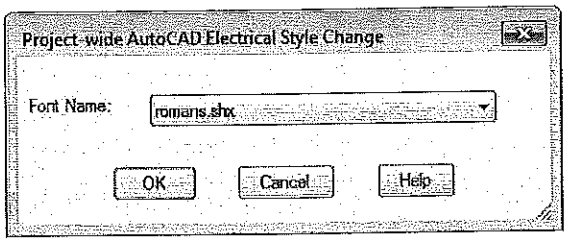

*Figure 13-35 The **Project-wide AutoCAD Electrical Style Change** dialog box*

For each drawing

This area is used to define script and run it for an active project. To define script for an active project, select the **Run command script file** check box and then choose the browse button located next to it; the **Select Script File** dialog box will be displayed. Select the desired script from it and then choose the **Open** button; the path for the selected script will be displayed in the edit box. Now, when you choose the **OK** button in the **Project-Wide Utilities** dialog box, the corresponding script will run automatically for the active project. The **Purge all blocks** check box is selected to remove all unused blocks or symbols from the active project.

Wire Types

This area is used to import wire types from other drawings. To do so, select the **Import from specified drawing** check box and then choose the browse button located next to the edit box; the **Wire Type Import - Select Master Drawing** dialog box will be displayed. Select the desired drawing from it and choose the **Open** button; the path for the selected drawing will be displayed in the edit box. Now, choose the **Setup** button; the **Import Wire Types** dialog box will be displayed. This dialog box displays a list of wire types used in the selected drawing. Select the wire types to be imported and use the other options in this dialog box as per your requirement. Now, choose the **OK** button to close it.

After specifying the desired options in the **Project-Wide Utilities** dialog box, choose the **OK** button; the **Batch Process Drawings** dialog box will be displayed. If you select the **Project** button in this dialog box and choose **OK**; the **Select Drawings to Process** dialog box will be displayed. Select the drawings from this dialog box in which you want to make changes and choose **OK**; the selected drawings will be updated. Similarly, if you want to make changes in the active drawing only, select the **Active Drawing** radio button from the **Batch Process Drawings** dialog box and choose **OK**; the changes will be carried out in the active drawing only.

TUTORIALS

Tutorial 1

In this tutorial, you will create a symbol, insert attributes into it, and then save it. Next, you will insert the symbol created into the drawing. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

a. Create a new drawing.
b. Insert a ladder in the drawing.
c. Create a symbol, insert attributes, and insert wire connection attributes to the symbol.

d. Save and insert the symbol.
e. Save the drawing file.

Creating a New Drawing

1. Create a new drawing *C13_tut01.dwg* in the **CADCIM** project with the *ACAD_ELECTRICAL.dwt* template and move it to the *TUTORIALS* subfolder, as already discussed in the previous chapters.

Inserting a Ladder

1. Choose the **Insert Ladder** tool from **Schematic > Insert Wires/Wire Numbers > Insert Ladder** drop-down; the **Insert Ladder** dialog box is displayed.

2. Set the following parameters in the **Insert Ladder** dialog box:

Width: **12.000**
1st Reference: **100**
1 Phase: Select this radio button
Spacing: **5.000**
Rungs: **4**
Yes: Select this radio button

Keep the values in the rest of the edit boxes intact.

3. Choose the **OK** button in the **Insert Ladder** dialog box; you are prompted to specify the start position of the first rung. Enter **8,18** at the Command prompt and press ENTER; the ladder is inserted in the drawing.

Creating a Symbol and Inserting Attributes in it

1. In order to create a symbol, first draw a circle in the drawing by choosing the **Center, Radius** tool from the **Circle** drop-down in the **Draw** panel of the **Home** tab or by choosing **Draw > Circle > Center, Radius** from the menu bar; you are prompted to specify the center point for the circle. Enter **7.5,30** at the Command prompt and press ENTER; you are prompted to specify the radius of the circle.

2. Enter **0.5** at the Command prompt and press ENTER; the circle is inserted in the drawing.

3. Choose **Draw > Polygon** from the menu bar; you are prompted to specify the number of sides.

4. Enter **4** at the Command prompt and press ENTER; you are prompted to specify the center of the polygon.

5. Select the center of the circle as the center of the polygon; you are prompted to enter an option.

Note

*To snap the center of a circle, right-click on the **Object Snap** button in the Status Bar; a shortcut menu is displayed. Choose the **Object Snap Settings** options from the shortcut menu; the **Drafting Settings** dialog box is displayed. Select the **Center** check box from the **Object Snap** tab and choose the **OK** button to save the changes made in this dialog box. Press F3, if the object snap is off.*

6. Enter **I** at the Command prompt and press ENTER; you are prompted to specify the radius of the circle.

7. Enter **6,30** at the Command prompt and press ENTER; the polygon is inserted in the drawing, as shown in Figure 13-36.

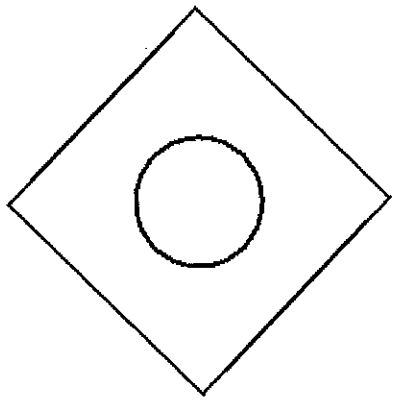

Figure 13-36 The polygon inserted in the drawing

8. Choose the **Symbol Builder** tool from **Schematic > Other Tools > Symbol Builder** drop-down; the **Select Symbol / Objects** dialog box is displayed.

9. By default, the **Unnamed** option is selected in the **Name** drop-down list. Do not change this option as this is used to create the symbol from scratch.

10. Choose the **Select objects** button from the **Select from drawing** area; you are prompted to select the objects. Select the circle and polygon and then press ENTER; the **Select Symbol / Objects** dialog box is displayed again and the preview of the polygon inscribed in circle is displayed in the **Preview** area of the **Select Symbol / Objects** dialog box.

11. In the **Attribute template** area, choose the **Browse** button located next to the **Library path** drop-down list; the **Browse For Folder** dialog box is displayed. In this dialog box, select the path *C:\users\public\ public doc...\NFPA* in the **Library path** drop-down list. Next, make sure **Horizontal Parent** is selected in the **Symbol** drop-down list, and **GNR (Generic)** is selected in the **Type** drop-down list. Keep rest of the values intact.

12. Choose the **OK** button in the **Select Symbol / Objects** dialog box; the **Block Editor** environment, the **Symbol Builder Attribute Editor** palette, the **Symbol Builder** and **Block Editor** tabs, and the **Block Authoring Palettes - All Palettes** palette are displayed. If the **Block Editor** environment is not invoked, choose the **Block Editor** tab to invoke it.

13. Select the **TAG1** row from the **Required** rollout of the **Symbol Builder Attribute Editor** palette and then choose the **Properties** button; the **Insert / Edit Attributes** dialog box is displayed. Enter **0.25** in the **Height** row of the **Text** area. Next, choose the **OK** button to save the changes made and exit the dialog box.

