

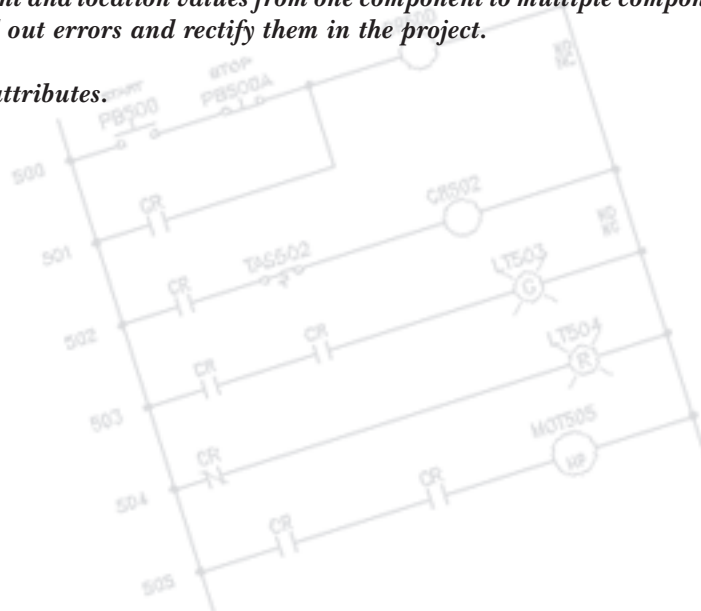
Chapter 6

Schematic Editing

Learning Objectives

After completing this chapter, you will be able to:

- Use basic schematic editing commands.
- Toggle between the normally open and normally closed contacts and vice versa.
- Copy the catalog assignment and location values from one component to multiple components.
- Use auditing tools to find out errors and rectify them in the project.
- Retag the drawings.
- Use the tools for editing attributes.



INTRODUCTION

In this chapter, you will learn about different tools that are used to create or modify electrical schematic. You can edit, move, copy, and align components in your drawing. These tools are very important as they provide extra functionalities to electrical schematic drawings. Also, later in this chapter you will learn about surfing of components, auditing tools, editing tools of attributes, and retagging of drawing.

CHANGING THE COMPONENT LOCATION WITH SCOOT

Ribbon:	Schematic > Edit Components > Scoot
Toolbar:	ACE:Main Electrical > Scoot or ACE:Scoot > Scoot
Menu:	Components > Scoot
Command:	AESCOOT



The **Scoot** tool is used to move objects such as components, terminals, PLC I/O modules, signal arrows, wire segments, wires with wire crossing-loops, ladder rungs, ladder buses, and so on in the drawing. The **Scoot** tool is similar to the AutoCAD **Move** tool. However, the **Scoot** tool has intelligence about electrical objects, which enables you to reposition electrical objects. Also, this tool is used to reconnect wires after repositioning the components. To scoot components, choose the **Scoot** button from the **Edit Components** panel of the **Schematic** tab; you will be prompted to select component, wire, or wire number for scoot. Select the component to scoot along with its connected wire; a temporary graphics indicating that you have selected the component will be displayed. Next, move the cursor to a place where you want to locate the component; the selected object will move in the orthogonal direction to the specified location. Also, the wires will be reconnected after you scoot the components, the components will be updated, and the existing wire numbers will be recentered. Next, press ESC or click on the screen to exit the command.

Figures 6-1(a) and 6-1(b) show the components and wires before and after scooting.

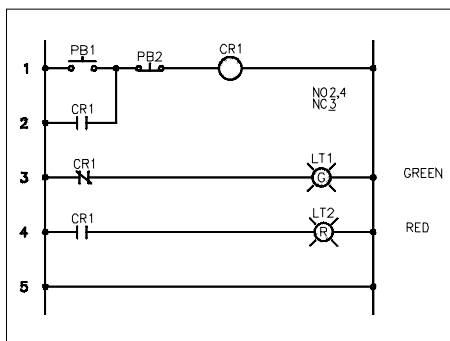


Figure 6-1(a) Components and wires before scooting

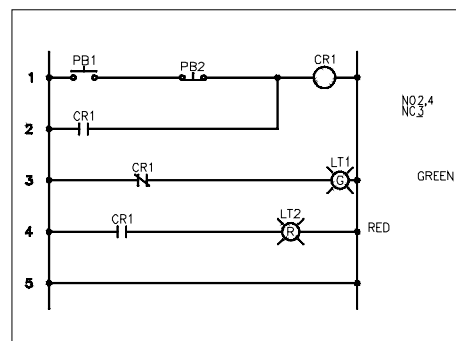


Figure 6-1(b) Components and wires after scooting

If you move components to a location that requires component updates, the **Component(s) Moved** dialog box will be displayed, as shown in Figure 6-2. The options in this dialog box are discussed next.

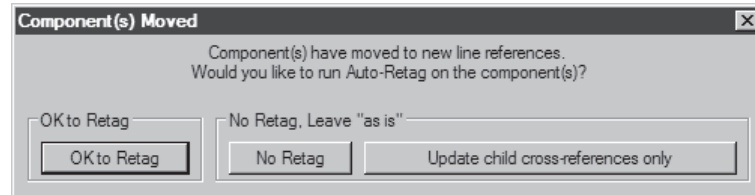


Figure 6-2 The *Component(s) Moved* dialog box

OK to Retag

The **OK to Retag** button is used to retag components. To do so, choose the **OK to Retag** button; the components will be automatically retagged. Now, if the moved component is linked to other components in the current drawing, the **Update Related Components?** dialog box will be displayed, as shown in Figure 6-3. Choose the **Yes-Update** button to update the related component in the drawing.

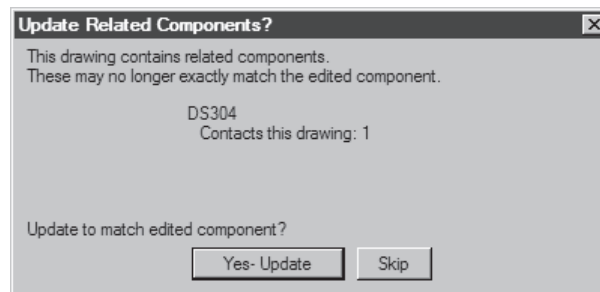


Figure 6-3 The *Update Related Components?* dialog box

Also, if the component that you have moved consists of child components or related panel components in the other drawings of the active project, the **Update other drawings?** dialog box will be displayed, as shown in Figure 6-4. Choose the **OK** button from the **Update other drawings?** dialog box to batch process or update the other drawings of a project.

