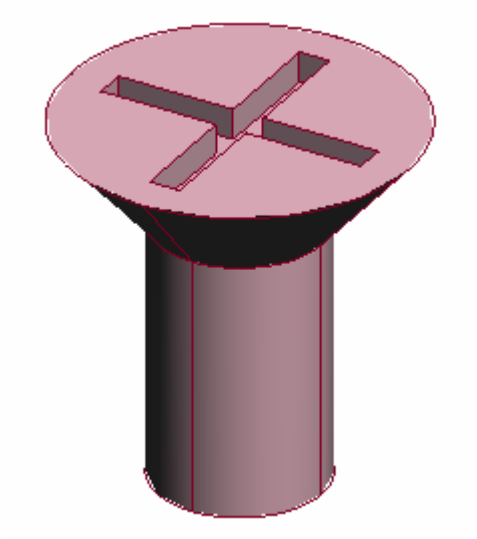
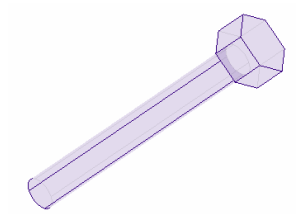
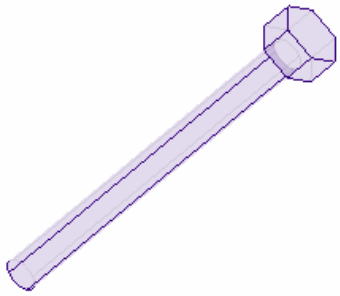


Figure 60. Completed Philips Head.

133. Save this design. Name it Philips Head if you have not already done so.





¼ x 2 and ¼ x 2-3/4 Bolts

134. Open a new design.

135. Draw a ¼" D circle at the origin in the base workplane.

136. Extrude the circle to a distance of 2". Figure 61 shows the resultant cylinder.

137. Select the top face of the cylinder, right click and create a new sketch. Name it Head.

138. Construct a ½" circle at the origin then construct a line from the origin to intersect the upper quadrant of the circle such as shown in figure 62.

Figure 61. Extruded shank of 1/4D bolt

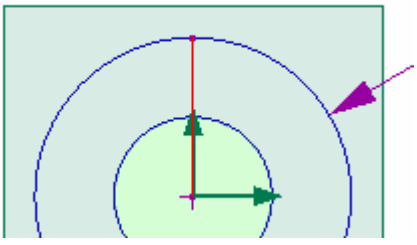


Figure 62. First step to make a hex head.

139. Use the duplicate feature to duplicate the line 6 times around the circle. It should look like figure 63.

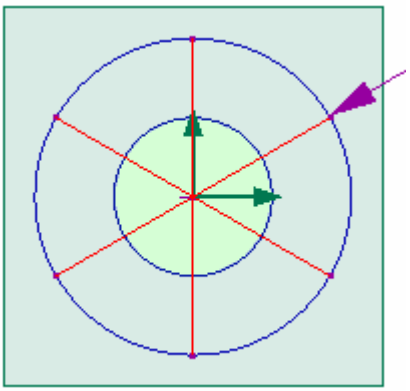


Figure 63. Duplicating 6 lines.

140. Connect the ends of the lines that touch the circle with straight lines then delete all the unneeded lines except the 6 that represent the outline of the hexagon.

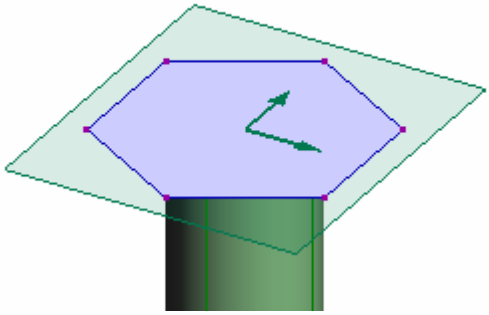


Figure 64. Hexagon profile ready for extrusion.

141. Figure 64 shows what the geometry should look like at this point

142. Extrude this profile by adding material a distance of ¼" above the workplane.

143. Figure 65 shows the completed bolt. Save it and name it .25 x 2 Bolt if you have not done so already.

Figure 65. Completed 2" Hex Head Bolt.

144. There is no need to create the 1/4 x 2-3/4" bolt since the only difference is the length. However, we can modify the length of the 2" bolt and save a copy.

145. With the 2" bolt design still open, (remember to save it) use the browser to select Features as in figure 66.

146. Right click on extrusion 1 and then click redefine. The Extrude Profile dialog box will appear. It will have the Distance of 2..0" for the length of the bolt.

147. Change this distance to 2.75 as shown in figure 67.

148. Press OK to execute the new distance. You may need to click on the green stop light up-date icon in the menu bar to effect the change.

149. This action creates the ¼ x 2.75" long" bolt. You should use Save Copy and give it a different name (.25 x 2.75 Bolt). If you just use save the original 2.0" bolt file will be overridden.

150. Figure 68 is what the new 2.75" bolt should look like.

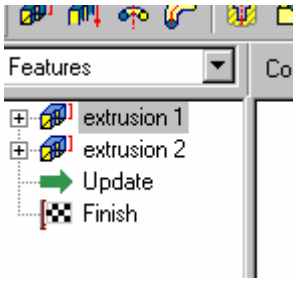


Figure 66. Select Features in Browser.

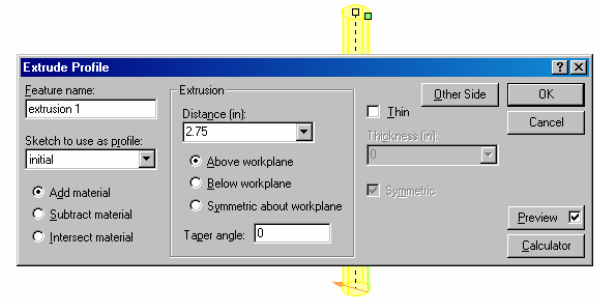


Figure 67. Editing the extrusion distance.

Figure 68. Completed ¼ x 2.75" bolt.

Triangle V Block

151. A triangle shaped block is necessary to increase the support for the fence so it will not bend back when the cam is applied.

152. Open a new design.

153. Select the Lateral workplane, right click and create a new sketch. Name the sketch V Block.

154. Use the dimensions in figure 69 to construct the triangle with the bottom and height both 3.5".

155. Extrude this profile a distance of ¾".

156. Figure 70 shows the finished V Block.

157. Save this design and name it V Block.

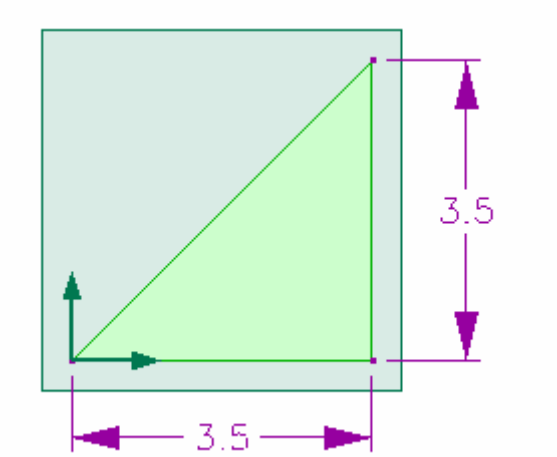


Figure 69. Layout for V Block.

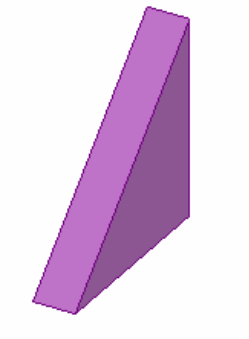
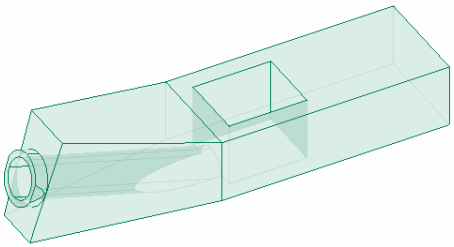


Figure 70. Completed V Block.

Drill Guide



158. The drill guide is a commercial device that is used in conjunction with a special drill bit to drill pocket holes.

159. Open a new design. On the frontal workplane create a new sketch. Name it rear section.

160. Use the dimensions in figure 71 to construct the base 5/8" (0.625") x 2-1/4" rectangle.

161. Extrude this design by adding material below the workplane a distance of 1".

162. Figure 72 shows the extruded solid.

163. Select the left face (red arrow in figure 72) and create a new sketch. Name the sketch Front section.

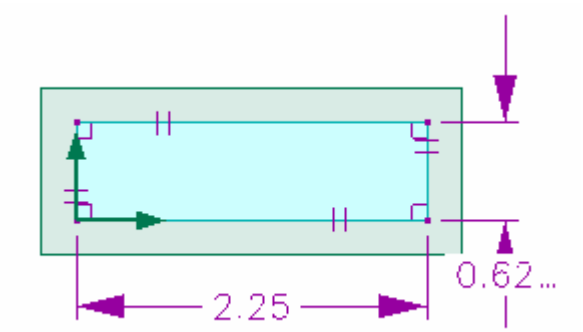


Figure 71. Layout for rear section.

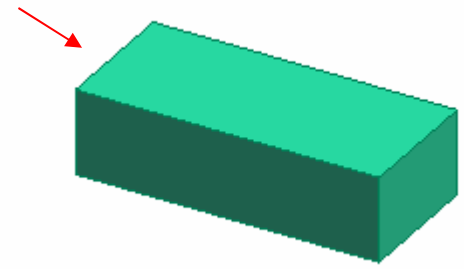


Figure 72. Extruded solid.

164. Create a 1" x 1" rectangle that is even with the sides and bottom of the existing rear section. This is shown in figure 73 where the overhang is at the top.

165. Extrude this rectangle a distance of 1-3/4" above the workplane.

166. Figure 74 shows this extrusion when it is completed.

167. Select the right face (red arrow in figure 74), right click and create a new sketch. Name it Side Profile.

168. Shift-W to see the orthographic view then use the dimensions in figure 75 to construct a triangle for removing material at the left of the block.

169. Use extrude to subtract material below the workplane distance of 1".

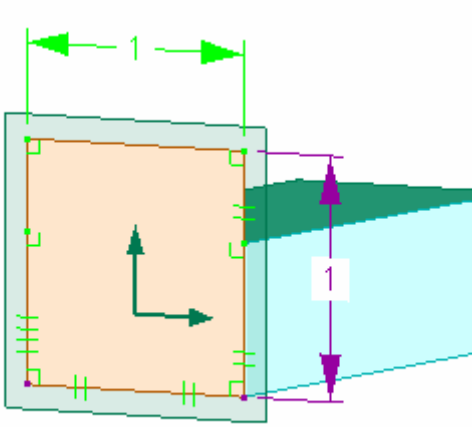


Figure 73. Profile ready for extrusion.