14. Choose the **Insert Attribute** button; you are prompted to specify the insertion base point of this tag.

15. Enter **7.5,31.65** at the Command prompt and press ENTER; the TAG1 attribute for the symbol is inserted above the symbol, as shown in Figure 13-37.

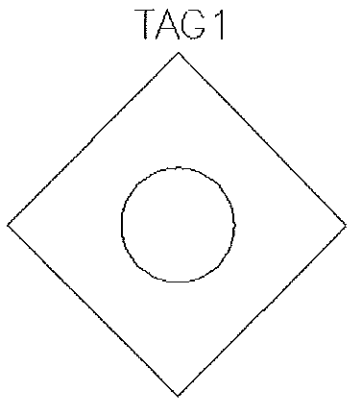

Figure 13-37 *The TAG1 attribute inserted into the symbol*

16. Select the **MFG** row from the **Symbol Builder Attribute Editor** palette and then choose the **Properties** button; the **Insert / Edit Attributes** dialog box is displayed. Enter **0.1** in the **Height** row. Choose the **OK** button to exit the dialog box.

17. Choose the **Insert Attribute** button; you are prompted to specify the insertion base point of the tag.

Note

You can also insert an attribute to the symbol by right-clicking on the attribute and choosing the ***Insert Attribute*** *option from the shortcut menu displayed.*

18. Insert MFG tag in the drawing, refer to Figure 13-38.

19. Change the height of the CAT, ASSYCODE, and FAMILY attributes to **0.1** in the **Insert / Edit Attributes** dialog box as discussed earlier and insert them into the symbol using the **Insert Attribute** button, refer to Figure 13-38. Make sure the **Snap Mode** button is deactivated to insert these attributes at desired places.

20. Similarly, change the height of the DESC1, DESC2, DESC3, INST, and LOC attributes to **0.25** using the **Properties** button in the **Required** rollout and then insert these attributes to the symbol using the **Insert Attribute** button, as shown in Figure 13-39.

21. To insert wire connections into the symbol, scroll down in the **Symbol Builder Attribute Editor** palette and make sure **Left/None** is selected in the **Direction/Style** row of the **Wire Connection** rollout.

22. Choose the **Insert Wire Connection** button from the **Wire Connection** rollout; you are prompted to select Left or (Top/Bottom/Right/rAdial).

23. Enter **6,30** at the Command prompt and press ENTER; the TERM01 attribute is inserted into the symbol and you are prompted to select the location for TERM02. Enter **R** at the Command prompt and press ENTER.

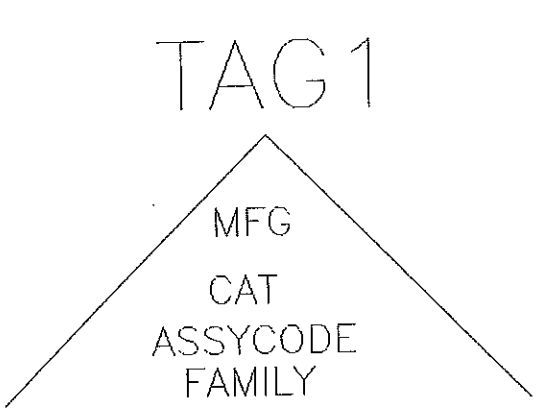

Figure 13-38 Different attributes inserted into the symbol

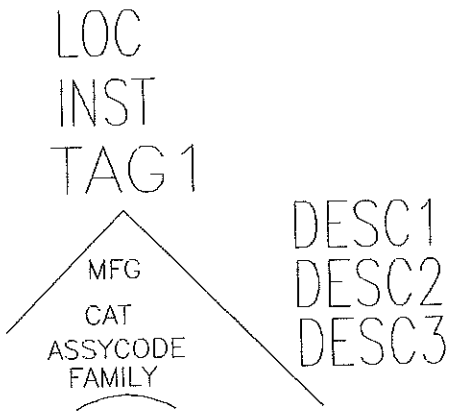

Figure 13-39 Different attributes inserted into the symbol

24. Enter **9,30** at the Command prompt and press ENTER; the TERM02 is inserted into the symbol. Figure 13-40 shows the TERM01 and TERM02 attributes inserted into the symbol.

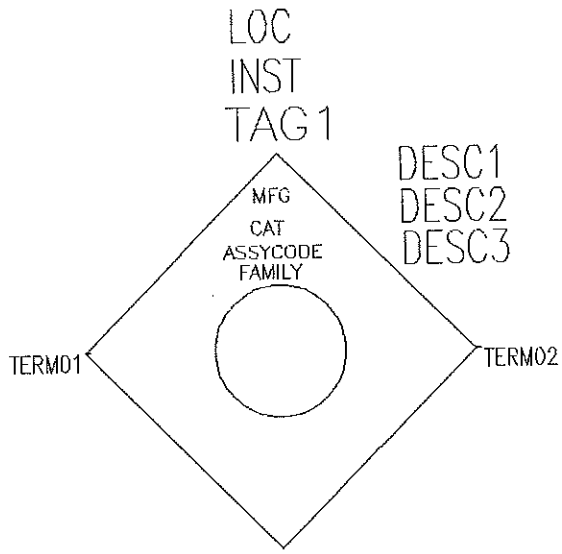

Figure 13-40 TERM01 and TERM02 attributes inserted into the symbol

25. Press ENTER to exit the command.

26. Select the **TERM01** attribute displayed in the **Pins** rollout and choose the **Properties** button; the **Insert / Edit Attribute** dialog box is displayed.

27. Enter **0.25** in the **Height** row and choose the **OK** button; the height of TERM01 is changed.

28. Similarly, change the height of the TERM02 attribute to **0.25**.

Saving and Inserting the Symbol

1. To save the symbol, choose the **Done** button from the **Edit** panel of the **Symbol Builder** tab; the **Close Block Editor: Save Symbol** dialog box is displayed.

2. Select the **Wblock** radio button from the **Symbol** area, if it is not selected.

3. Choose the **Pick point** button from the **Base point** area; you are prompted to specify the insertion base point. Select the center of the circle as the base point.

4. Enter **_MY SYMBOL** in the **Unique identifier** edit box and click in the **Symbol name** edit box; the name of the symbol is displayed in the **Symbol name** edit box as HDV1_MY SYMBOL. Keep the rest of the values in this dialog box intact.

5. Choose the **OK** button in the **Close Block Editor: Save Symbol** dialog box; the **Close Block Editor** message box is displayed.

6. Choose the **Yes** button in the **Close Block Editor** message box for inserting the symbol; you are prompted to specify the insertion point for the symbol.

7. Enter **13.5,18** at the Command prompt and press ENTER; the **Insert / Edit Component** dialog box is displayed. Enter **100A** in the edit box of the **Component Tag** area and choose the **OK** button; the symbol is inserted into the ladder, as shown in Figure 13-41. Figure 13-42 shows the zoomed view of the symbol inserted into the ladder.