Choose the **Task** button to save all modifications in a task list to be run later. The task list is maintained inside the project task list database file (project_update.mdb). You can access this list by choosing the **Project Task List** button from the **Project Manager**. Choose the **Skip** button to skip the batch process.

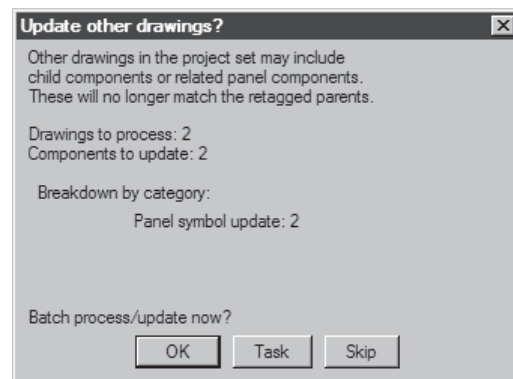


Figure 6-4 The *Update other drawings?* dialog box

**Note**

*If the moved component does not have related components in the current drawing, the **Update other drawings?** dialog box will be displayed directly after choosing the **OK to Retag** button from the **Component(s) Moved** dialog box.*

*If the moved component has related components in the current drawing as well as in the other drawings of the project, the **Update other drawings** will be displayed after choosing the **Yes-Update** button in the **Update Related Component?** dialog box.*

No Retag

If you do not want to retag a component, choose the **No Retag** button.

Update child cross-references only

Choose the **Update child cross-references only** button to update only the child cross-references.

**Note**

*If you select the wire segment that contains components to scoot, the entire wire along with its components and bus will be scooted. If they move to a new line reference, you can auto-retag components by choosing the **OK to Retag** button from the **Component(s) Moved** dialog box.*

CHANGING COMPONENT LOCATIONS WITH MOVE COMPONENT

Menu:	Schematic > Edit Components > Scoot > Move Component
Toolbar:	ACE:Main Electrical > Scoot > Move Component or ACE:Scoot > Move Component
Menu:	Components > Move Component
Command:	AEMOVE



The **Move Component** tool is used to move the selected component from its current location or wire location to the specified location. To do so, choose the **Move Component** button from the **Scoot** flyout of the **Edit Components** panel of the **Schematic** tab; you will be prompted to select a component to move. Select the component; you will be prompted to specify insertion point. Specify the insertion point for the component; the selected component will be moved to the specified location, the underlying wires will be reconnected, and the component tags will be updated automatically. Figures 6-5(a) and 6-5(b) show the components before and after moving.

If you move objects to a location that needs component updates, the **Component(s) Moved** dialog box will be displayed, refer to Figure 6-2. The options in this dialog box have been discussed earlier.

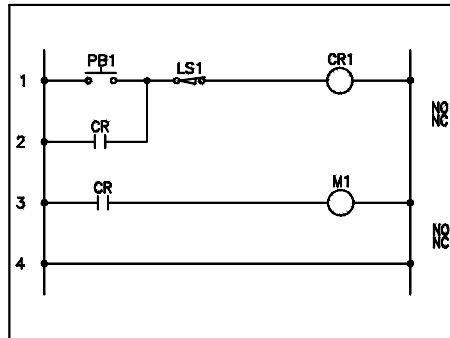


Figure 6-5(a) Components before using the *Move Component* tool

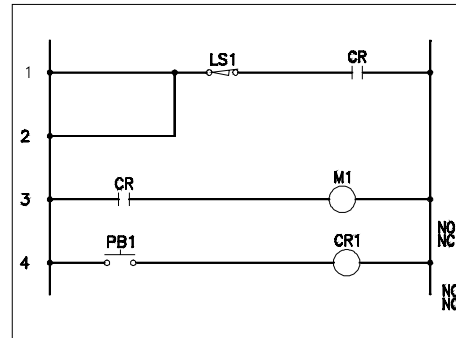


Figure 6-5(b) Components after using the *Move Component* tool

COPYING A COMPONENT

Ribbon:	Schematic > Edit Components > Copy Component
Toolbar:	ACE:Main Electrical > Copy Component
Menu:	Components > Copy Component
Command:	AECOPYCOMP



The **Copy Component** tool is used to copy the selected component and insert the copied component to the specified location in the drawing. To do so, choose the **Copy Component** button from the **Edit Components** panel of the **Schematic** tab; you will be prompted to select component to copy. Select the component to be copied; you will be prompted to specify the insertion point for the copied component. Specify the insertion point; the **Insert / Edit Component** dialog box will be displayed, as shown in Figure 6-6. The options in this dialog box have already been discussed in Chapter 5.

Next, enter the required information in this dialog box and choose the **OK** button; the values along with the copied component will be displayed in your drawing. If you do not change the values displayed in the **Insert / Edit Component** dialog box, the information of the original component will be transferred to the copied component. Also, the copied component will be retagged automatically and wire numbers will get updated accordingly.



Note

*If the selected component that you want to copy is a child component, the **Insert / Edit Child Component** dialog box will be displayed instead of the **Insert / Edit Component** dialog box.*

*You can also copy the components using the **COPY** command of AutoCAD but in this case, the components will not get retagged and the wires will not get reconnected automatically.*