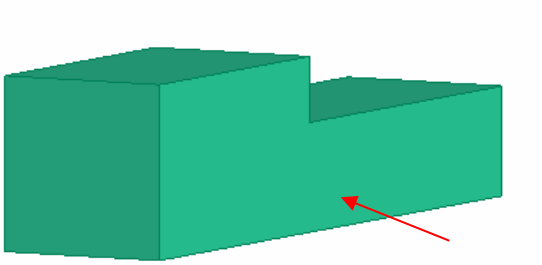


Figure 74. Completed front section

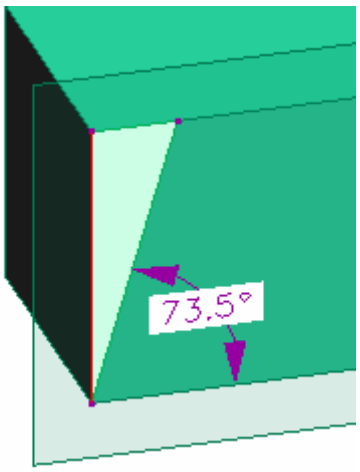


Figure 75. Layout triangles for material removal.

170. Figure 76 shows the resulting solid.

171. Select the slanted front face, right click and create a new sketch. Name it Hole Boss.

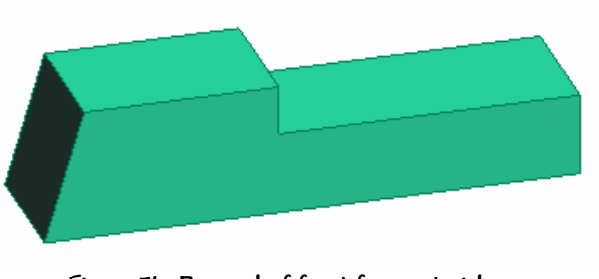


Figure 76. Removal of front face material.

172. Use the dimensions in figure 77 to construct a ½" D circle.

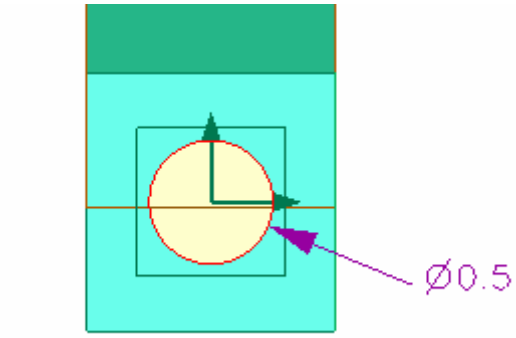


Figure 77. Layout for Hole Boss.

173. Extrude this circle above the workplane, adding material a distance of 1/8".

174. This boss is shown completed in figure 78.

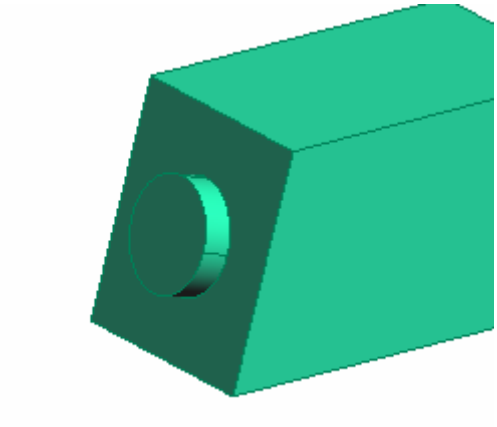


Figure 78. Completed Hole Boss.

175. Select the face of the just completed boss and right click. Create a new sketch and name it Drill Hole.

176. Draw a 0.378" D hole at the origin as shown in figure 79.

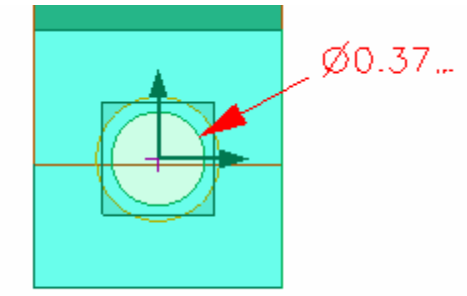


Figure 79. Drill Hole layout.

177. Use Project Profile to subtract material below the workplane and through the entire part.

178. Figure 80 shows the completed hole. The view is transparent so you can see the internal lines.

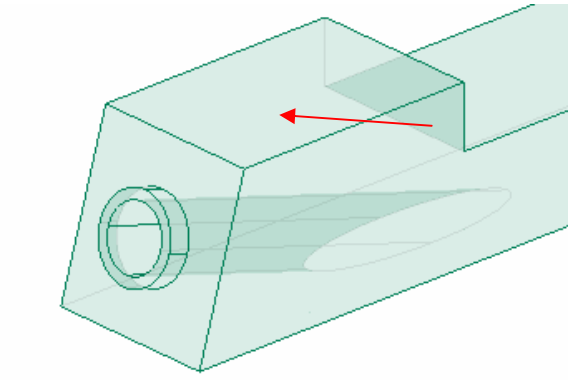


Figure 80. Completed Drill Hole.

179. Select the top of the front section (red arrow points to this face in figure 80), right click and create a new sketch. Name it Vertical Taper.

180. Use the dimensions in figure 81 to create two triangles that will be used to subtract material to create a vertical taper to the front section.

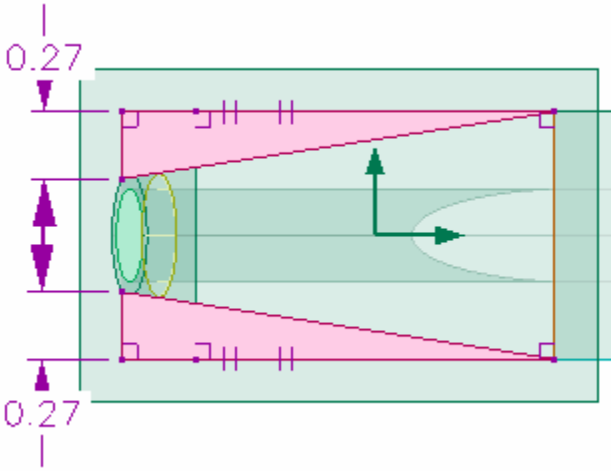


Figure 81. Layout for vertical taper.

181. Use Project Profile to subtract material below the workplane and through the entire part.

182. Figure 82 shows the completed part after material has been subtracted resulting in tapered sides.

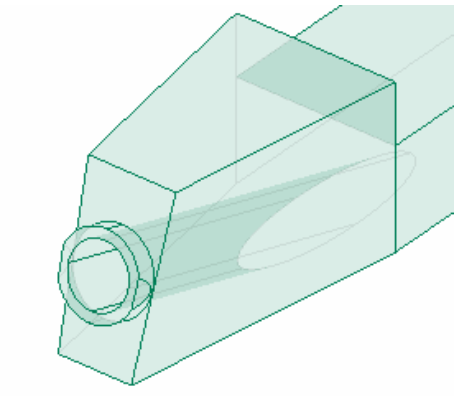


Figure 82. Face selected for waste hole sketch.

183. Select the Lateral workplane in the browser, right click and create a new sketch. Name it top taper. It is necessary to create a

184. Vertical lateral workplane since the sides of the front part are tapered and cannot be used for this projection.

185. Use figure 83 to create a triangle for removing the top material to make a taper. The red arrow points to the triangle. It is filled with color indicating it is a valid profile...

186. Use Project profile to subtract material below the workplane and through the entire part.

187. The finished part with taper is shown in figure 84.

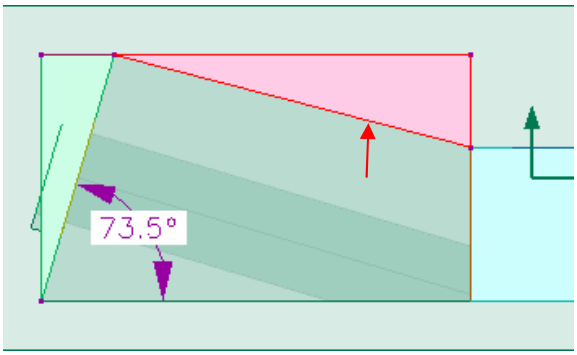


Figure 83. Layout for creating top taper.

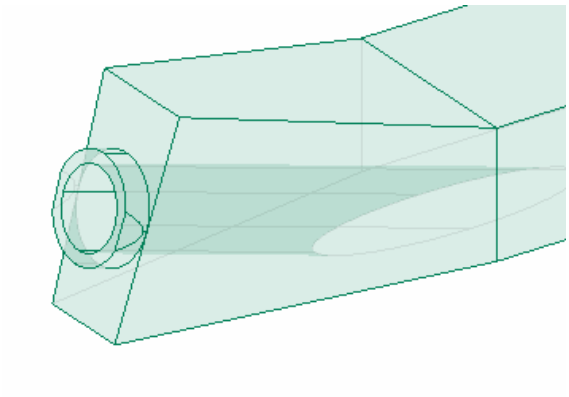


Figure 84. Completed top taper.

188. Select the rear section top face shown in red in figure 85. Right click and create a new sketch. Name the sketch waste hole.

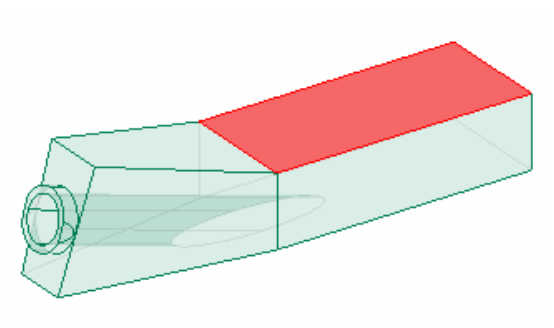


Figure 85. Selected face for waste hole

189. Use the dimensions in figure 86 to create a rectangular opening (0.64" x 0.77") on this sketch. Notice that the rectangle is centered on the face.

190. Use Project Profile to project this rectangle. Subtract material below the workplane and through the entire part.

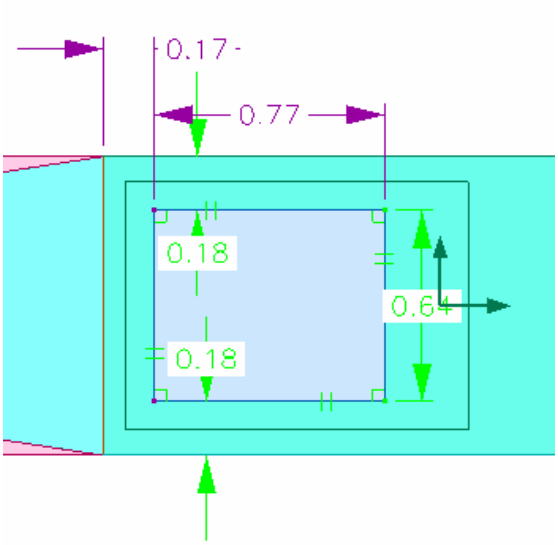


Figure 86. Layout for waste hole.