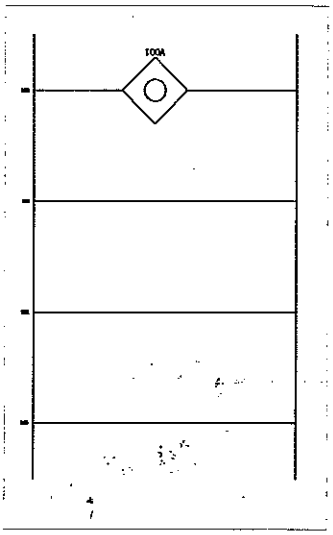

Figure 13-41 Symbol inserted into the ladder

Saving the Drawing File

1. Choose **File > Save** from the menu bar or choose **Save** from the **Application Menu** to save the drawing file, *C13_tut01.dwg*.

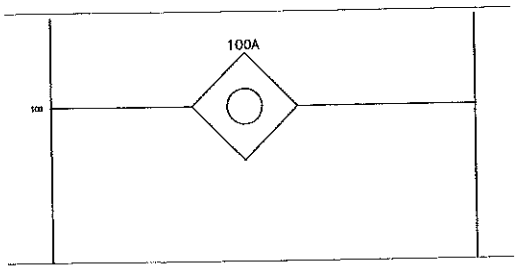

Figure 13-42 The zoomed view of the symbol inserted into the ladder

Tutorial 2

In this tutorial, you will add the symbol that you created in Tutorial 1 of this chapter to the **Insert Component** dialog box using the **Icon Menu Wizard** tool. You will then insert the component to the drawing. **(Expected time: 20 min)**

The following steps are required to complete this tutorial:

a. Open, save, and add the drawing to the active project.
b. Add a new icon to the menu.
c. Insert the component.
d. Save the drawing.

Opening, Saving, and Adding the Drawing to the Active Project

1. Open the *C13_tut01.dwg* from the **CADCIM** project and activate the CADCIM project.

2. Save the *C13_tut01.dwg* file with the name *C13_tut02.dwg*. You can also download this file from the CADCIM website. The path of the file is as follows:

   Textbooks > CAD/CAM > AutoCAD Electrical > AutoCAD Electrical 2018 for Electrical Control Designers

3. Add the drawing *C13_tut02.dwg* to the **CADCIM** project, as discussed in the previous chapters.

Adding a New Icon to the Menu

1. To add the icon of the new symbol, choose the **Icon Menu Wizard** tool from the **Other Tools** panel of the **Schematic** tab; the **Select Menu file** dialog box is displayed.

2. Choose the **Schematic** button, if *ACE_NFPA_MENU.DAT* is not displayed in the edit box.

3. Next, choose the **OK** button; the **Icon Menu Wizard** dialog box is displayed.

4. Click on the **Add** drop-down list; various options are displayed. Select the **Component** option from the drop-down list; the **Add Icon - Component** dialog box is displayed.

5. Enter **MY SYMBOL** in the **Name** edit box of the **Icon Details** area.
6. Next, choose the **Pick <** button on the right of the **Image file** edit box; you are prompted to select the block.

7. Select the symbol you created in Tutorial 1 of this chapter; the name of block (HDVI_MY SYMBOL) is displayed in the **Image file** edit box.

8. Clear the **Create PNG from current screen image** check box, if it is selected.

9. Next, choose the **Browse** button on the right of the **Block name** edit box; the **Select File** dialog box is displayed. Browse to *"C:\Users\Public\Public Documents\Autodesk\Acade 2018\ Libs\ NFPA"*. Enter **HDV1_MY SYMBOL** in the **File Name** edit box and choose the **Open** button; the path and location of *HDV1_MY SYMBOL.dwg* is displayed in the **Block name** edit box.

10. Choose the **OK** button in the **Add Icon - Component** dialog box; the icon is added to the **Icon Menu Wizard** dialog box, as shown in Figure 13-43.

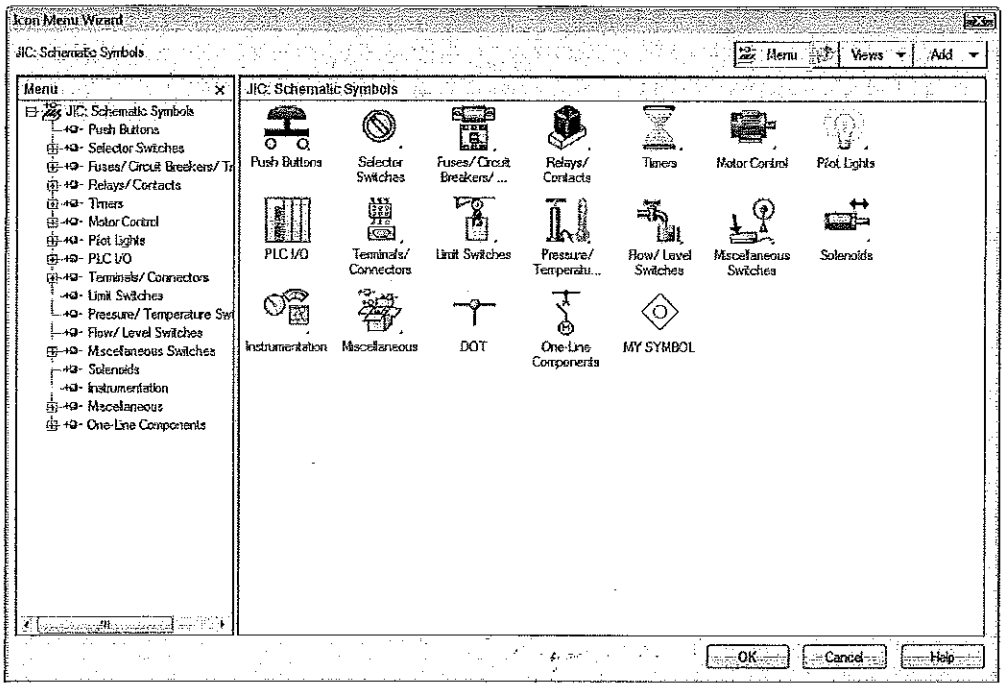

*Figure 13-43 The **Icon Menu Wizard** dialog box showing the icon of the symbol*

11. Choose the **OK** button in the **Icon Menu Wizard** dialog box to save the changes made and exit the dialog box.

Inserting the Component

1. In order to insert the component, choose the **Icon Menu** tool from **Schematic > Insert Components > Icon Menu** drop-down; the **Insert Component** dialog box is displayed.

2. Select **MY SYMBOL** from this dialog box; you are prompted to specify the insertion point.

3. Enter **13,3** at the Command prompt and press ENTER; the **Insert / Edit Component** dialog box is displayed. Enter **103** in the edit box of the **Component Tag** area and choose the **OK** button; the symbol is inserted in the rung 103 of the ladder, as shown in Figure 13-44. Figure 13-45 shows the zoomed view of the symbol inserted into the ladder.