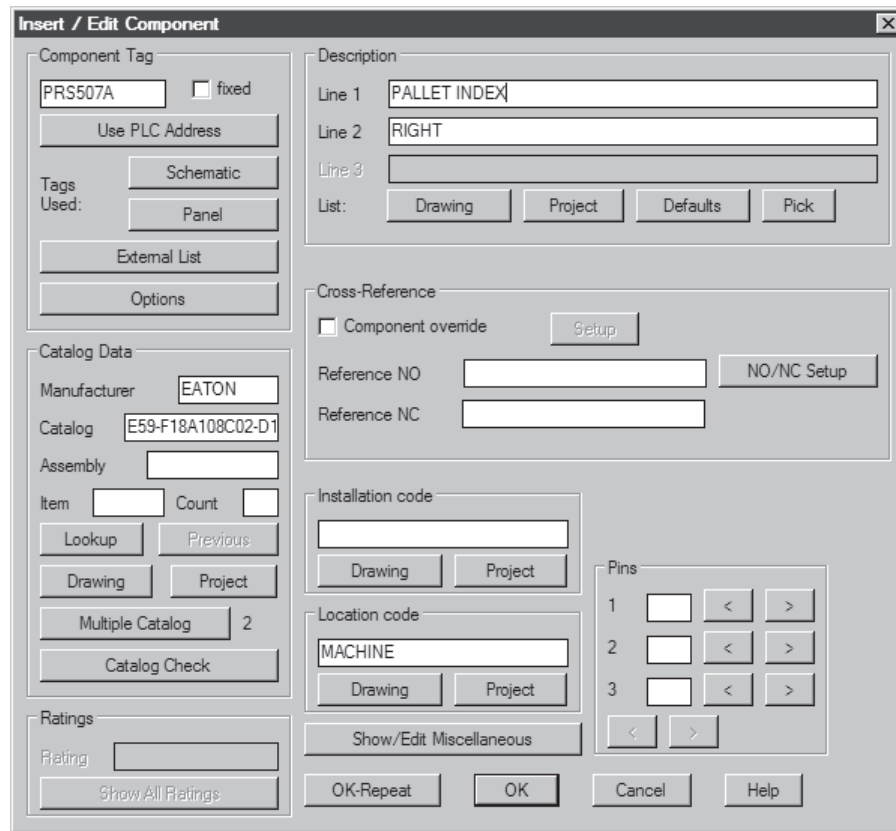


Figure 6-6 The Insert / Edit Component dialog box

ALIGNING COMPONENTS

Ribbon: Schematic > Edit Components > Scoot > Align
Toolbar: ACE:Main Electrical > Scoot > Align or ACE:Scoot > Align
Menu: Components > Align
Command: AEALIGN



The **Align** tool is used to line up components vertically or horizontally. To do so, choose the **Align** button from the **Scoot** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Align** from the menu bar; you will be prompted to select the components to be aligned. Select the component; a temporary line passing through the center of the component will be displayed. This component is treated as the reference or master component with which you need to align the rest of the components that you select. Next, select the components to align with the master component and then press the ENTER key; all the selected components will be aligned to the master component.

Figure 6-7 (a) shows the components before alignment, Figure 6-7 (b) shows the master component along with a dashed line, and Figure 6-7 (c) shows the components after alignment.

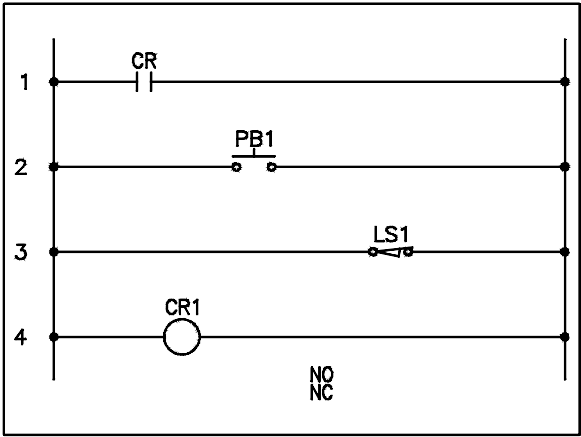


Figure 6-7(a) Components before alignment

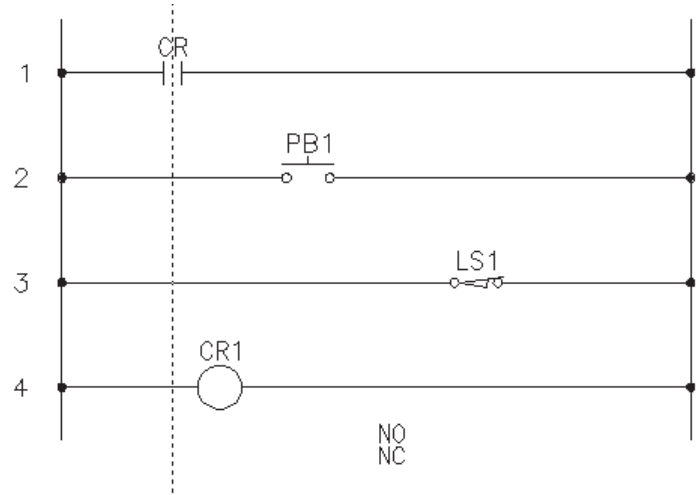


Figure 6-7 (b) The master component along with a dashed line

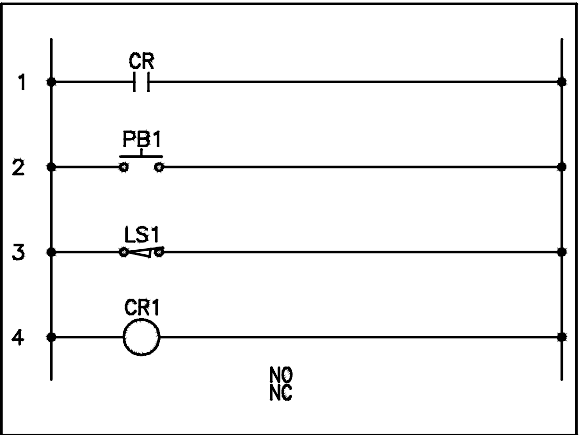


Figure 6-7(c) Components after alignment

DELETING COMPONENTS

Ribbon:	Schematic > Edit Components > Delete Component
Toolbar:	ACE:Main Electrical > Delete Component
Menu:	Components > Delete Component
Command:	AEERASECOMP



The **Delete Component** tool is used to delete a component from the drawing. Also, after deleting the component, the wires get reconnected and the wire numbers get updated. To delete a component, choose the **Delete Component** button from the **Edit Components** panel of the **Schematic** tab or choose **Components > Delete Component** from the menu bar; you will be prompted to select components. Select the component(s) and press the ENTER key or right-click on the screen; the component(s) will be deleted.

If you select a parent component that has child contacts, the **Search for / Surf to Children?** dialog box will be displayed, as shown in Figure 6-8.

Choose the **No** button in the **Search for / Surf to Children?** dialog box to delete the component. If you choose the **OK** button, the **QSAVE** dialog box will be displayed, as shown in Figure 6-9.

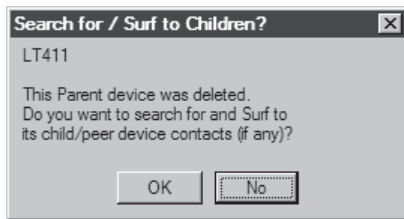


Figure 6-8 The **Search for / Surf to Children?** dialog box

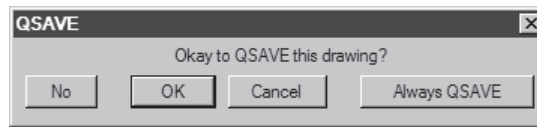


Figure 6-9 The **QSAVE** dialog box

Next, choose the **OK** button from the **QSAVE** dialog box; the active drawing will be saved. After choosing the **OK** button, the component will be deleted and the drawing will be updated. But if the selected component has its references in the current drawing and you choose the **OK** button in the **QSAVE** dialog box, the **Surf** dialog box will be displayed. The options in this dialog box are used to modify or delete the related components. The options in the **Surf** dialog box are discussed later in this chapter.