191. Figure 87 shows the completed part in transparent mode so you can see the internal structure.

192. This completes the drill guide. Save it and name it Drill Guide if you have not already done so.

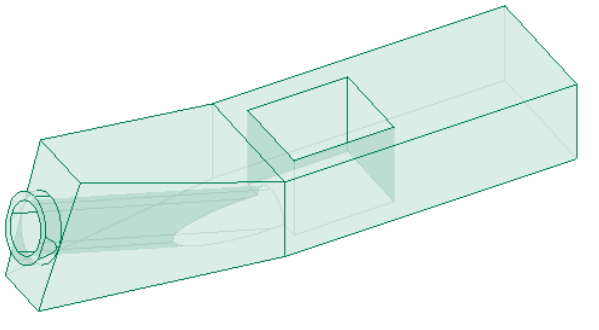
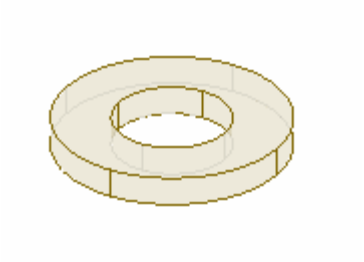


Figure 87. Completed Drill Guide.



Washer

193. Washers are needed to protect the wood where bolts are used to allow for adjustments. In this case we will make one washer and use it a number of times in the assembly.

194. Open a new design.

195. Use the dimensions in figure 88 to draw two concentric circles on the base workplane.

196. Extrude this profile above the workplane a distance of 1/16" (0.0625).

197. Figure 89 is the completed washer. Save it and name it Washer if you have not yet done so.

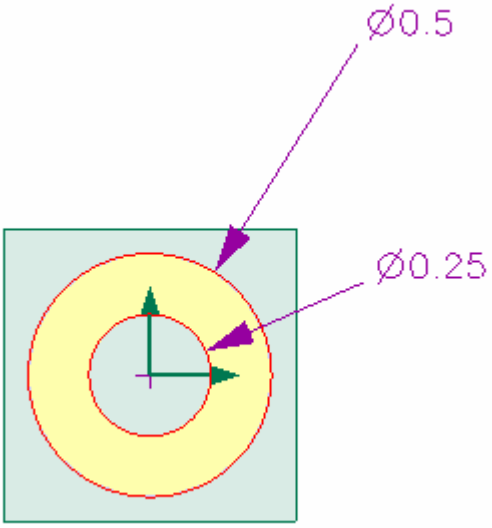
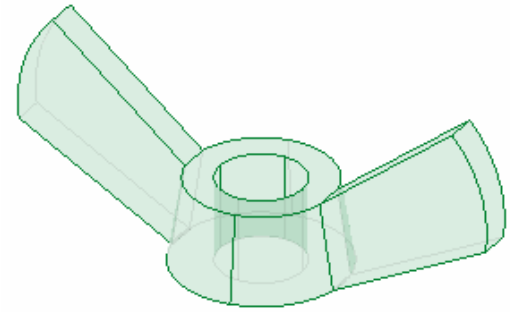


Figure 88. Dimensions for a washer.

Figure 89. Completed Washer.

Wing Nut



198. Open a new design.

199. On the base workplane, draw a ½" D circle centered on the origin.

200. Extrude this profile above the workplane a distance of ¼". Set the taper angle to 9° as shown in figure 90.

201. Press OK to execute the extrusion.

202. Select the top of the extruded hub and right click to create a new sketch. Name it hole. Draw a ¼"D hole at the origin. This is shown in figure 91.

203. Use Project Profile to subtract material, below the workplane and through the entire part.

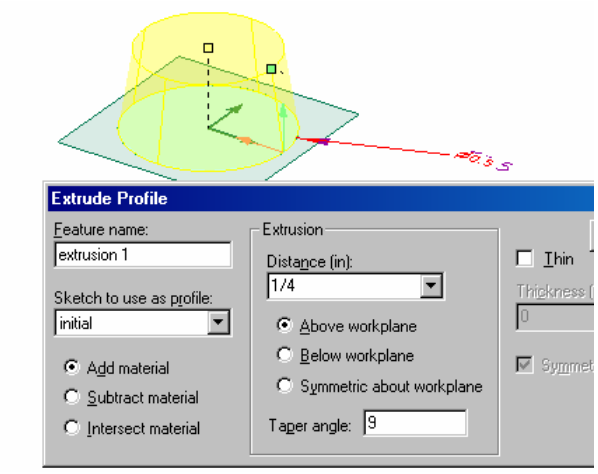


Figure 90. Extruding the tapered hub.

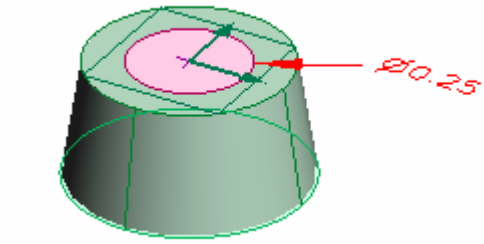


Figure 91. Profile for hole.

204. Figure 92 shows the completed Hub.

205. In the browser, select the Frontal Workplane. Right click and create a new sketch. Name it Wings.

206. Press Shift-W to see the orthographic view of this sketch. Use the dimensions in figure 93 to construct the profile of the right wing. You will need to do this by eye then use constraint dimensions to set the actual value. Notice that the top and bottom lines of the profile have their constraints fixed (in drop down Constraints menu). This is done to keep them from changing when you use the arc or fillet tool to drag the 0.4R curve at the right end of the profile.

207. Draw a temporary line vertically from the origin either up or down. This will be the axis for mirroring the left wing.

208. Select the right wing profile lines then under Line, select mirror. Select the Axis tab then the temporary line from the origin. You will see a trial view of the mirror image to the left. Click OK to execute.

209. Delete the temporary axis line.

210. The wings are now ready to be extruded as shown in figure 94.

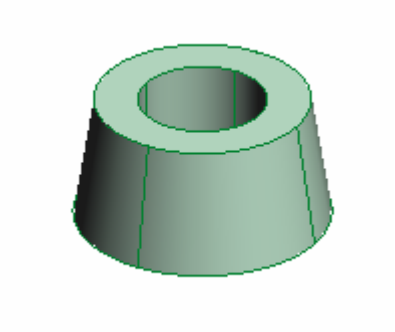


Figure 92. Completed Hub.

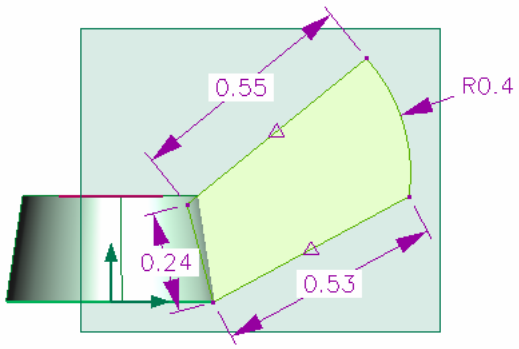


Figure 93. Layout for right wing.

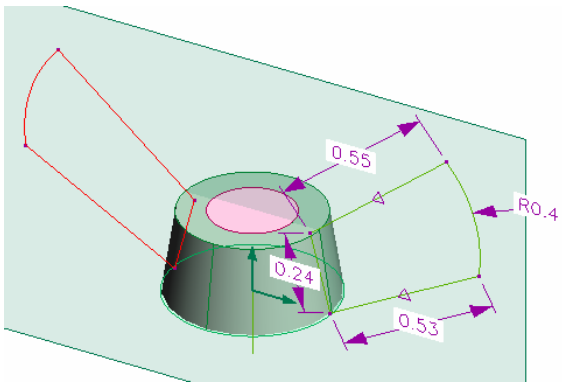


Figure 94. Wings ready for extrusion

211. Since the sketch plane is centered on the hub, it is necessary to extrude on both sides of the workplane.

212. Use Extrude Profile to add material symmetrically, a distance of 3/32" (0.09375") as shown in figure 95.

213. Click OK to execute the extrusion

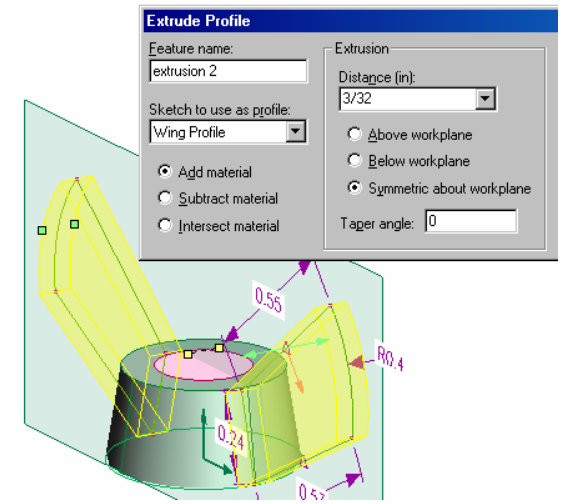


Figure 95. Wings ready for extrusion.

214. Figure 96 shows the completed wing nut. Save this design and name it Wing Nut if you have not already done so.

215. Open a new design and draw a rectangle on the base workplane of 2" wide by 8" long. Extrude this profile a distance of ¾". This will be the work piece into which pocket holes will be drilled.

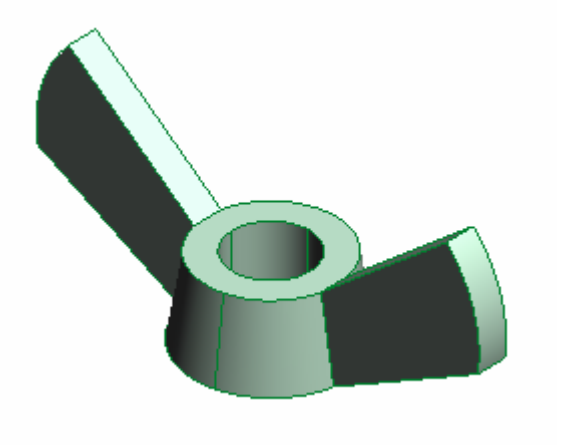


Figure 96. Completed wing nut.

216. Save it and name it Work Piece. Figure 97 shows the finished work piece.

This completes the construction of all the parts for the Pocket Hole Jig. We will now proceed to use these parts to assemble the Jig.