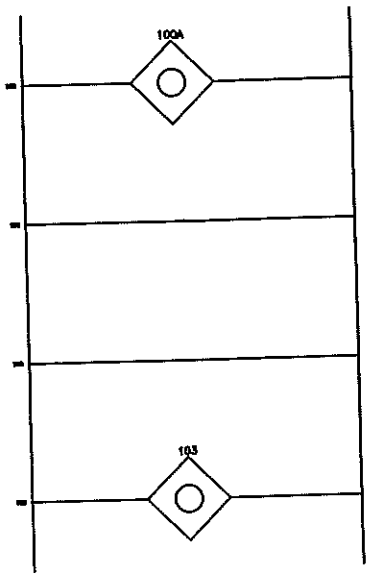

Figure 13-44 Symbol inserted into the ladder

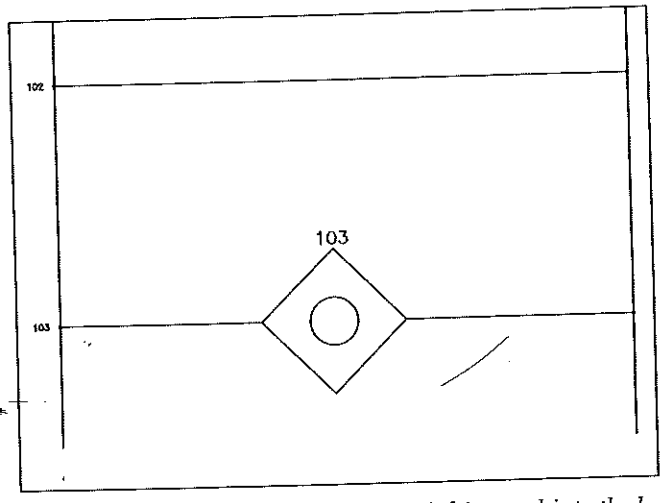

Figure 13-45 The zoomed view of the symbol inserted into the ladder

Saving the Drawing File

1. Choose **Save** from the **Application Menu** or **File > Save** from the menu bar to save the drawing file.

Tutorial 3

In this tutorial, you will export data from a drawing file to an excel sheet, make changes in that sheet, and then import the sheet data to the drawing file. **(Expected time: 15 min)**

The following steps are required to complete this tutorial:

a. Open, save, and add the drawing to the active project.
b. Export the data.
c. Modify the data.
d. Import the data.
e. Save the drawing.

Opening, Saving, and Adding the Drawing to the Active Project

1. Open the *C05_tut01.dwg* from the **CADCIM** project and activate the **CADCIM** project.

2. Save the *C05_tut01.dwg* file with the name *C13_tut03.dwg*. You can also download this file from the CADCIM website. The path of the file is as follows:

   Textbooks > CAD/CAM > AutoCAD Electrical > AutoCAD Electrical 2018 for Electrical Control Designers

3. Add the drawing *C13_tut03.dwg* to the **CADCIM** project, as discussed in the previous chapters.

Exporting the Data

1. Choose the **To Spreadsheet** tool from the **Export** panel of the **Import/Export Data** tab; the **Export to Spreadsheet** dialog box is displayed, as shown in Figure 13-46.

2. In this dialog box, make sure the **Components** radio button is selected. Next, choose the **OK** button; the **Component Data Export** dialog box is displayed.

3. In this dialog box, select the **Active Drawing** radio button from the **Data export for** area. Next, select the **Excel file format (.xls)** radio button from the **Output format** area and then choose the **OK** button; the **Select file name for drawing's XLS output** dialog box is displayed.

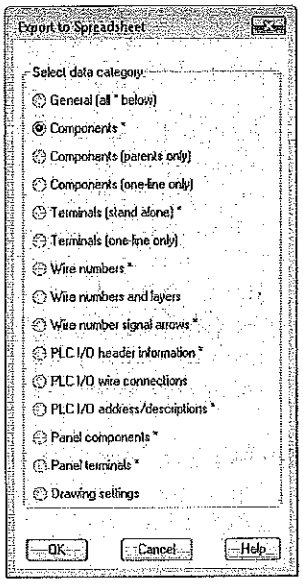

Figure 13-46 The Export to Spreadsheet dialog box

4. In this dialog box, enter **C13_tut03_components** in the **File name** text box and select **C:\Users\User Name\Documents\AcadE 2018\AeData\Proj\CADCIM** from the **Save in** drop-down list, refer to Figure 13-47. Next, choose the **Save** button; an Excel file with the name *c13_tut03_components* is created at the specified location.

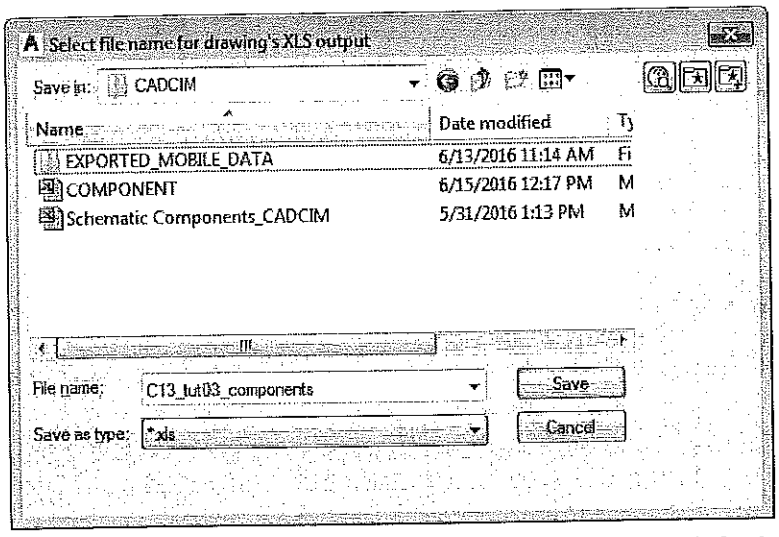

Figure 13-47 The Select file name for drawing's XLS output dialog box

5. Choose the **To Spreadsheet** tool again from the **Export** panel of the **Import/Export Data** tab; the **Export to Spreadsheet** dialog box is displayed.

6. In this dialog box, select the **Drawing settings** radio button and choose the **OK** button; the **Drawing Settings Data Export** dialog box is displayed. In this dialog box, make sure that the **Project** radio button is selected from the **Data export for** area and the **Excel file format [.xls]** is selected from the **Output format** area. Next, choose the **OK** button; the **Select Drawings to Process** dialog box is displayed.

7. In this dialog box, select all the tutorial files in the **CADCIM** project from the top list (Chapter 2 to Chapter 13) and then choose the **Process** button; all the tutorial files are shifted to the bottom part of the **Select Drawings to Process** dialog box. Next, choose the **OK** button; the **Select file name for Project-wide XLS output** dialog box is displayed.

8. In this dialog box, enter **CADCIM_drawing settings** in the **File name** text box and select the **C:\Users\User Name\Documents\AcadE 2018\AeData\Proj\CADCIM** from the **Save in** drop-down list. Next, choose the **Save** button; an Excel file with the name *CADCIM_drawing settings* is created at the specified location.

