58. Use the Extrude command to extrude or cut out the circle that was just completed. Change the view to Isometric as shown in Figure 51.

Figure 51

59. Save the part as Pistoncase1.ipt where it can be easily retrieved later.

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

61. Begin a sketch as shown in Figure 52. Make sure that the circles are located on the endpoint of a line. Also make sure the circles are NOT the same diameter.

Figure 52

62. Move the cursor to the upper middle portion of the screen and left click on the Equal constraint icon as shown in Figure 53.

Figure 53
63. Left click on each of the circles as shown in Figure 54.

**Figure 54**

64. Inventor will create two circles of the same size as shown in Figure 55. When one circle is dimensioned, Inventor will automatically update the size of the other circle.

**Figure 55**
65. Finish completing the sketch shown in Figure 56.

Figure 56

66. Extrude the sketch into a solid with a thickness of 0.25 as shown in Figure 57.

Figure 57
67. Complete the following sketch. Use the center of the outside fillet radius as the center of the circle as shown in Figure 58.

Figure 58

68. Extrude the sketch into a solid with a thickness of .25 as shown in Figure 59.

Figure 59
69. Rotate the part around to gain access to the opposite side as shown in Figure 60.

Figure 60

70. Complete the following sketch as shown in Figure 61.

Figure 61
71. Extrude the sketch into a solid with a thickness of 0.25 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