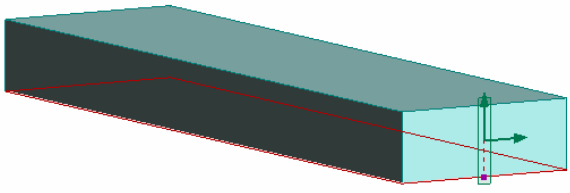
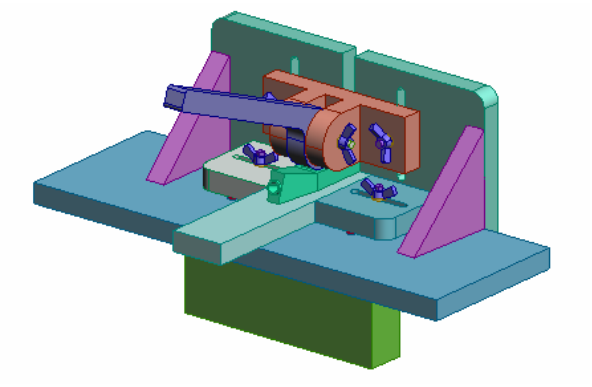


Figure 97. Completed Work Piece.



Pocket Hole Jig Assembly

217. Open a new design. Save it an name it Pocket Hole Jig Assembly.

218. Under the Assembly drop down menu, select Add Component. Browse to the correct location and select the Base

219. While the base is still selected (Red), Under Assembly, click on Fix Components. This will fix the base so that it will not move when adding other parts.

220. Figure 98 shows the base added to the assembly.

221. Use Add Component to add the Fence. It may be located at an inconvenient place but you can select it (part selection) and when it is red, drag it around to a better view. It may help to turn on the transparent view to better see internal lines. Figure 99 shows the assembly at this point.

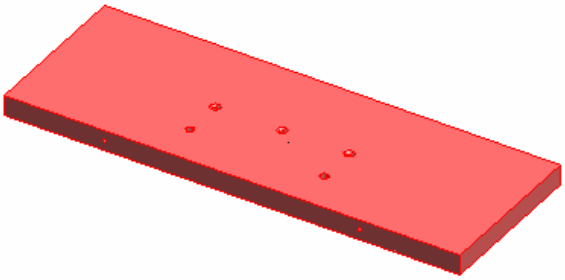


Figure 98. Base added to assembly.

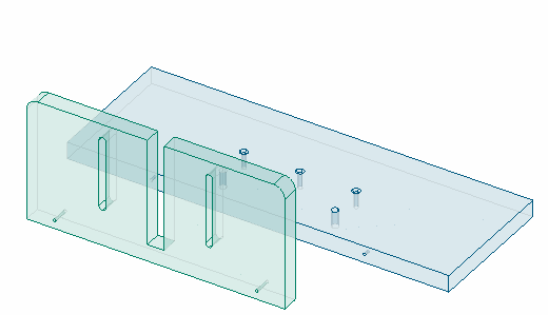


Figure 99. Fence added to assembly

222. Zoom in so that you can see the right mounting holes in the base and fence as in figure 100.

223. Use edge select to select the two end circles of the mounting holes as shown in red in figure 100.

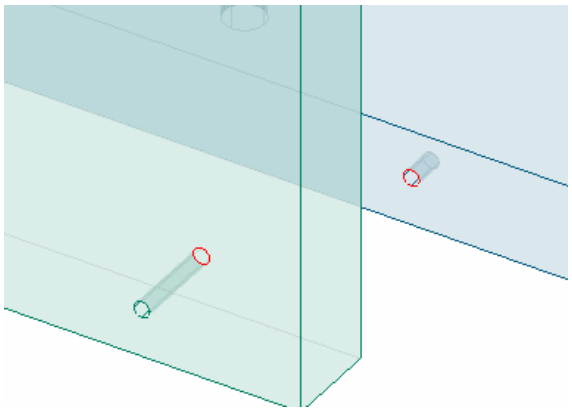


Figure 100. Mounting holes selected.

224. Right click and select Center Axes. The fence should move and the mounting holes should line up as shown in figure 101.

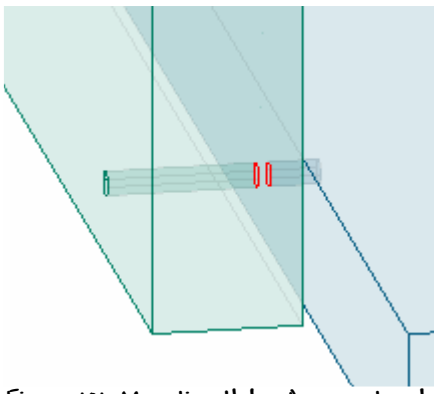


Figure 101. Mounting Holes Axes centered.

225. In a similar fashion, select the two circles of the left mounting holes and Center Axes as well. The fence should now be lined up with the back of the base.

226. Rotate to view the bottom so you can see the mating lines of the fence and base. Use edge selection to select these two lines as shown in red in figure 102.

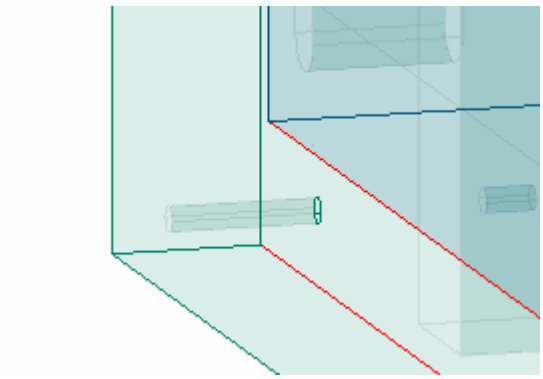


Figure 102. Fence and Base Lines selected.

227. Right click and select Center Axes. The fence should move and be correctly attached to the base as shown in figure 103. Fix the fence so it will not subsequently move.

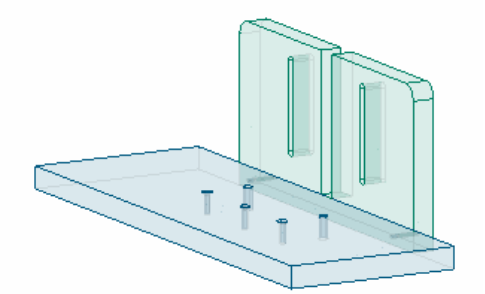


Figure 103. Fence and Base correctly assembled.

228. Use Add Component to add the Cam Holder.

229. The cam holder may need to be rotated if it is facing the wrong way. To do this select the part (it turns red) then right click and select Transform... In the dialog box for transform, select the rotate button and enter 180 in the angle box. Press OK and the part will rotate to the correct orientation as in figure 104.

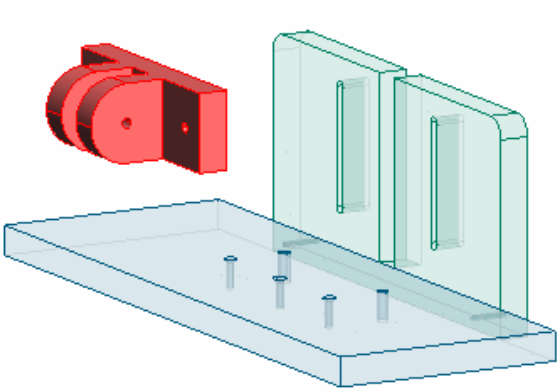


Figure 104. Correctly orientated cam holder.

230. Zoom in to view the right slot in the fence and the corresponding hole in the cam holder. You may need to move the cam holder in order to get them close as in figure 105.

231. Use edge selection to select the top arc of the slot and the corresponding hole in the cam holder as shown in red in figure 105.

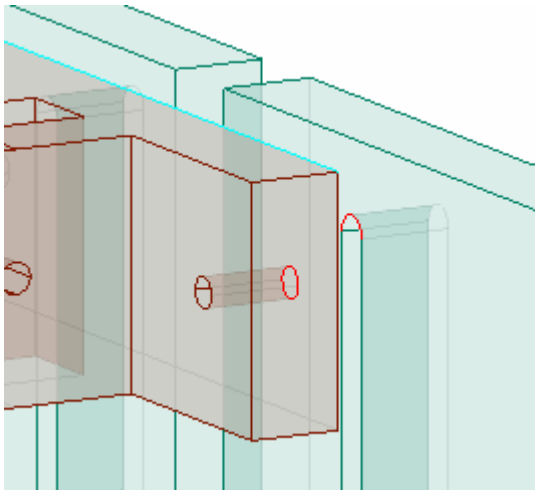


Figure 105. Slot and bolt hole selected for centering.

232. Right click and select Center Axes.

233. Do the same for the left slot and hole so their axes are centered.

234. Both holes should now be in alignment.

235. In the browser, select Components then at the bottom of the browser select Constraints. Right click the two center axes constraints and suppress them. This will allow the next step of mating.

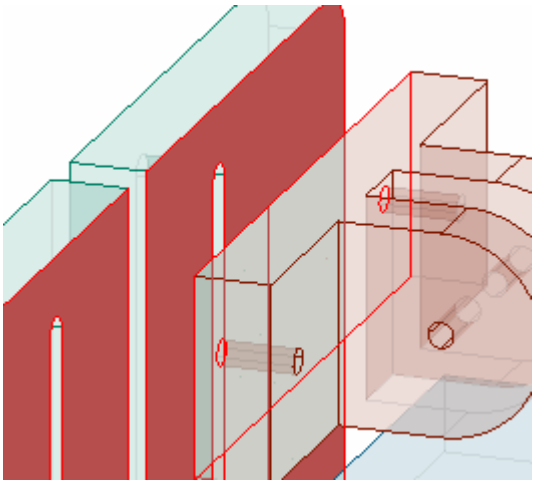


Figure 106. Faces selected for mating

236. Use face selection to select the rear face of the cam holder and the front face of the fence. They both should turn red as in figure 106.

237. Right click and select mate. The two parts will come together as shown in figure 107.

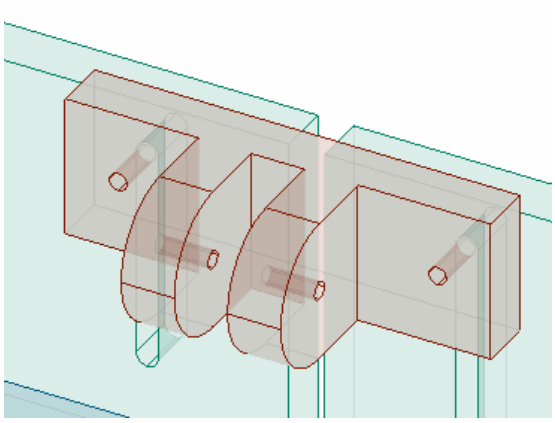


Figure 107. Cam holder and fence mated.

238. You can unsuppress the two center axes constraints if you wish.

239. Use part select and select the Cam Holder. Under constraints, select Toggle Fix so that this part will not move during the next step.