Modifying the Data

1. Open the Windows Explorer and browse to the location *C:\Users\User Name\Documents\AcadE 2018\AeData\Proj\CADCIM*. Next, open the *C13_tut03_components* file from this location.

2. Modify the data in the **DESC1** and **DESC2** columns of this file, refer to Figure 13-48. Next, save and close the file.

	A	B	C	D	E	F	G
1	(PAR1 CH	FAMILY	TAGNAME	DESC1	DESC2	DESC3	(REF)
2	1	PB	PB1	NO	PUSH BUTTON		1
3	1	PB	PB1A	NC	PUSH BUTTON		1
4	1	CR	CR1	CTRL RELAY			1
5	2	CR	CR1	CTRL RELAY	NO		2
6	2	CR	CR1	CTRL RELAY	NC		3
7	1	LT	LT3	GREEN	OFF		3
8	2	CR	CR1	CTRL RELAY	NO		4
9	1	LT	LT4	RED	ON		4
10							
11							
12							

Figure 13-48 *The data changed in the* ***DESC1, DESC2*** *columns*

3. Open the *CADCIM_drawing settings* file from the *C:\Users\User Name\Documents\AcadE 2018\AeData\Proj\CADCIM* location.

4. Change the data in the **SEC**, **SUBSEC**, and **SHDWGNAM** columns of this file, as shown in Figure 13-49. Next, save and close the file.

	A	B	C	D	E
1	(DWGNAM)	SEC	SUBSEC	SH	SHDWGNAM
2	C02_TUT01	C2	1	01	201
3	C03_TUT01	C3	1	05	301
4	C03_TUT02	C3	2	05	302
5	C03_TUT03	C3	3	05	303
6	C03_TUT04	C3	4	05	304
7	C04_TUT01	C4	1	01	401
8	C04_TUT03	C4	3		403
9	C04_TUT04	C4	4	A	404
10	C05_TUT01	C5	1		501
11	C05_TUT02	C5	2		502
12	C05_TUT03	C5	3		503
13	C05_TUT04	C5	4		504
14	C06_TUT01	C6	1		601
15	C06_TUT03	C6	3		603
16	C06_TUT01_UPDATE	C6	1	01	601
17	C06_TUT03_UPDATE	C6	3	02	603
18	C07_TUT01	C7	1		701
19	C07_TUT02	C7	2		702
20	C07_TUT03	C7	3		703
21	C07_TUT04	C7	4		704
22	C08_TUT01	C8	1		801
23	C08_TUT02	C8	2		802
24	C08_TUT03	C8	3		803
25	C08_TUT04	C8	4		804
26	C09_TUT01	C9	1		901
27	C09_TUT02	C9	2		902
28	C10_TUT01	C10	1		1001
29	C10_TUT2	C10	2		1002
30	C10_TUT03	C10	3	01	1003
31	C10_TUT04	C10	4	02	1004
32	C10_TUT05	C10	5	03	1005
33	C11_TUT01	C11	1		1101
34	C11_TUT02	C11	2		1102
35	C11_TUT03	C11	3		1103
36	C12_TUT01	C12	1	01	1201
37	C12_TUT02	C12	2	01	1202
38	C04_TUT02	C4	2	01	402
39	C12_TUT03	C12	3		1203
40	C12_TUT04	C12	4	01	1204
41	C12_TUT05	C12	5	05	1205
42	C13_TUT01	C13	1		1301
43	C13_TUT02	C13	2		1302
44	C13_TUT03	C13	3		1303

Figure 13-49 *The data changed in the* ***SEC, SUBSEC,*** *and* ***SHDWGNAM*** *columns*

Importing the Data

1. Make sure the *C13_tut03* file is open in AutoCAD Electrical. Next, choose the **From Spreadsheet** tool from the **Import** panel of the **Import/Export Data** tab; the **Update Drawing from Spreadsheet File** dialog box is displayed.

2. In this dialog box, browse to the *C:\Users\User Name\Documents\AcadE 2018\AeData\Proj\CADCIM* in the **Look in** drop-down list and then select the *c13_tut03_components* file from the list displayed. Next, choose the **Open** button; the **Update Drawings per Spreadsheet Data** dialog box is displayed.

3. In this dialog box, select the **Active Drawing** radio button from the **Update drawings per spreadsheet data file. Process** area. Next, choose the **OK** button. You will notice that the description of components in the *C13_tut03* file is changed.

4. Choose the **From Spreadsheet** tool again from the **Import** panel of the **Import/Export Data** tab; the **Update Drawing from Spreadsheet File** dialog box is displayed.

5. In this dialog box, browse to the *C:\Users\User Name\Documents\AcadE 2018\AeData\Proj\CADCIM* in the **Look in** drop-down list and then select the *CADCIM_drawing settings* file from the list displayed. Next, choose the **Open** button; the **Update Drawings per Spreadsheet Data** dialog box is displayed.

6. In this dialog box, select the **Project** radio button from the **Update drawings per spreadsheet data file. Process** area. Next, choose the **OK** button; the **Select Drawings to Process** dialog box is displayed.

7. In this dialog box, select all the tutorial files in the **CADCIM** project from the top list (Chapter 2 to Chapter 13) and then choose the **Process** button; all the tutorial files are shifted to the bottom part of the **Select Drawings to Process** dialog box. Next, choose the **OK** button. If the **QSAVE** message box is displayed, choose the **OK** button in it.

   You will notice that all the tutorial files are opened, changed, and then closed one by one.

   To verify the changes in the drawing settings of the tutorial files, you need to follow the steps given next.

8. Right-click on any of the tutorial files in the **CADCIM** project; the shortcut menu is displayed. Choose **Properties > Drawing Properties** from this shortcut menu, refer to Figure 13-50; the **Drawing Properties** dialog box is displayed.

9. In this dialog box, the values in the **Drawing, Section, Sub-Section** edit boxes of the **Sheet Values** area are changed, refer to Figure 13-51.

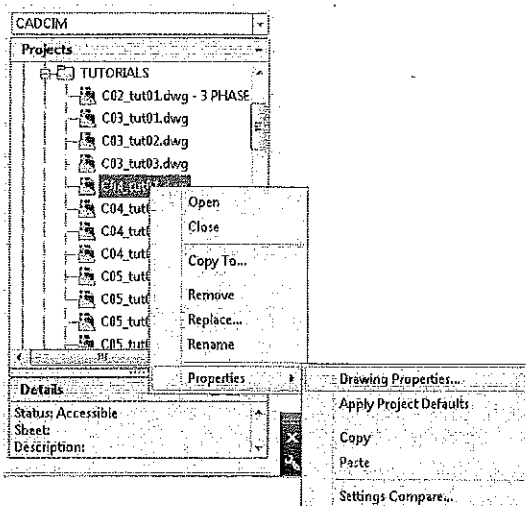

Figure 13-50 Choosing ***Properties > Drawing Properties*** *from the shortcut menu*

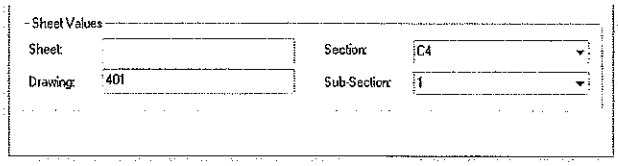

Figure 13-51 Values changed in the ***Drawing, Section, Sub-Section*** *edit boxes*

Saving the Drawing File

1. Choose **Save** from the **Application Menu** or **File > Save** from the menu bar to save the drawing file.

Tutorial 4

In this tutorial, you will mark a drawing in the **CADCIM** project, make changes in the drawing and then verify the drawing using the **Mark/Verify DWGs** tool. **(Expected time: 15 min)**

The following steps are required to complete this tutorial:

a. Open, save, and add the drawing.
b. Mark the drawing.
c. Modify the drawing.
d. Verify the drawing.
e. Save the drawing.