If you choose the **No** button from the **QSAVE** dialog box, the drawing will not be saved and the component will be deleted. But if the selected component has its references in the current drawing and you choose the **No** button in the **QSAVE** dialog box, the **Surf** dialog box will be displayed.

If you choose the **Always QSAVE** button, AutoCAD Electrical will save the active drawing for that session without displaying the **QSAVE** dialog box each time.

If you choose the **Cancel** button, the component will be deleted and the references of the component will not be displayed.

SURFING A REFERENCE

Ribbon: Project > Other Tools > Surfer
Toolbar: ACE:Main Electrical 2 > Surfer or ACE:Surfer
Menu: Projects > Surfer
Command: AESURF



The **Surfer** tool is used to search for the related references of a component, wire number, terminal, cables, and signal. Using this tool, you can move from one reference to another across the drawings of a project. This tool is also used to search for component tag, wire number, and catalog number in reports, and so on. To search the related references of a component, choose the **Surfer** button from the **Other Tools** panel of the **Project** tab; you will be prompted to select the tag for surfer trace. Select the component or the component tag; the **Surf** dialog box will be displayed, as shown in Figure 6-10. This dialog box displays the name of component, manufacturer code, and catalog number of the selected component at the top. Different options in this dialog box are discussed next.

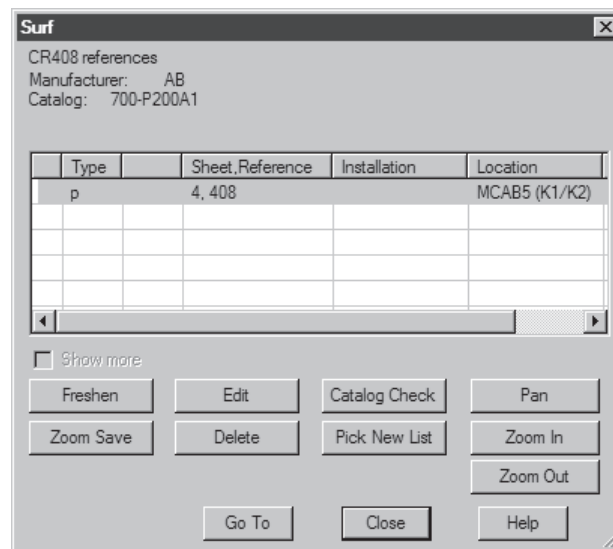


Figure 6-10 The **Surf** dialog box

Type

Different codes for different types of symbols are displayed in the **Type** column. These codes are:

- c - Component symbol
- p - Parent symbol or one-line symbol. Note that in case of one-line symbol, '1-' will be displayed in the **Category** column in the **Surf** dialog box
- t - Terminal symbol
- w - Wire number
- # - Panel symbol
- #np - Panel nameplate

Dst - Destination arrow

Src - Source arrow

Sheet,Reference

The sheet number and line reference number of the component is displayed in the **Sheet,Reference** column.

Installation

The installation value of the selected component is displayed in the **Installation** column.

Location

The location value of the selected component is displayed in the **Location** column.

Category

The category of the selected component is displayed in the **Category** column such as '1-' in case of one-line symbols.

Show more

The **Show more** check box will be available only if you are in the IEC tagging mode or if any non-installation/location matching references are not present.

Select the **Show more** check box above the **Freshen** button of the dialog box to display extra non-installation/location matching reference if you are using the IEC tagging mode.

By default, the **Show more** check box is clear. As a result, only the exact surf matches in the list will be displayed.

Freshen

The **Freshen** button is used to refresh the **Surf** dialog box with the changes made in the active drawing. To refresh the **Surf** dialog box, select a reference from the list displayed and choose the **Freshen** button; the **QSAVE** dialog box will be displayed. Next, choose the **OK** button in the **QSAVE** dialog box to save the changes made in the drawing; the **Surf** dialog box will be displayed again. If you choose the **NO** button from the **QSAVE** dialog box, the active drawing will not be saved and the **Surf** dialog box will be displayed again.

Edit

The **Edit** button is used to edit the selected reference. To do so, choose the **Edit** button; the **Insert / Edit Component** dialog box will be displayed. Using this dialog box, you can edit references. The options in the **Insert / Edit Component** dialog box have been discussed in the earlier chapters.

Catalog Check

The **Catalog Check** button will be available only if the reference has catalog and manufacturer information added to it. This button is used to display the Bill of Material of

the selected reference. To do so, choose the **Catalog Check** button; the **Bill of Material Check** dialog box will be displayed. This dialog box displays the BOM of the selected reference.

Pan

The **Pan** button allows you to view the portion of the drawing that is outside the current display area.



Note

*When the **Tool Palettes - All Palettes** is docked, the **Pan** button does not work.*

Zoom Save

The **Zoom Save** button is used to save the current zoom factor on the WD_M block of the drawing. To do so, choose the **Zoom Save** button; the Command prompt will display “**Surf zoom factor saved on WD_M block**”.

Zoom In

Choose the **Zoom In** button to double the size of the drawing.

Zoom Out

Choose the **Zoom Out** button to decrease the size of the drawing by half.

Pick New List

The **Pick New List** button is used to select a new component to surf. To do so, choose the **Pick New List** button; you will be prompted to select a tag for surfer trace. Select the component; the **Surf** dialog box will be displayed again and the references of the new selected component will be displayed in the list.

Delete

The **Delete** button is used to delete the selected component. To do so, choose the **Delete** button; the **Delete: Are you sure?** dialog box will be displayed, as shown in Figure 6-11. Choose the **OK-Delete it** button from this dialog box; the active component will be deleted and you will return to the **Surf** dialog box again. Note that the child and the other related devices will not be deleted.

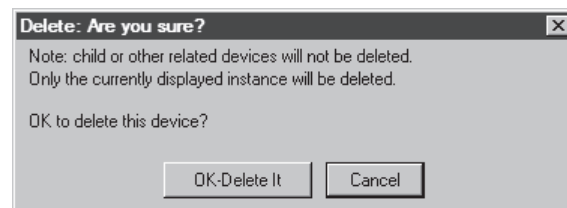


Figure 6-11 The **Delete: Are you sure?** dialog box

Go To

To go to the reference of the selected component, choose the **Go To** button. This button zooms the selected component. If the component is in a different drawing, the drawing that is opened will be saved and other drawing where the component is located will be opened and the selected component will be zoomed. Also, note that when you select the reference

from the list displayed and choose the **Go To** button; the selected reference will be zoomed and in the **Surf** dialog box, 'x' will be displayed in the left of the **Type** column in the **Surf** dialog box.