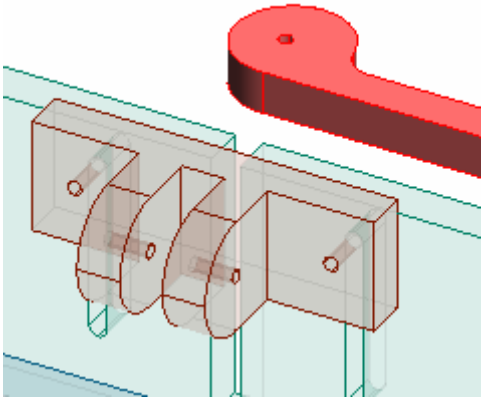


Figure 108. Cam lever added to assembly.

240. Use Add Component and add the cam lever. Position the lever near the cam holder such as in figure 108

241. Zoom in so you can see the center hole of the cam lever and the horizontal holes in the cam holder. This selection is shown by the arrows pointing to the red circles in figure 109.

242. Right click and select Center Axes. The Cam Lever will flip and align with the holder holes.

243. Use Face Select to select the inside surface of the left holder arm and the left face of the cam lever.

244. Right click and select mate. The cam lever should now be correctly aligned within the holder arms. Use part select to select the cam lever and it will rotate as you drag it from end of the handle. Position it similar to figure 110.

245. Use Add Component and add a Washer. Position the washer near the left arm of the cam holder as in figure 111.

246. Use edge selection to select the center circle of the washer and the circle of the hole in the cam arm.

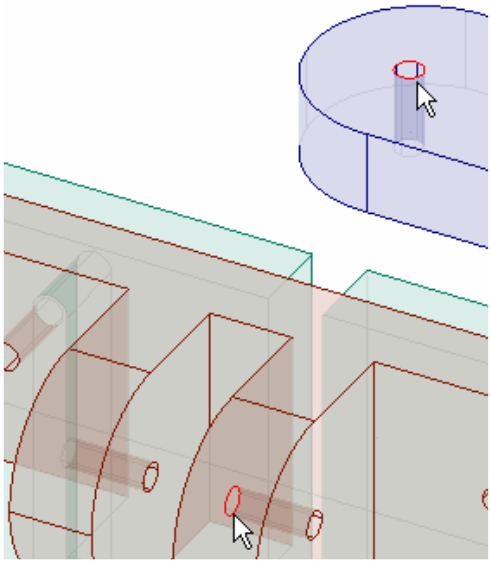


Figure 109. Edge selection to center axes.

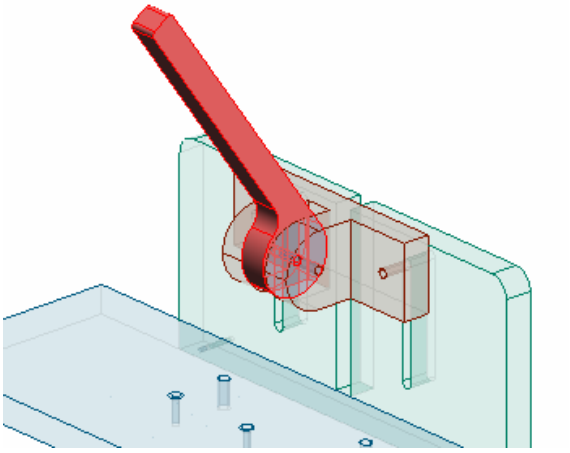


Figure 110. Cam Lever Installed and positioned.

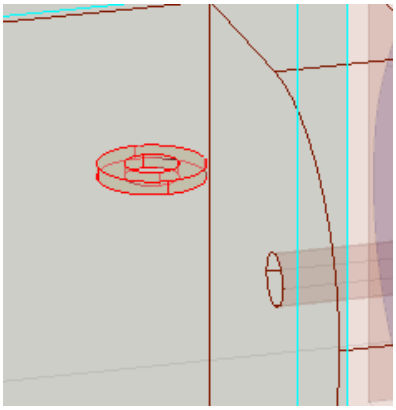


Figure 111. Washer added and positioned.

247. Right click and select Center Axes. The washer will align with the hole. You may need to move the washer if it moves to the interior of the cam.

248. Select the two mating faces - one on the washer and one on the cam holder arm.

249. Right click and select mate. The washer will now be correctly installed as in figure 112.

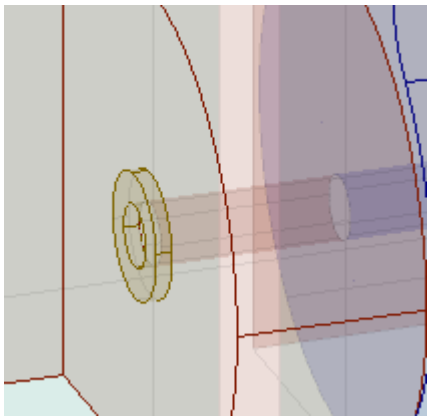


Figure 112. Washer correctly installed.

250. Use Add Component to add the 2-3/4" bolt. It should be moved to a position similar to figure 113.

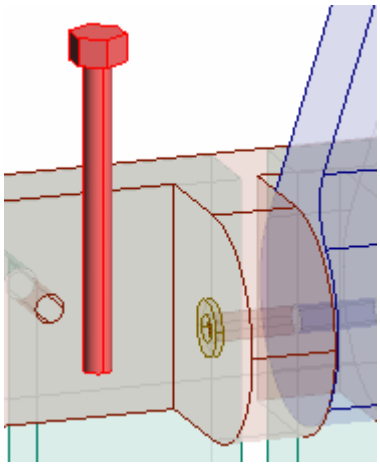


Figure 113. Adding the 2-3/4" bolt.

251. Rotate the view and use edge select to select the bottom circle of the bolt and the corresponding circle in the hole through the left arm of the cam holder. These are shown in figure 114 in red.

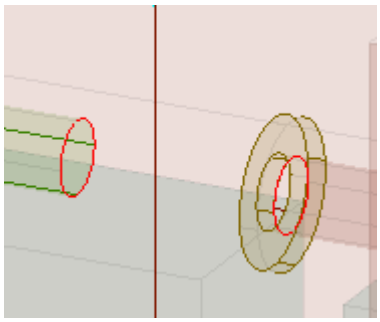


Figure 114. Selecting Bolt and holder circles.

252. Right click and select Center Axes. The bolt will flip and align with the hole in the cam holder. If it faces the wrong way use Transform to rotate it 180°.

253. Use face selection to select the under surface area of the bolt head and its corresponding mating surface on the outside of the washer. This is shown in figure 115.

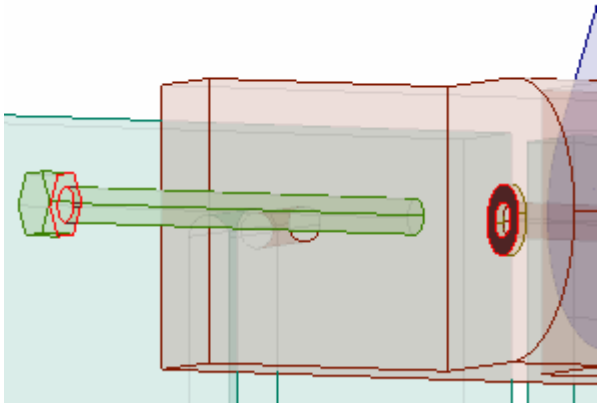


Figure 115. Surfaces selected to mate bolt to holder arm.

254. Right click and select Mate. The bolt will now be inserted correctly through the cam arms and lever.

255. This mating is shown in figure 116.

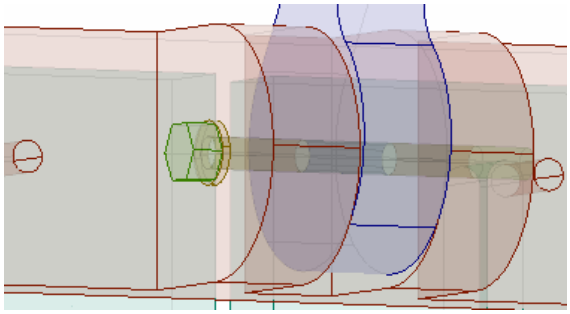


Figure 116. Bolt inserted.

256. Add another washer and using the same procedure as before, center it to the right arm hole and face. It may be easier to just select the existing washer and duplicate it.

257. Add a Wing Nut and Center Axes on the protruding part of the 2-3/4" bolt. Select the washer surface and the under surface of the Wing Nut and right click then select Mate. The added washer and Wing Nut are shown in figure 117.

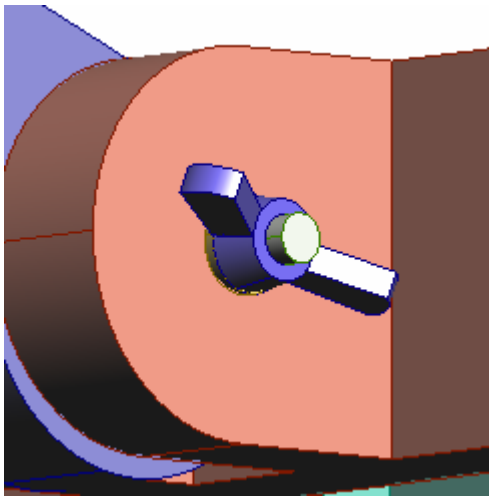


Figure 117. Washer and Wing Nut added.

258. Add a Philips Head screw to the assembly. Move it to near the center countersunk hole on the base as shown in figure 118.

Figure 118. Adding a Philips Head Screw.

259. Use edge select to select the center circle of the hole and the shank circle of the Screw. Right click and select Center Axes.

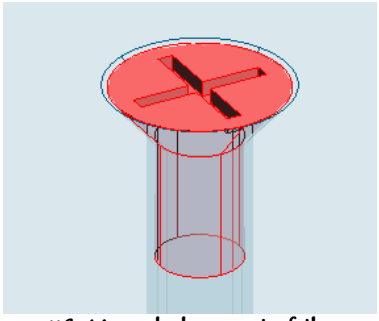


Figure 119. Manual placement of the screw.

260. Zoom in on the screw area and while the screw is selected drag it until it fits just below the surface of the countersunk hole. This is shown in figure 119.

261. Rather than going through the previous process to install the other screws, duplicate the one just installed.

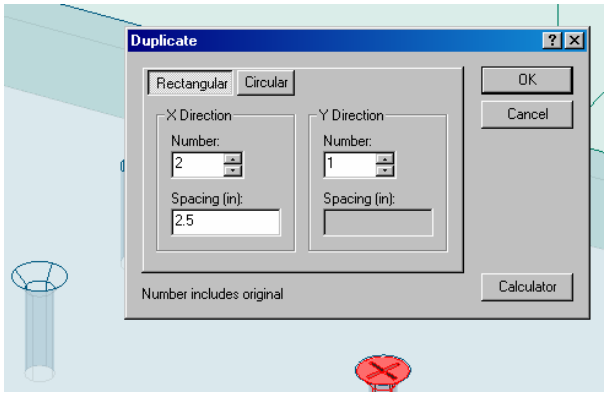


Figure 120. Duplicating the Philips Head Screw.