Opening, Saving, and Adding the Drawing to the Active Project

1. Open the *C13_tut03.dwg* from the **CADCIM** project and activate the **CADCIM** project.

2. Save the *C13_tut03.dwg* file with the name *C13_tut04.dwg*. You can also download this file from the CADCIM website. The path of the file is as follows:

Textbooks > CAD/CAM > AutoCAD Electrical > AutoCAD Electrical 2018 for Electrical Control Designers

3. Add the drawing *C13_tut04.dwg* to the **CADCIM** project, as discussed in the previous chapters.

Marking the Drawing

1. Choose the **Mark/Verify DWGs** tool from the **Project Tools** panel of the **Project** tab; the **Mark and verify** dialog box is displayed, as shown in Figure 13-52.

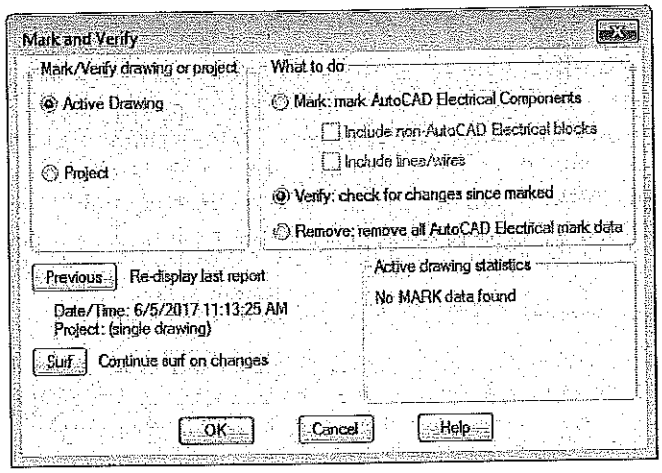

Figure 13-52 The Mark and Verify dialog box

2. In this dialog box, select the **Mark: mark AutoCAD Electrical Components** radio button from the **What to do** area and make sure that the **Active Drawing** radio button is selected in the **Mark/Verify drawing or project** area. Next, choose the **OK** button; the **Enter Your Initials** dialog box is displayed.

3. In this dialog box, enter the initials and the comments, if any in the respective edit boxes and choose the **OK** button; the AutoCAD electrical data is marked.

Note

1. To include non-AutoCAD electrical blocks in the marked data, select the ***include non-AutoCAD Electrical blocks*** *check box from the* ***What to do*** *area.*

2. To include lines and wires in the marked data, select the ***include lines/wires*** *check box from the* ***What to do*** *area.*

Modifying the Data

1. Choose the **Edit** tool from **Schematic > Edit Components > Edit Components** drop-down; you are prompted to select the component to be edited. Select the PB1 push button; the **Insert/Edit Component** dialog box is displayed.

2. Choose the **Lookup** button in the **Catalog Data** area; the **Catalog Browser** dialog box is displayed. Select **800H-BR6D1** from this dialog box and choose the **OK** button; the catalog data is added in the **Catalog Data** area of the **Insert/Edit Component** dialog box. Also, the **AutoCAD Message** message box is displayed. Choose the **OK** button from this message box. Next, choose the **OK** button from the **Insert/Edit Component** dialog box; the **Update other drawings** message box is displayed. Choose the **Task** button from this message box.

3. Choose the **Edit** tool from **Schematic > Edit Components > Edit Components** drop-down; you are prompted to select the component to be edited. Select the PB1A push button; the **Insert/Edit Component** dialog box is displayed.

4. Choose the **Lookup** button in the **Catalog Data** area; the **Catalog Browser** dialog box is displayed. In this dialog box, select **800H-BR6D2** from the **Catalog Browser** dialog box and choose the **OK** button; the catalog data is added in the **Catalog Data** area of the **Insert/Edit Component** dialog box. Next, choose the **OK** button from this dialog box; the **Update other drawings** message box is displayed. Choose the **Task** button from this message box.

Verifying the Data

1. Choose the **Mark/Verify DWGs** tool from the **Project Tools** panel of the **Project** tab; the modified **Mark and Verify** dialog box is displayed, as shown in Figure 13-53.

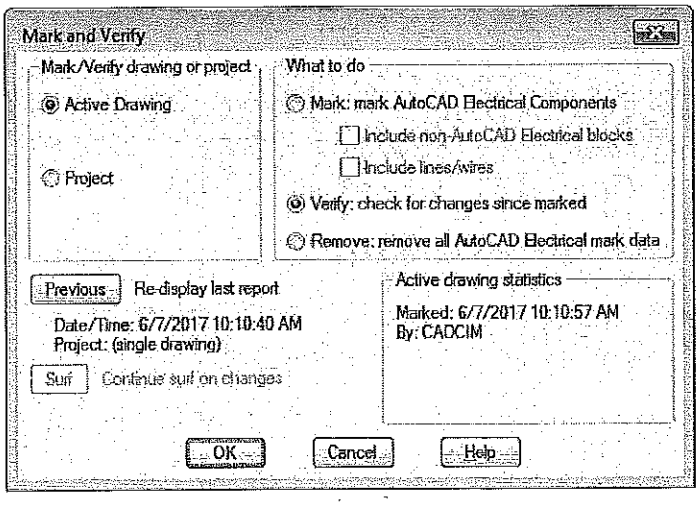

Figure 13-53 The modified Mark and Verify dialog box

Notice that the **Active drawing statistics** area displays the date and time when the data was marked for the active drawing along with the initials entered in the **Enter Your Initials** dialog box.

2. Make sure that the **Verify: check for changes since marked** radio button is selected in the **What to do** area. Next, choose the **OK** button; the **REPORT: Changes made on this drawing since last Mark command** dialog box is displayed, as shown in Figure 13-54.