Close

Choose the **Close** button to close the **Surf** dialog box.



Note

*If you select the source or destination symbols in the drawing using the **Surfer** tool, then the **Surf** dialog box will look a little bit different. However, the options will be the same.*

TOGGLING BETWEEN THE NORMALLY OPEN AND NORMALLY CLOSED CONTACTS

Ribbon:	Schematic > Edit Components > Toggle NO/NC
Toolbar:	ACE:Main Electrical > Toggle NO/NC
Menu:	Components > Toggle NO/NC
Command:	AETOGGLENONC



The **Toggle NO/NC** tool is used to toggle contacts from normally open to normally closed and vice-versa. In other words, this tool is used to flip contacts from one state to the other (open/closed). To toggle between contacts, choose the **Toggle NO/NC** button from the **Edit Components** panel of the **Schematic** tab or choose **Components > Toggle NO/NC** from the menu bar; you will be prompted to select the component to toggle NO/NC. Select the component; you will notice that if the component is normally open (NO), it will change to normally closed (NC) and vice-versa. Also, the attribute information will be extracted from the selected component and the component data will be copied onto the replacement symbol. The command will continue till you press ENTER or right-click on the screen.

Figures 6-12 (a) and 6-12 (b) show the contacts of components before and after toggling.

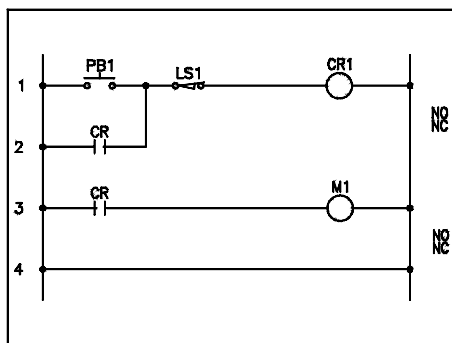


Figure 6-12(a) Contacts of components before toggling

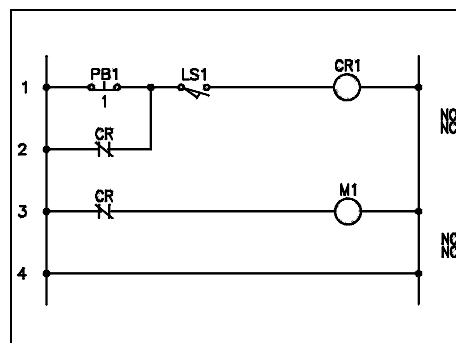


Figure 6-12(b) Contacts of components after toggling

COPYING THE CATALOG ASSIGNMENT

Tool Palettes:	Schematic > Edit Component > Edit > Copy Catalog Assignment
Toolbar:	ACE:Main Electrical > Edit Component > Copy Catalog Assignment or ACE:Edit Component > Copy Catalog Assignment
Menu:	Components > Component Miscellaneous > Copy Catalog Assignment
Command:	AECOPYCAT



The **Copy Catalog Assignment** tool is used to copy the catalog part numbers from one component to another. Also, this tool is used to insert or edit the catalog data of the selected component or footprint. This tool helps in assigning the same catalog information to multiple components. To copy manufacturer, catalog, assembly, and multiple catalog attributes from one component to another selected components, choose the **Copy Catalog Assignment** button from the **Edit** flyout of the **Edit Components** panel of the **Schematic** tab; you will be prompted to select the master component. Select the component to be treated as master component; the **Copy Catalog Assignment** dialog box will be displayed, as shown in Figure 6-13. This dialog box displays the catalog data of the selected component. The options in the **Copy Catalog Assignment** dialog box are discussed next.

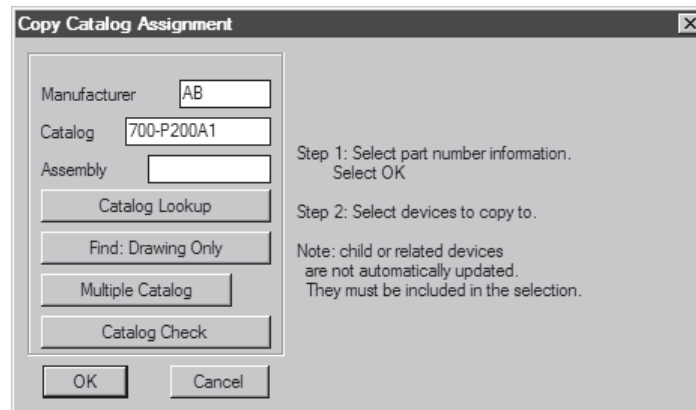


Figure 6-13 The Copy Catalog Assignment dialog box



Note

If the catalog assignment is not assigned to the master component, the **Copy Catalog Assignment** dialog box will not display any catalog data. Also, if you want to modify the catalog assignment both for the master and child components, choose the **Catalog Lookup** button from the **Copy Catalog Assignment** dialog box; the **Parts catalog** dialog box will be displayed. Next, select the desired catalog information, and choose the **OK** button from this dialog box; the catalog data will be assigned to the selected component.

Manufacturer

In the **Manufacturer** edit box, the manufacturer data of the master component is displayed. You can edit the manufacturer data or enter new data by choosing the **Catalog Lookup** button, which is discussed later in this chapter.

Catalog

The catalog data of the master component is displayed in the **Catalog** edit box. If it is not displayed, you can enter it by choosing the **Catalog Lookup** button and then selecting the desired catalog.

Assembly

The **Assembly** edit box is used to display the assembly code of the master component.

Catalog Lookup

Choose the **Catalog Lookup** button from the **Copy Catalog Assignment** dialog box; the **Parts catalog** dialog box will be displayed, which has been discussed in the previous chapters. Select the catalog information from the **Parts catalog** dialog box for the master component and choose the **OK** button; the catalog information will be displayed in the **Manufacturer** and **Catalog** edit boxes.

Find: Drawing Only

The **Find: Drawing Only** button is used to search for the symbols of the same family block name within the active drawing. To do so, choose the **Find: Drawing Only** button from the **Copy Catalog Assignment** dialog box; the **catalog values** dialog box will be displayed. This dialog box displays catalog information of the symbols of the same family type. Select the catalog values from this dialog box and choose **OK**; the values will be displayed in the **Manufacturer** and **Catalog** edit boxes of the **Copy Catalog Assignment** dialog box.