262. Use part select to select the screw just installed. Under Edit, select Duplicate. Set the duplicate dialog box as in figure 120. Notice the spacing is set to 2.5" so the duplicate will be placed in the right place in the left hand hole.

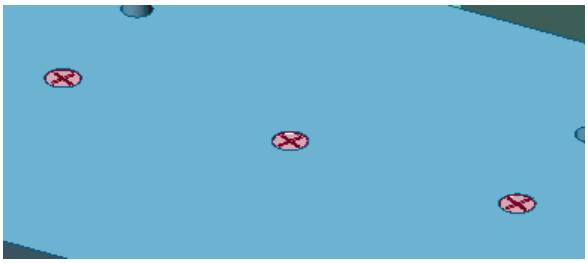


Figure 121. All three Philips Head Screws installed.

263. In a similar fashion, duplicate the center Philips head screw only this time make the spacing -2.5". This will place the duplicate in the right side hole. Figure 121 shows all three screws installed.

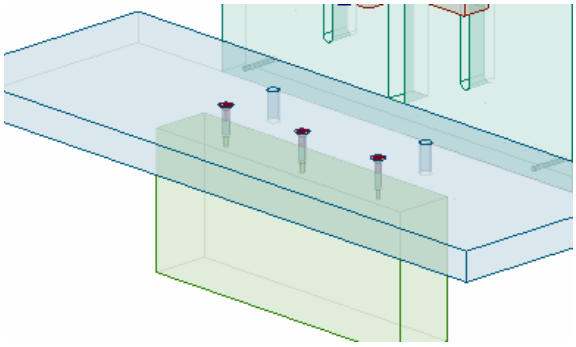


Figure 122. Vice clamp block attached to base.

264. Add the Clamp Block to the assembly. Position it below the base. Edge select the left hand hole on the block and the bottom circle of the hole in the base. Right click and select Center Axes.

265. If necessary, use this procedure to Center Axes with the screw and hole on the right side.

266. Use face select to select the underside face of the base and the top face of the clamp block. Right click and select mate. The clamp block will mate with the base as shown in figure 122.

267. View the back of the unit where the base and fence come together. Using the procedure to install the Philips Head screw, install one screw then duplicate the other which is 9.5" away. The completed installation should look like figure 123.

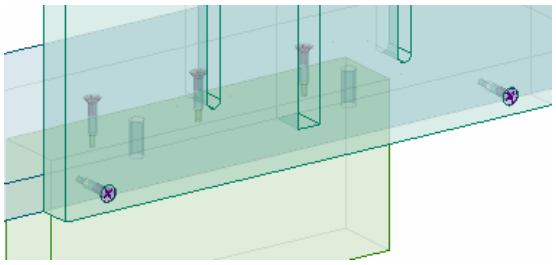


Figure 123. Rear fence mounting screws installed.

268. Add the work piece. Rotate it to the correct orientation and center Axes and mate it to the base.

269. In a similar fashion, add the Drill guide to the top rear of the work piece. You will need to center axes and move the piece by eye to center it under the cam.

270. Suppress the fixed constraint on the cam holder then move it so that the cam surface just touches the top of the drill guide and is centered on the work piece. Figure 124 shows how this should look.

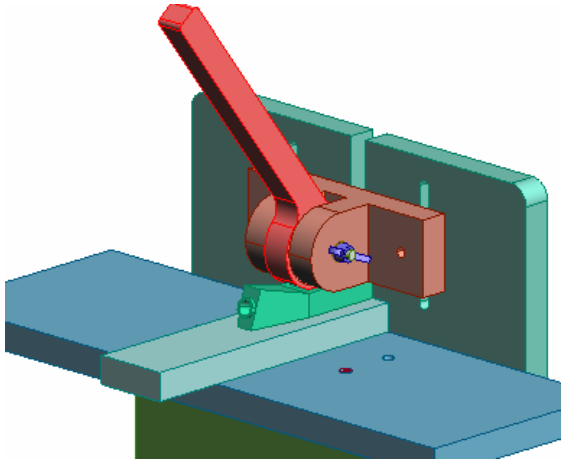


Figure 124. Work piece and drill guide installed.

271. Using the procedures you used to install the bolt, washers and wing nut to the cam and holder, add the shorter bolts (2") through the fence slots and the corresponding holes in the cam holder. Add the washers first, then the bolts and finally the wing nuts.

272. Figure 125 shows the design with these bolts installed. You will need to frequently suppress or delete some constraints in order to apply others. As you move components to their final location. Be sure to Fix Components so that subsequent parts do not move previously added parts.

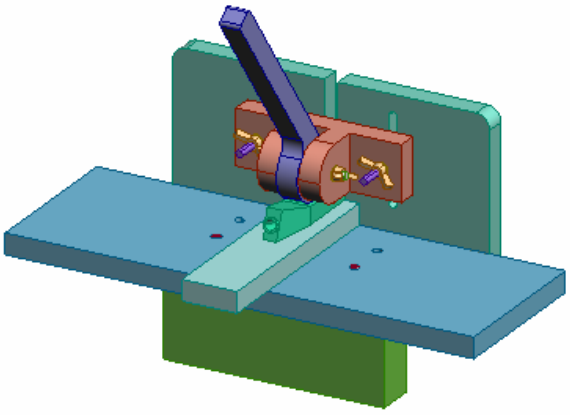


Figure 125. Cam Holder Bolts and Wing Nuts Installed.

273. Add the right adjustable Right Guide Block. You may need to rotate it for proper orientation. Orient it near the mounting slot and use center axes and mating to mate it to the work piece and the base. Figure 126 shows the installed right guide block.

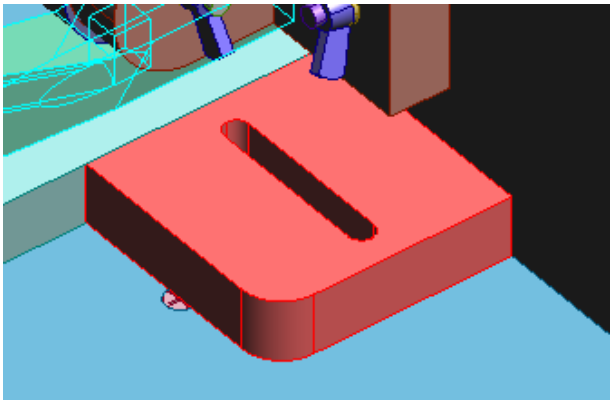


Figure 126. Right Guide Block installed.

274. In a similar fashion, add the left guide block. You may need to rotate the guide block in order to orient it correctly. Figure 127 shows the left guide block installed.

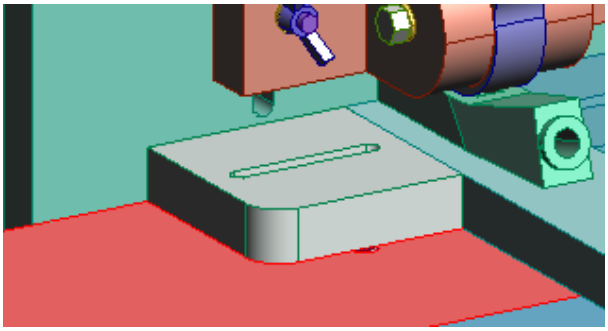


Figure 127. Left Guide block installed.

275. Add washers and 2" bolts through the base and the two guide blocks. Install the bottom washers first then the bolts, then the top washers and finally the wing nuts.

276. Figure 128 shows the bolts and wing nuts installed.

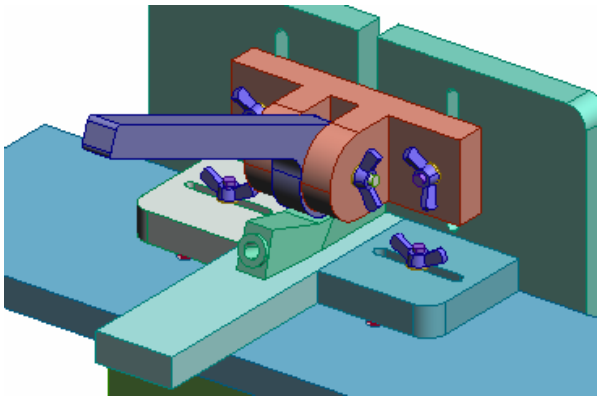


Figure 128. Bolts, washers and Wind Nuts installed.

277. The last components to install are the corner braces or triangle blocks. Add a triangle block to the assembly. You may need to rotate it for correct orientation.

278. Use center axes to align the edge of the block with the edge of the fence and mate the base of the triangle block to the base.

279. Figure 129 shows the right triangle block installed.

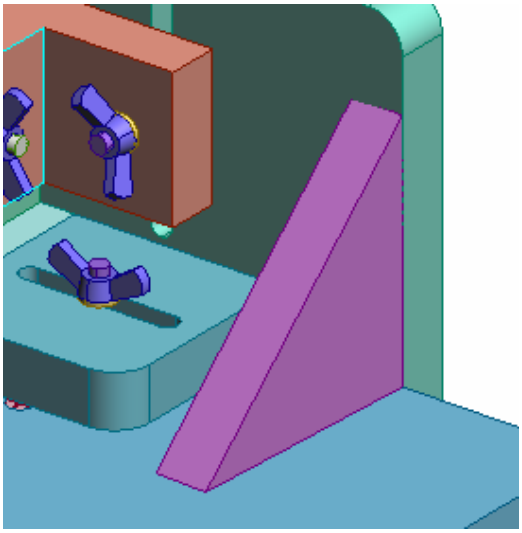


Figure 129. Right Triangle Block installed.

280. In a similar fashion, add the left triangle block at the left edge of the fence.

281. Figure 130 shows the assembly at this point with all components installed. Save this design and name it Pocket Hole Jig Assembly if you have not yet done so.

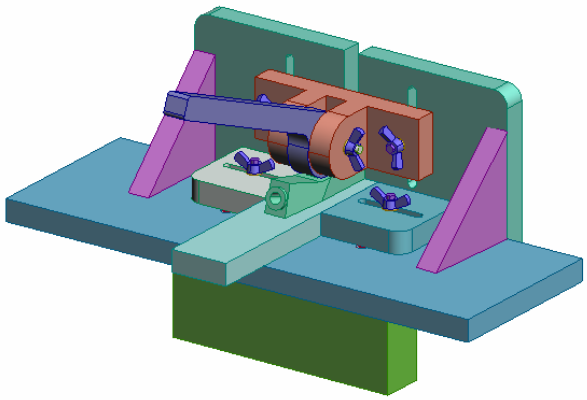
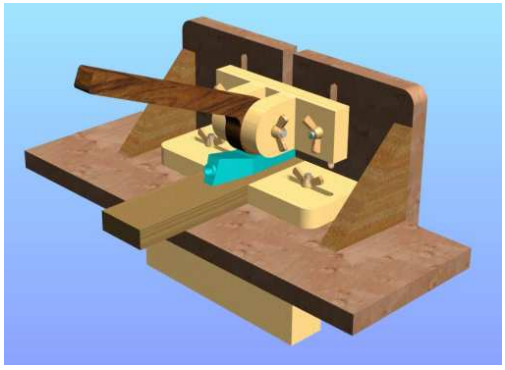


Figure 130. Completed Assembly.