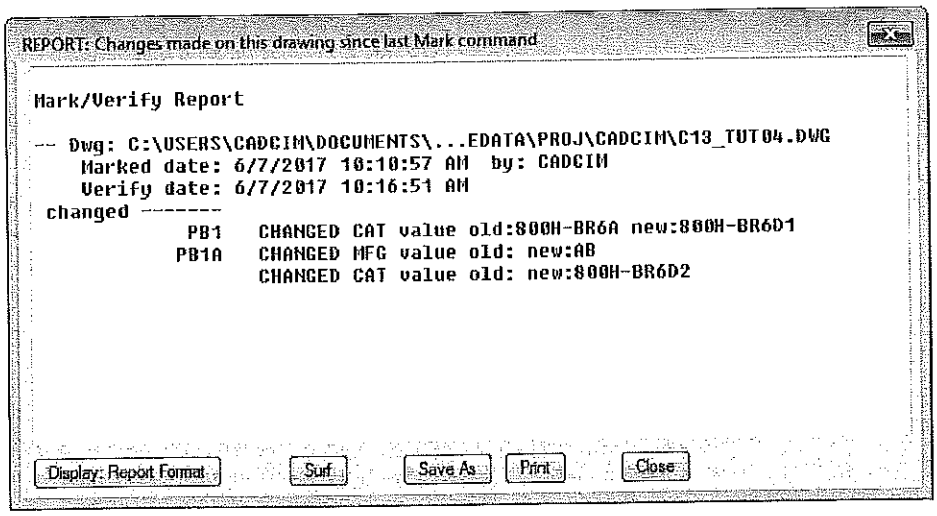

*Figure 13-54 The **REPORT: Changes made on this drawing since last Mark command** dialog box*

You can use the **Display Report Format** button from this dialog box to display the changes in the **Report Generator** dialog box, refer to Figure 13-55. Using this dialog box, you can print the changes, save them to a file or can put them in the table format in a drawing, as discussed in detail in Chapter 9.

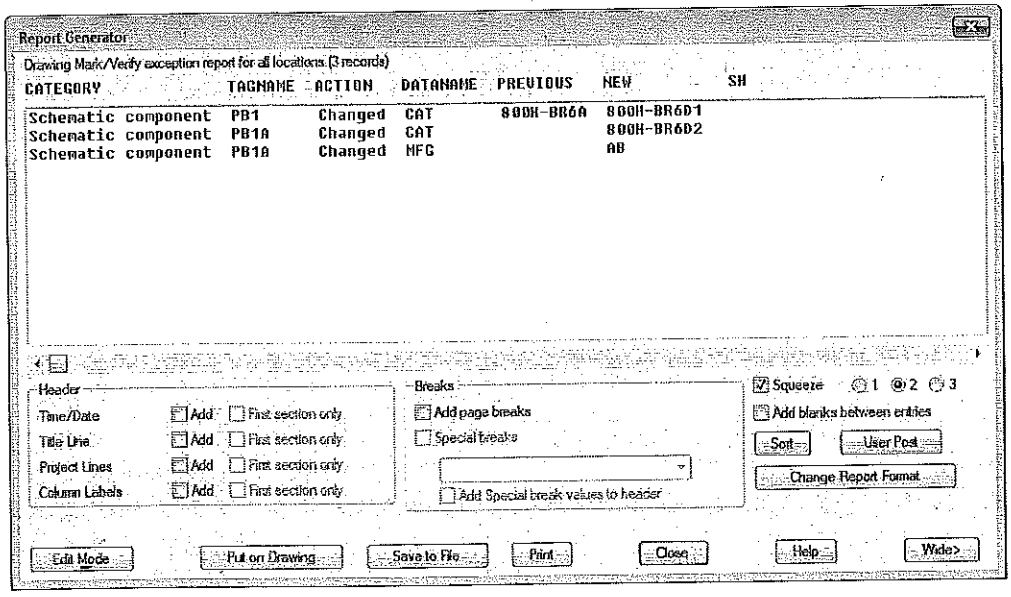

*Figure 13-55 The **Report Generator** dialog box*

Saving the Drawing File

1. Choose **Save** from the **Application Menu** or **File > Save** from the menu bar to save the drawing file.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. Which of the following dialog boxes will be displayed if you choose the **Symbol Builder** tool?

   (a) **Symbol Audit** (b) **Symbol Configuration**
   (c) **Select Symbol / Objects** (d) None of these

2. The **To Spreadsheet** tool is used to _____ data from the active drawing or project to an external file.

3. In AutoCAD Electrical, a symbol can be of any size and width. (T/F)

4. The **Icon Menu Wizard** tool is used to add or modify only the schematic symbol libraries. (T/F)

5. The AutoCAD blocks can be converted into AutoCAD Electrical intelligent symbols using the **Symbol Builder** tool. (T/F)

Review Questions

Answer the following questions:

1. Which of the following rollouts is used to select the style and direction of the wire connection attributes?

   (a) **Wire Connection** (b) **Required**
   (c) **Optional** (d) All of these

2. The icon menus can be customized using the _____ tool.

3. The _____ option is used to create a new circuit.

4 The _____ tool is used to import data from an external file to an active drawing or project.

5. The **Mark/Verify** tool is used to add an invisible mark on the components and wire of a drawing before sending it to the client. (T/F)

EXERCISES

Exercise 1

Create a new drawing with the name *C13_exer01.dwg* and insert a single-phase ladder with width = 15, spacing between rungs as 3, and number of rungs = 4. Next, create the symbol and add the attributes to it, as shown in Figure 13-56 and then save it as HDV1_SYMBOL. You will

also insert the symbol that you created into the ladder, as shown in Figure 13-57. Figure 13-58 shows the zoomed view of the symbol inserted into the ladder. **(Expected time: 25 min)**

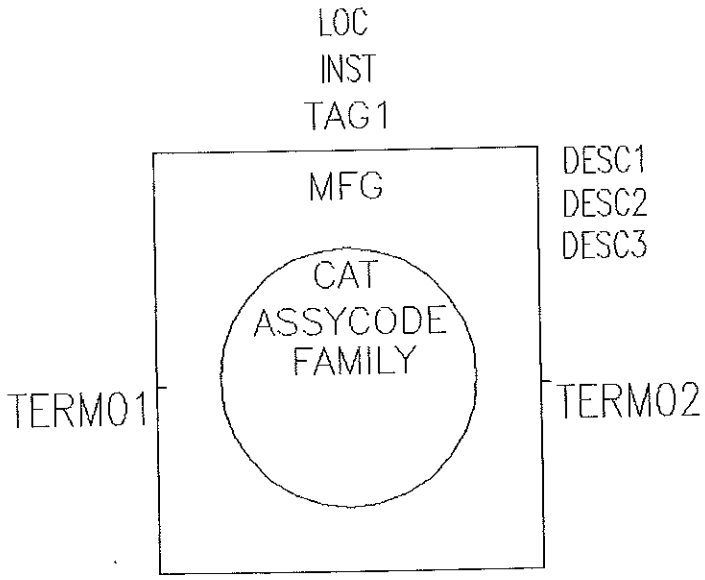

Figure 13-56 Attributes added to the symbol

Hint: Change the height of all attributes shown in Figure 13-56 to 0.1 in the **Symbol Builder Attribute Editor** palette. Also, insert TERM01 and TERM02 attributes at the mid-point of rectangle.