Multiple Catalog

The **Multiple Catalog** button is used to add extra catalog information to the selected component. Also, this button is used to verify the catalog information. To add extra catalog information, choose the **Multiple Catalog** button; the **Multiple Bill of Material Information** dialog box will be displayed. In this dialog box, you can add extra catalog information. You can add up to 99 additional part numbers to any schematic or panel component. Specify the required options in this dialog box and then choose the **OK** button to return to the **Copy Catalog Assignment** dialog box; the extra catalog assignment will be displayed adjacent to the **Multiple Catalog** button.

Catalog Check

The **Catalog Check** button is used to check for the catalog information of the selected component. To do so, choose the **Catalog Check** button; the **Bill Of Material Check** dialog box will be displayed. It displays the Bill Of Material of the master component.

After specifying the options in the **Copy Catalog Assignment** dialog box, choose the **OK** button from this dialog box; you will be prompted to pick the target component(s). Select the target component(s) and press ENTER; the catalog information will be copied to the target component(s). If the target component consists of catalog data different from the master component, the **Caution: Existing Data on Target** dialog box will be displayed, as shown in Figure 6-14. Different options in this dialog box are discussed next.

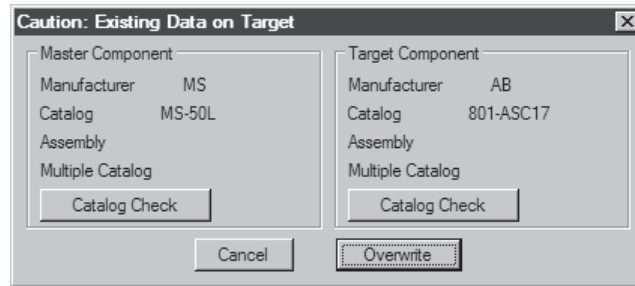


Figure 6-14 The **Caution: Existing Data on Target** dialog box

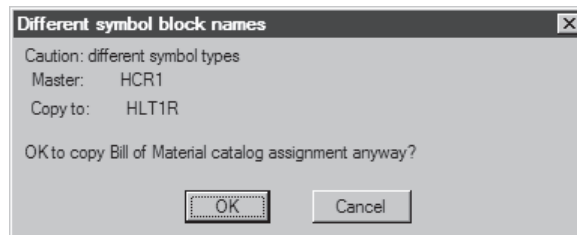
The **Catalog Check** button is used to display the catalog information of the master and target component(s).

To overwrite the catalog information of the master component to the target component(s), choose the **Overwrite** button; the catalog information of the master component will be applied to the target component(s). But, if the target component has its child contacts or references, then after choosing the **Overwrite** button, the **Update Related Components?** dialog box will be displayed. In this dialog box, choose the **Yes-Update** button in the **Update Related Components?** dialog box; the related components will be updated. Choose the **Skip** button to skip the update of the related components.



Note

If the target component has different symbol block name than the master component, then after selecting the master and target component, the **Different symbol block names** dialog box will be displayed, as shown in Figure 6-15. Choose the **OK** button from the **Different symbol block names** dialog box; the **Caution: Existing Data on Target** dialog box will be displayed.



COPYING INSTALLATION/LOCATION CODE VALUES

Ribbon:	Schematic > Edit Components > Copy Installation/Location Code Values
Toolbar:	ACE:Main Electrical 2 > Location Symbols > Copy Installation/Location Code Values or ACE:Location Symbols > Copy Installation/Location Code Values
Menu:	Components > Component Tagging > Copy Installation/Location Code Values
Command:	AECOPYINSTLOC



The **Copy Installation/Location Code Values** tool is used to copy the installation and location code assignments from one component to another. To do so, select the arrow on the right of the **Edit Components** panel; a flyout will be displayed. Next,

choose the **Copy Installation/Location Code Values** button from the flyout of the **Schematic** tab or choose **Components > Component Tagging > Copy Installation/Location Code Values** from the menu bar; the **Copy Installation/Location to Components** dialog box will be displayed, as shown in Figure 6-16. Different options in this dialog box are discussed next.

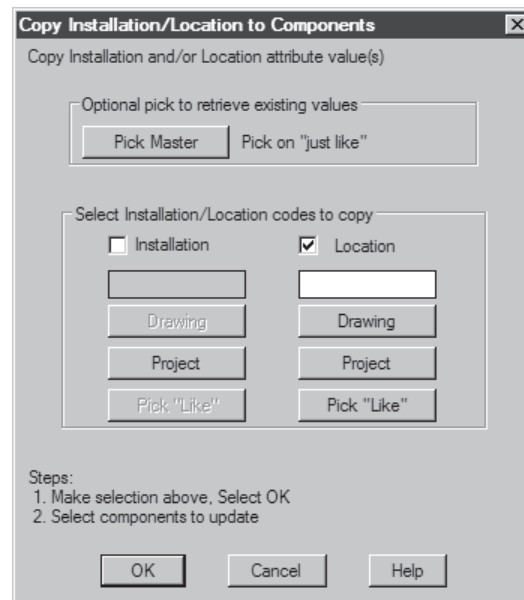


Figure 6-16 The *Copy Installation/Location to Components* dialog box

Pick Master

The **Pick Master** button is used to select the master component from where attributes will be copied. To select the master component, choose the **Pick Master** button; the **Copy Installation/Location to Components** dialog box will disappear and you will be prompted to select the master component to get the installation and location values. Select the component from the drawing; the **Copy Installation/Location to Components** dialog box will be displayed again with the installation and location values of the selected component.

Select Installation/Location codes to copy Area

This area is used to copy the installation and location codes to the component that you will select. The options in this area are discussed next.

Installation

Select the **Installation** check box; the edit box below this check box as well as the **Drawing** and the **Pick “Like”** buttons will become available. Next, enter the installation code that you want to copy from the master component to the other components in the edit box.

Drawing

The **Drawing** button is used to display the list of installation code present in the active drawing.

Project

The **Project** button is used to display the list of installation code present in the active project.

Pick “Like”

The **Pick “Like”** button is used to select the component from where installation values are copied.

Location

The **Location** check box is selected by default. As a result, the edit box below this check box as well as the **Drawing**, **Project** and the **Pick “Like”** buttons will be available. Enter the location code that you want to copy from the master component in the edit box below the **Location** check box.

Drawing

The **Drawing** button is used to display the list of location codes present in the active drawing. To select the location code present in the active drawing, choose the **Drawing** button; the **All Locations** dialog box will be displayed. Select the location code from the list used in the current drawing and then choose the **OK** button; the location code will be displayed in the edit box.