Album

282. This part of the procedure is designed to make an Album or rendering of the completed Jig. You will add material to various parts and export a graphic image of the finished product.

283. From the File menu, select New then Photo Album. A new Window will open.

284. Under the Image drop down menu, select New Image. A dialog box will open with all Pro/DESKTOP files that are currently open. If no other designs are open, you will need to navigate to the Pocket Hole Assembly design and open it. If it is already open, it will be listed in the dialog box and you need only select it and click on OK.

285. Figure 131 shows the image that has been loaded.

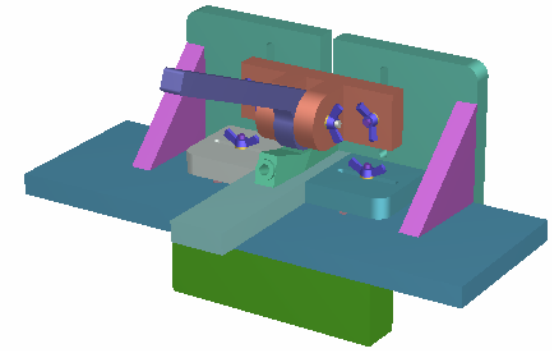


Figure 131. Album image.

286. In the browser window, click on the box down arrow and select Materials as in figure 132.

287. By clicking on the Plus sign in front of the category, the category will expand to reveal a large number of specific materials.

288. Notice a "Bag" to the left of each material. If you click and drag this bag it will turn into Container with something spilling out of it. When you pass over any component, the component will highlight and if you release the mouse button at that point the material in question will be poured into the component.

289. Select Polished Brass and pour it into one of the Wing Nuts. Notice that the up-date icon at the top will turn green. This means that you must up-date the image by clicking on the green icon. Do so and the Wing Nuts will turn to a brass color. Since there is only one design for the wing nut, all copies will take on the material.

290. Figure 133 shows the wing nuts after up-dating.

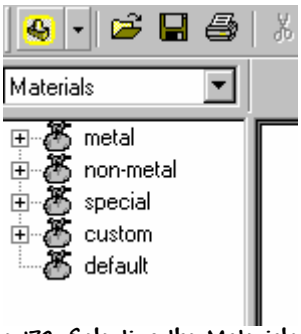


Figure 132. Selecting the Materials list.]

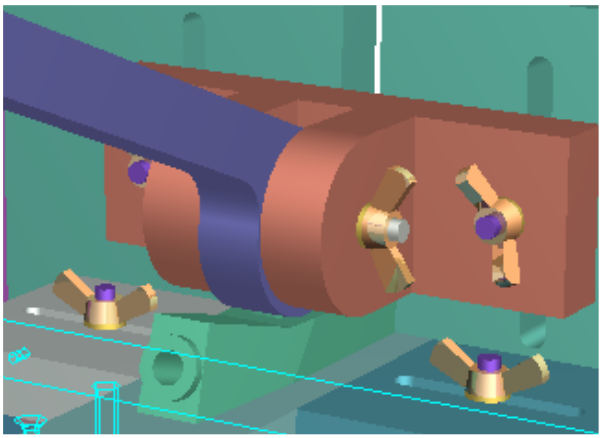


Figure 133. Wing Nuts of Polished Brass.

291. Use various woods and plastics to add material to all the components of the image. Brass is a good choice for the Philips Head screws and Stainless Steel shows up well for the bolts.

292. Figure 134 is one example of applying material to the various components of the image.

293. You can also change the background and foreground (figure 134 has a graduated scheme from light blue at the top (top color) to dark blue at the bottom (bottom color).

294. These features can be accessed by selecting Image then Image properties from the drop down menu. A dialog box will be available with tabs for Image, effects and Studio. Each of these will cause different effects. You are able to control studio light as well as camera lens and the quality of the image.

295. You can also add any bitmap designs to the list (.bmp) by downloading them into the Pro/DESKTOP Bitmap folder that is located in Program Files/PTC/PRODESKTOP 8/Program/Bitmaps. There are a number of internet sites that have free background or material images but remember to save them as .bmp files. Remember the jpeg images will not be usable.

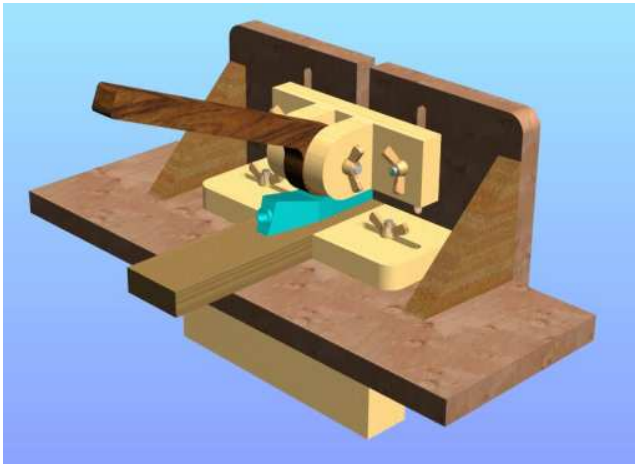


Figure 134. Completed Album Image.

296. Figure 135 shows an image with the background of leaves. This is done by selecting Custom under the Image background and browsing to select the custom leave bitmap. You can acquire the leaves.bmp file at: http://www.backgroundsarchive.com/index.php. This site has a number of other bmp files that you may want to examine and download. Experiment with changing all the attributes of the image.

297. You can save your image as a jpeg or bitmap file using the export command under File.

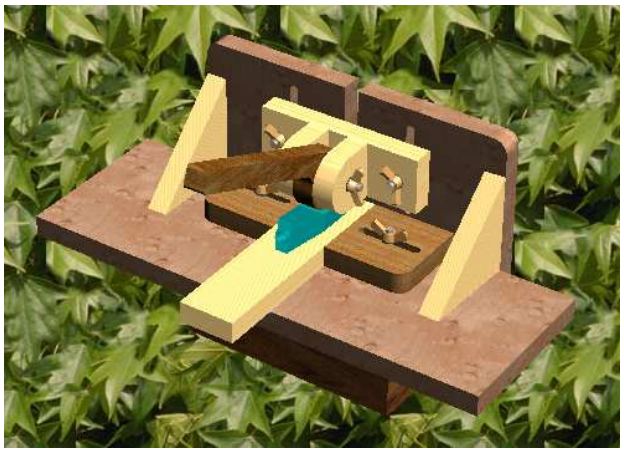
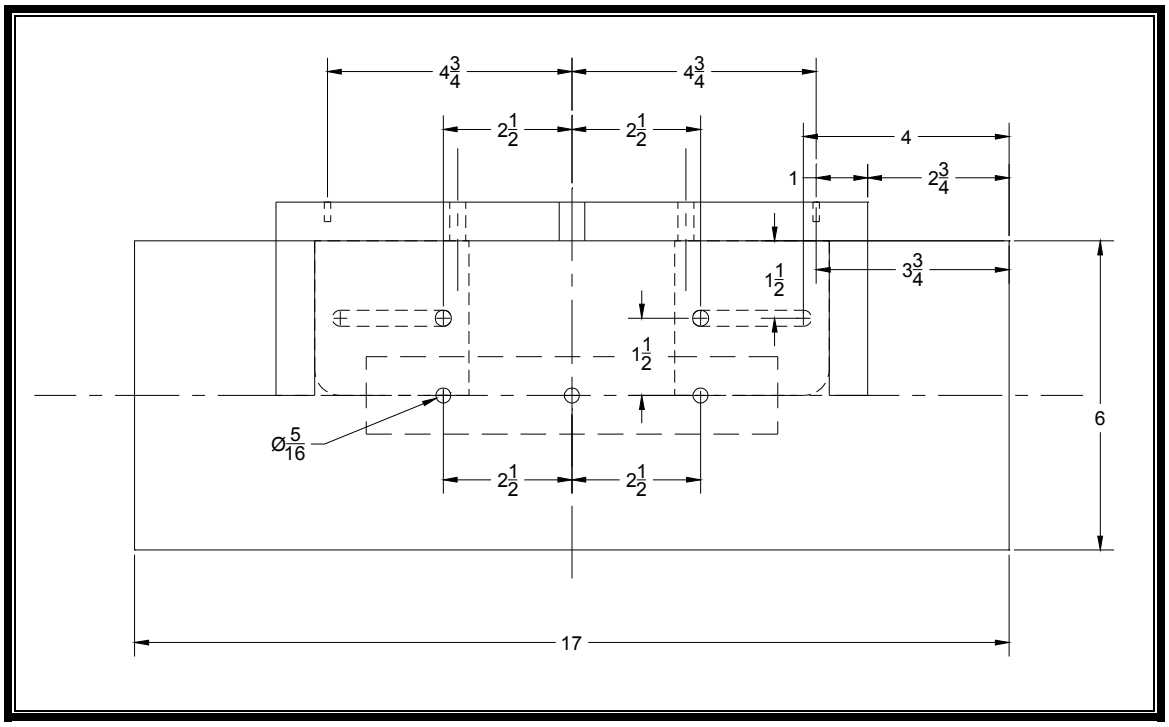
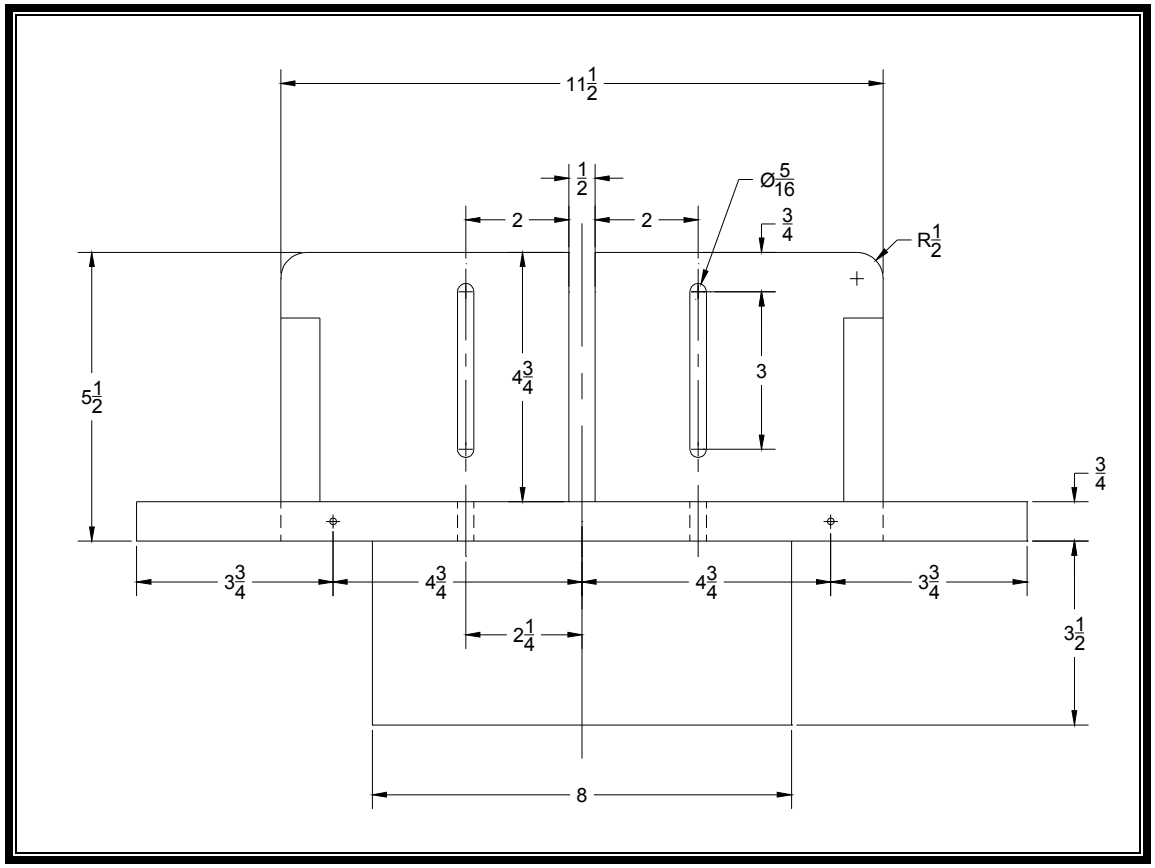


