71. Extrude the sketch into a solid with a thickness of .25 as shown in Figure 62.

Figure 62

72. Save the part as Crankshaft1.ipt where it can be easily retrieved later.

73. Begin a new drawing as described in Chapter 1.

74. Create a new sketch as shown. Make sure each of the circles are NOT sharing the same center and are NOT in line with each other as shown in Figure 63.

Figure 63
75. Move the cursor to the upper middle portion of the screen and left click on the Concentric constraint icon as shown in Figure 64.

**Figure 64**

76. Holding the Shift key down, left click on each circle as shown in Figure 65.

**Figure 65**

77. Inventor will create concentric circles as shown in Figure 66.

**Figure 66**
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78. Move the cursor to the upper middle portion of the screen and left click on the Vertical constraint icon as shown in Figure 67.

![Figure 67](image)

79. Left click on the centers of the circles as shown in Figure 68.

![Figure 68](image)

80. Inventor will create a vertical constraint between the centers of the circles as shown in Figure 69.

![Figure 69](image)
81. Use the Free Orbit/Rotate command to rotate the sketch around on to its side as shown in Figure 70.

**Figure 70**

![Figure 70](image)

82. Using the geometry created above, complete the sketch shown. Extrude the sketch to a thickness of 0.25 inches as shown in Figure 71.

**Figure 71**

![Figure 71](image)

83. Save the part as Conrod1.ipt where it can be easily retrieved later.

84. All of these parts will be used in the next chapter.