Project

The **Project** button is used to display the list of location code present in the active project. To select the location code present in the active project, choose the **Project** button; the **All Locations** dialog box will be displayed. Select the location code from the list used in the project and then choose the **OK** button; the location code will be displayed in the edit box.

Pick “Like”

The **Pick “Like”** button is used to select the component from where the location values will be copied. To do so, choose the **Pick “Like”** button; you will be prompted to select the component to get the location code. Select the component; the location code will be displayed in the edit box, which is below the **Location** check box.

After specifying the required options in the **Copy Installation/Location to Components** dialog box, choose the **OK** button; you will be prompted to select components. Select the component(s) to copy the installation/location values and press ENTER; the installation/location values will be copied to the selected component.



Note

AutoCAD Electrical does not show any warning if you overwrite the existing installation and location codes.

AUDITING DRAWINGS

The auditing tools are used to check errors in the project. These tools help in troubleshooting and improving the accuracy of a drawing. There are two auditing tools such as **Electrical Audit** and **Drawing Audit**. These tools are discussed next.

Electrical Auditing

Ribbon:	Reports > Schematic > Electrical Audit
Toolbar:	ACE:Main Electrical 2 > Schematic Reports > Electrical Audit or ACE:Schematic Reports > Electrical Audit
Menu:	Projects > Reports > Electrical Audit
Command:	AEAUDIT



The **Electrical Audit** tool is used to find out the problems that affect the drawings of a project. This tool can be used to correct some of the errors in drawings. This tool is used to display errors in the **Electrical Audit** dialog box. To find out errors, choose the **Electrical Audit** button from the **Schematic** panel of the **Reports** tab or choose **Projects > Reports > Electrical Audit** from the menu bar; the **Electrical Audit** dialog box will be displayed, as shown in Figure 6-17. The progress of the electrical audit will be displayed in the edit box. Once the audit is finished, an edit box will display the total number of errors that occurred in the active project. In this dialog box, the **Project** radio button is selected by default. As a result, the total number of errors found in the active project will be displayed in the edit box of the **Electrical Audit** dialog box. If you select the **Active Drawing** radio button, then the total number of errors found in the active drawing will be displayed in the edit box. This dialog box also displays the date and time of electrical audit report.

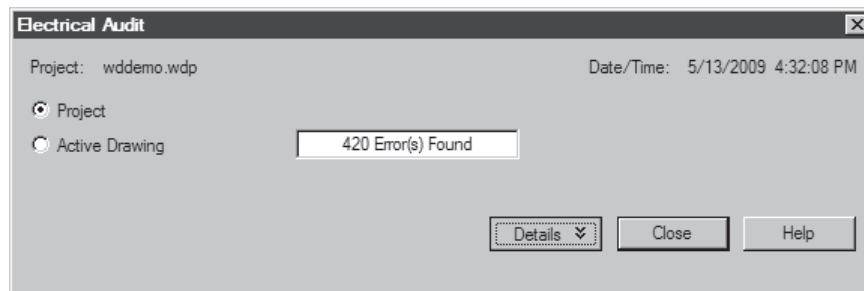


Figure 6-17 The **Electrical Audit** dialog box

The **Details** button is used to view errors as well as to expand or collapse the **Electrical Audit** dialog box. To view errors and to expand the **Electrical Audit** dialog box, choose the **Details** button; the **Electrical Audit** dialog box will get expanded and display the detailed information about the errors found in the project, as shown in Figure 6-18. If you choose this button again, the **Electrical Audit** dialog box will be collapsed, refer to Figure 6-17.



Note

*If the active drawing is not a part of the active project, then the **Active Drawing** radio button will not be available in the **Electrical Audit** dialog box.*

*If errors are not found in a project or active drawing, then the **Details** button in the **Electrical Audit** dialog box will not be available.*

The expanded **Electric Audit** dialog box has ten different tabs. If the tab has a red circle and a white 'x', it means an error is present in that category. To view the errors present in a tab, choose the tab; the information about the errors will be displayed in the lower part of the **Electrical Audit** dialog box.

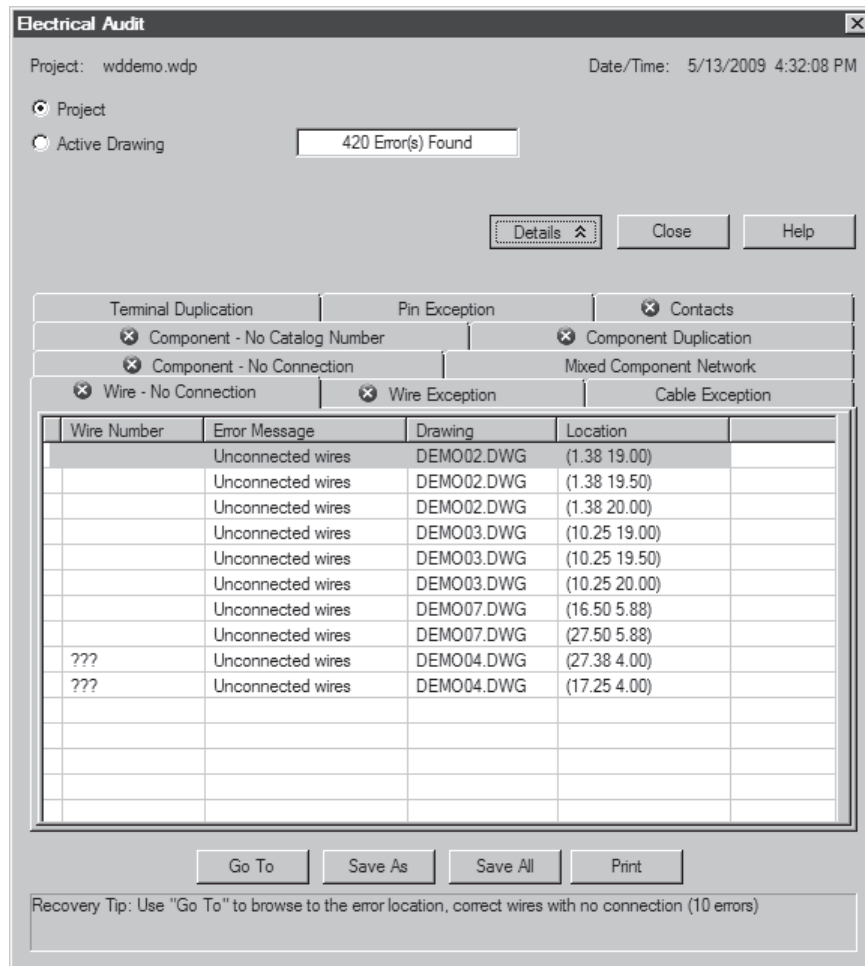


Figure 6-18 The expanded **Electrical Audit** dialog box displaying errors on choosing the **Details** button

Different tabs in the **Electrical Audit** dialog box are discussed next.