Figure 135. Album Image with Leaves background.

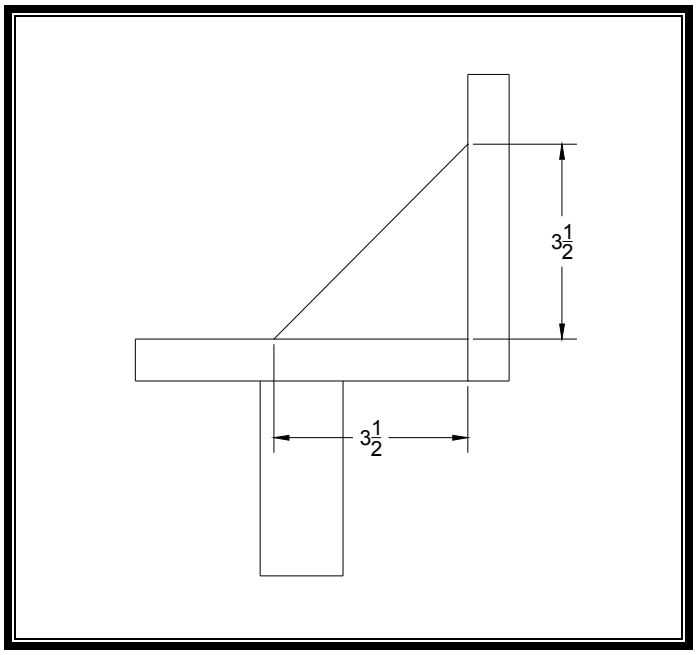
Note: The following pages contain dimensioned orthographic drawings of most of the parts used in this design. Refer to them for accurate values while developing the various designs.



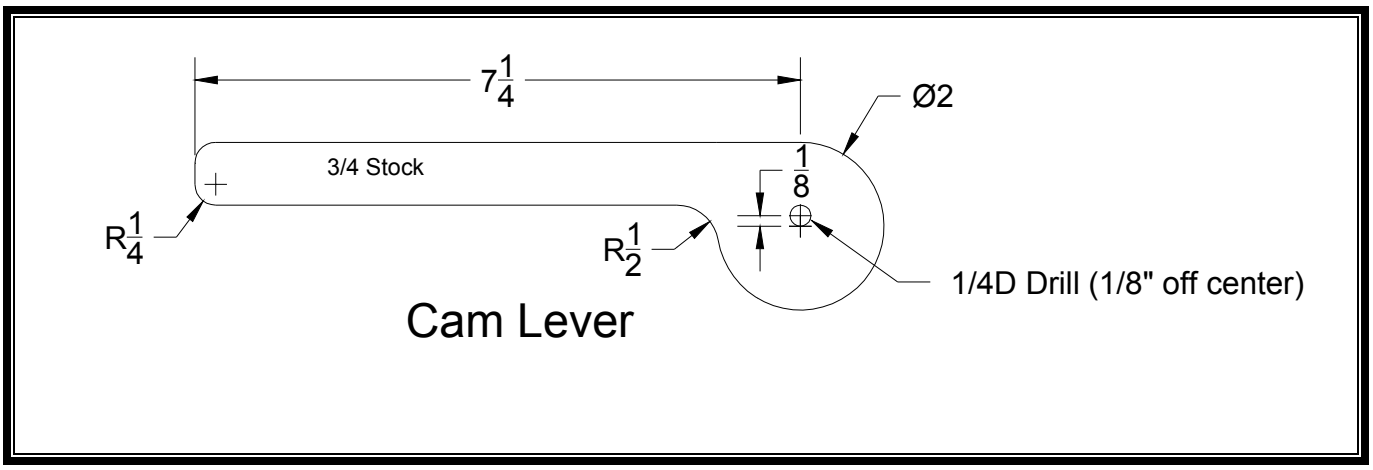
Top View



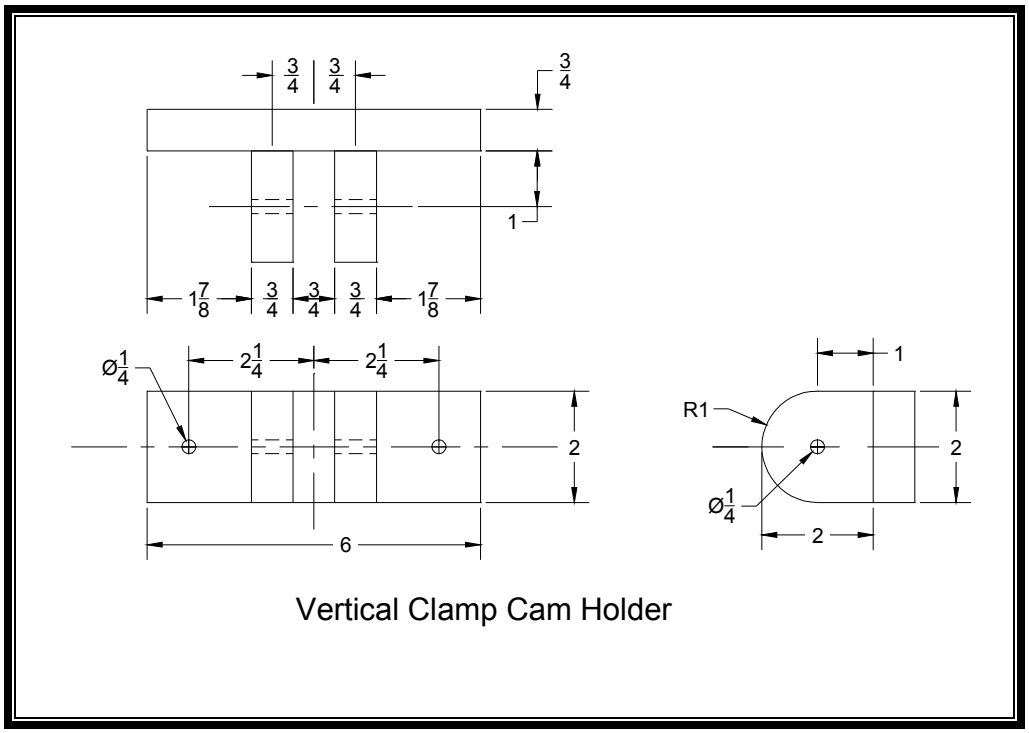
Front View



Right Side View

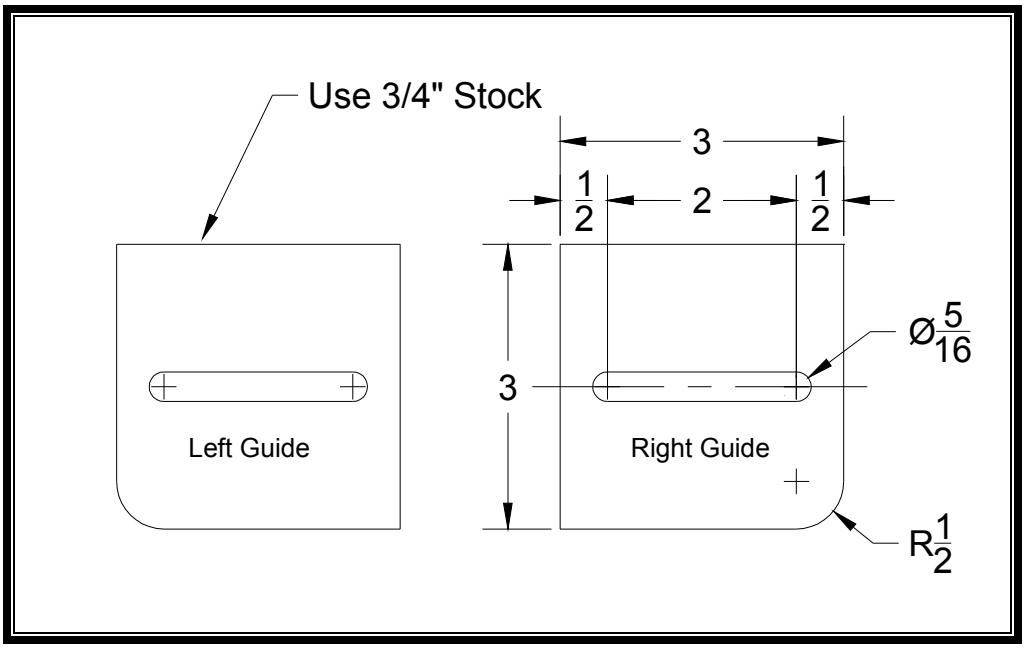


Cam Lever Dimensions



Vertical Clamp Cam Holder

Cam Holder Views



Left and Right Guide Blocks.