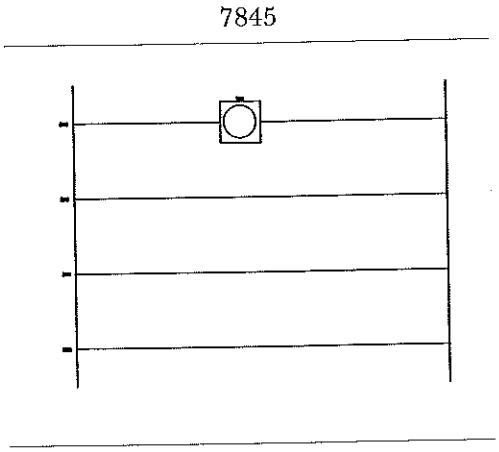

Figure 13-57 Symbol inserted into the ladder

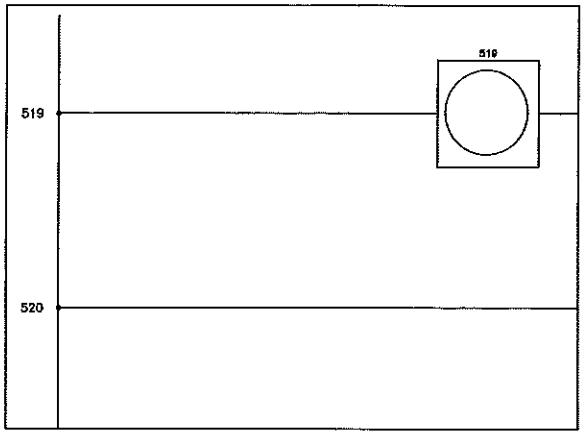

Figure 13-58 The zoomed view of the symbol inserted into the ladder

Exercise 2

Open the *C13_exer01.dwg* drawing file. Use the **Icon Menu Wizard** tool to create a sub menu in the **Icon Menu Wizard** dialog box, as shown in Figure 13-59, and then add the icon of the symbol that you created in Exercise 1 to the **Miscellaneous** sub menu, as shown in Figure 13-60. Next, insert the symbol into the ladder, as shown in Figure 13-61.

(Expected time: 20 min)

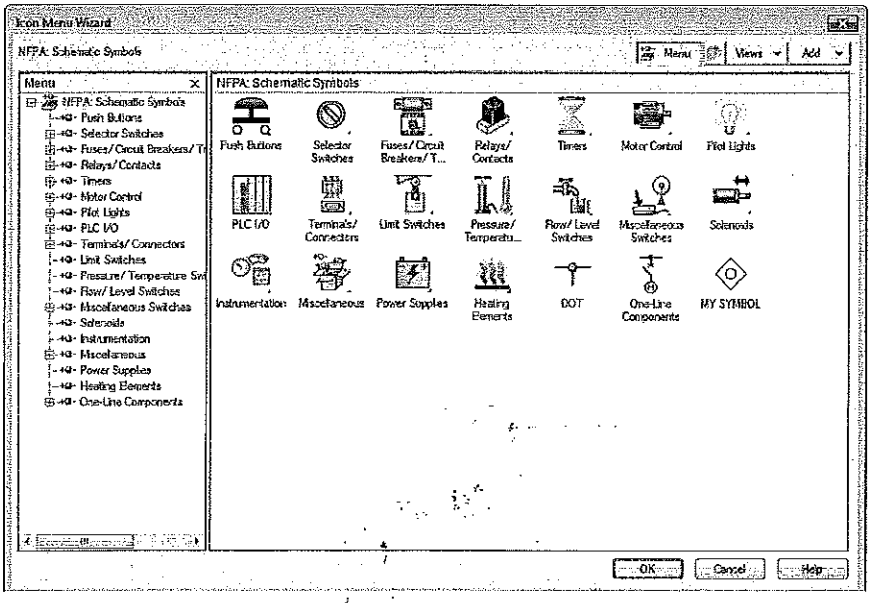

Figure 13-59 Submenu created in the ***Icon Menu Wizard*** *dialog box*

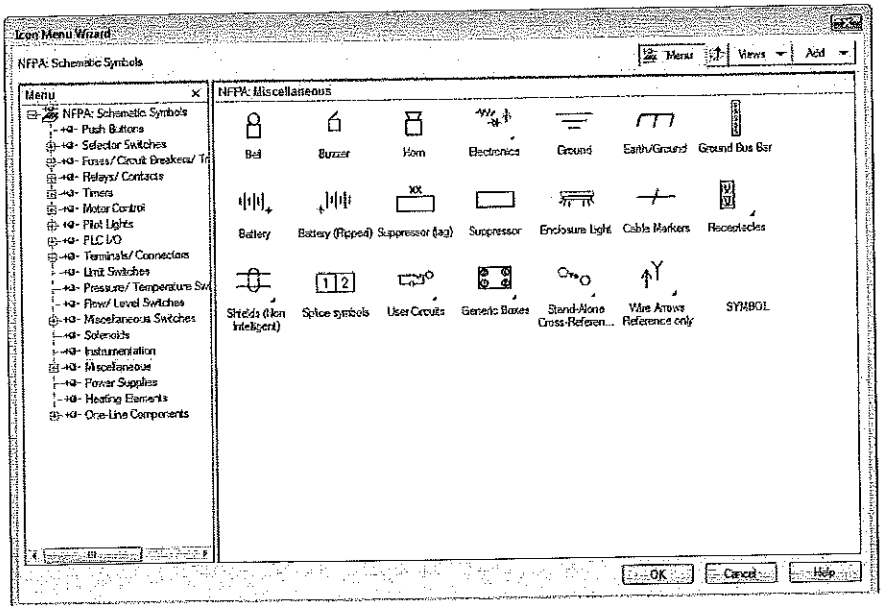

Figure 13-60 The **Icon Menu Wizard** *dialog box displaying the* **Miscellaneous** *submenu*

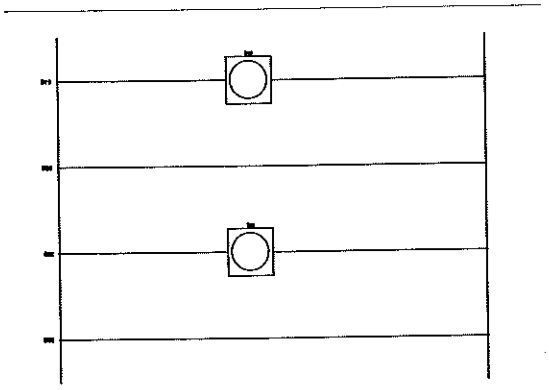

Figure 13-61 Symbol inserted into the ladder

Answers to Self-Evaluation Test

1. c, 2. export, 3. T, 4. F, 5. T

Project 1

Schematic Drawing

PROJECT DESCRIPTION

In this project, you will create a motor control circuit, as shown in Figure P1-1. This will be created by using ladder, multiple phase bus, and components such as 3 Phase Motor, 3 Pole Thermal Circuit Breaker, and terminals. Then, you will copy the circuit and save it in the icon menu. Next, you will insert the saved circuit in the drawing, audit the drawing and save the auditing reports. Also, you will generate the **Wire From/To** report and place it in the new drawing.