1. Go to Manage→Projects.

2. Set the tutorial_files as your active project.

3. Browse to the Weldments folder.

4. On the Assembly ribbon:

   Select Convert to Weldment.

5. Select Yes.
6. Set the Standard to **ANSI**.

   Set the Weld Bead Material to **Welded Steel Mild**.

   Press **OK**.

7. The feature browser updates to show three types of operations:

   - Preparations, Welds, and Machining.

8. **Highlight Welds.**

   Right click and select **Edit**.

   You can create three types of weld features: fillet, groove, and cosmetic welds.

   *Cosmetic* weld features, the preferred type, are represented by graphical elements. You can represent a wide variety of weld beads as cosmetic welds, including fillet welds and various groove welds.

   Weld preparations are not required for cosmetic welds. The weld symbol contains the weld preparation required for the selected edges.

9. **On the ribbon:**

   Select **Weld ➔ Cosmetic**.
10. Enable Chain.
Select the lower edge of the brace.

11. Enable Create Welding Symbol.

12. Click inside the lower weld symbol box.
Select the symbol for Bevel Weld.

13. Fill in the symbols.
Set the depth to 6 mm.
Type G for Grind.
Set to Flat.
Press OK.

14. An annotation is attached to the edge.
15. Right click on the Beads folder in the browser.

Select **Finish Edit**.
16. Change the view using the view cube.

17. On the ribbon:
Select **Weld ➔ Fillet**.

18. You need to make two selections of the two faces/edges to be welded together.

19. Select the side face of the brace.
   Click on the 2 button.
   Select the cylindrical face of the button.

20. Set the size to 6 mm x 6.
21. Enable **Create Welding Symbol**.

22. Fill in the values for the symbol.

Press **OK**.

23. The weld is applied.

24. Right click on the Welds category in the browser.

Uncheck **Symbol Visibility** to turn off the weld annotations.

25. Right click on the Beads folder in the browser.

Select **Finish Edit**.
26. 
On the ribbon:

Go to Process ➔ Machining.

27. 
On the ribbon:

Select Hole from the Preparation and Machining panel.

28. 
Set the Placement to Concentric.
Select the top face of the boss.
Select the edge of the boss for the Concentric Reference.
Set the diameter to 30 mm.
Set the Termination to Through All.

Press OK.

29. 
Note that the Hole appears under Machining.

30. 
On the ribbon:

Select 2D Sketch from the Sketch panel.
31. Select the top face of the boss.

32. Select the top edge of the boss and use Project Geometry to copy the edge into the sketch.

33. Right click and select Done.

34. Select Finish Sketch.
35. On the ribbon:

Select **Extrude** from the **Preparation and Machining** panel.

36. Set the distance for the cut to **1 mm**.

Press **OK**.

37. Right click on Machining in the browser.

Select **Finish Edit**.

38. On the ribbon:

Select **Preparation** from the **Process** panel.

39. Note that Preparations are now highlighted in the browser.
40. Adjust the view as shown.

*Note that the welds and machining operations are suppressed.*

41. On the ribbon:

Select **Chamfer** from the Preparation and Machining panel.

42. Select the two edges indicated.

Set the Distance to **30 mm**.

Press **OK**.

43. Right click on Preparations in the browser.

Select **Finish Edit**.

44. Save the file as *Welding2.iam*. 
45. Open **Welding.idw**.

46. Select **Base** from the Create Panel on the Place Views ribbon.

47. Select the **Model State** tab. *Note you can select which stage you want to appear in the view.*

48. Set the Orientation to **Left**.

49. Set the Weldment to **Assembly**.
50. Set the view to **shaded**.
Press **OK** and place the view.

51. Select **Base** from the Create Panel on the Place Views ribbon.

52. Select the **Model State** tab.  
*Note you can select which stage you want to appear in the view.*

53. Set the Orientation to **Left**.
54. Set the Weldment to **Machining**.

55. Set the view to **shaded**.

Press **OK** and place the view below the first view.

56. Select **Base** from the Create Panel on the Place Views ribbon.

57. Select the **Model State** tab.
58. Set the Orientation to **Left**.

59. Set the Weldment to **Welds**.

60. Set the view to **shaded**.

   Press **OK** and place the view below the other views.

61. Select **Base** from the Create Panel on the Place Views ribbon.
62. Select the **Model State** tab.

63. Set the Orientation to **Left**.

64. Under Preparation, you can select the entire assembly, a specific component or the hole features.

Select the assembly.
65. Set the view to **shaded**.

Press **OK** and place the view below the other views.

Note the differences between the views.

66. Create a projected view from the second view.

67. Highlight the second view. Right click and select **Get Model Annotations** → **Get Welding Symbols**.
68. The welding symbols are added to the view.

69. Zoom into the projected view.

70. On the Annotate ribbon:

   Select the **Caterpillar** tool from the Symbols panel.

71. Set the Type to **Partial**.
72. Select **Edges**.

73. Select the edges indicated.

*Note the caterpillar annotation is added as the edges are selected.*

74. Select the **Options** tab.

Set the **Width** to **6 mm**.
Set the **Spacing** to **2 mm**.
Enable **Seam Visibility**.

Press **OK**.

75. You may need to check the selection of the edges to get the caterpillar annotation oriented properly.

76. Close without saving.