Wire - No Connection

By default, the **Wire - No Connection** tab is chosen in the **Electrical Audit** dialog box. This tab displays the disconnected wires that are found in the project. The report list below this tab consists of disconnected wire number, error message, drawing, and the location point where an error occurs. If no wire number record is found in the drawings of a project, the **Wire Number** column will be blank.

Wire Exception

The **Wire Exception** tab displays the missing or the duplicated wire numbers that are found in an active project. To view wire exception errors, choose this tab; the report list will be displayed, containing missing or duplicated wire number, error message, and the name of the drawing where an error occurs. If no wire number record is found in the drawing, the **Wire Number** column will be left blank.

Cable Exception

The **Cable Exception** tab displays the duplicated cable and the wire ID's that are found in an active project. To view cable exception errors, choose this tab; the report list will be displayed containing duplicated cable tags or cable tags with duplicated wire ID, error message, reference of cable tag, and the name of the drawing where an error occurs.

Component - No Catalog Number

The **Component - No Catalog Number** tab displays the components that have no catalog data. To view components that do not have catalog data, choose the **Component - No Catalog Number** tab; the report list will be displayed containing component tag, category of the component, reference of component tag, error message, and the name of the drawing where an error occurs.

Component Duplication

The **Component Duplication** tab displays the duplicated schematic/panel components. To view the duplicated components, choose the **Component Duplication** tab; the report list will be displayed containing tag name of component, category of the component, error message, and the name of the drawing where an error occurs.

Component - No Connection

The **Component - No Connection** tab displays the component connections with disconnected wires. To view the component connections with no connected wires, choose the **Component - No Connection** tab; the report list will be displayed containing component tag, category of the component, reference of the component tag, error message, and the name of the drawing where an error occurs.



Note

When the **Category** column in the **Electrical Audit** dialog box is left blank, it indicates a schematic component.

Mixed Component Network

The **Mixed Component Network** tab displays components in a wire network that consists of components of different types such as one-line symbols, schematic symbols, and so on. To view the mixed components in a wire network, choose the **Mixed Component Network** tab; the report list will be displayed containing the tag name of the component (from/to), component type (from/to), error message, and the name of the drawing (from/to) where an error occurs.

Terminal Duplication

The **Terminal Duplication** tab displays the duplicated schematic terminal numbers. To view the duplicated schematic terminal, choose the **Terminal Duplication** tab; the report list will be displayed containing tag ID of the terminal strip, terminal number, wire number of terminal, error message, and the name of the drawing where an error occurs.

Pin Exception

The **Pin Exception** tab displays the pin assignments of a component that has been duplicated.

To view the duplicated component pin assignments, choose the **Pin Exception** tab; the report list will be displayed containing component tag, terminal pin number, error message, and the name of drawing where an error occurs.

Contacts

The **Contacts** tab displays a child component without a parent. To view a child component without a parent component, choose the **Contacts** tab; the report list will be displayed containing the tag name of the child component, reference of the child component tag, error message, and the name of the drawing where an error occurs.

Recovery Tip

The **Recovery Tip** area at the bottom of the **Electrical Audit** dialog box displays the recovery tip to fix an error in the drawing.

Go To

The **Go To** button takes you to an error location within the project and corrects the error. To go to an error location, select the error from the report list displayed in the **Electrical Audit** dialog box and then choose the **Go To** button. Alternatively, double-click on the error displayed in the **Electrical Audit** dialog box; the current drawing will be saved. Also, the drawing that contains the selected error will be opened and the selected component will be zoomed. Now, you can edit the selected component without closing the **Electrical Audit** dialog box. To do so, right-click on the selected component in the drawing area; a shortcut menu will be displayed. Choose the required component editing option from the shortcut menu; the corresponding editing dialog box will be displayed. In this dialog box, specify the required options and then choose the **OK** button to exit the dialog box. Once you go through an error location, an 'x' will appear in the extreme left column of the **Electrical Audit** dialog box.

Save As

The **Save As** button is used to save an active audit report to a text file.

Save All

The **Save All** button is used to save a complete audit report to a text file.

Print

The **Print** button is used to print an audit report. To print an audit report, choose the **Print** button; the **Print** dialog box will be displayed, as shown in Figure 6-19. Select the name of the printer from the **Name** drop-down list and then choose the **OK** button; the audit report will be printed.

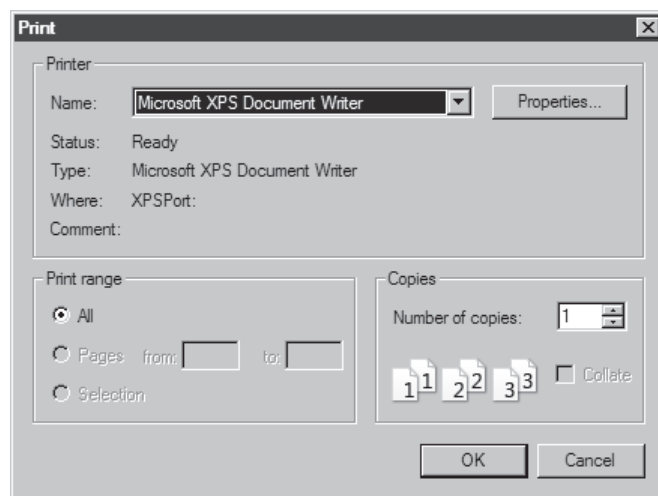


Figure 6-19 The **Print** dialog box

Auditing a Drawing

Ribbon:	Reports > Schematic > DWG Audit
Toolbar:	ACE:Main Electrical 2 > Schematic Reports > Drawing Audit or ACE:Schematic Reports > Drawing Audit
Menu:	Projects > Reports > Drawing Audit
Command:	AEAUDITDWG



The **DWG Audit** tool is used to find out the problems in wiring that affect the wire connectivity of a design. Using this tool, you can audit a single drawing or multiple drawings in a project. The auditing of a drawing is performed to check for wire gaps, wire number, color, zero length wires, gauge labels, and wire number floaters for errors, and so on. To find out errors in wires and to rectify them, choose the **DWG Audit** button from the **Schematic** panel of the **Reports** tab or choose **Projects > Reports > Drawing Audit** from the menu bar; the **Drawing Audit** dialog box will be displayed, as shown in Figure 6-20. Different options in the **Drawing Audit** dialog box are discussed next.

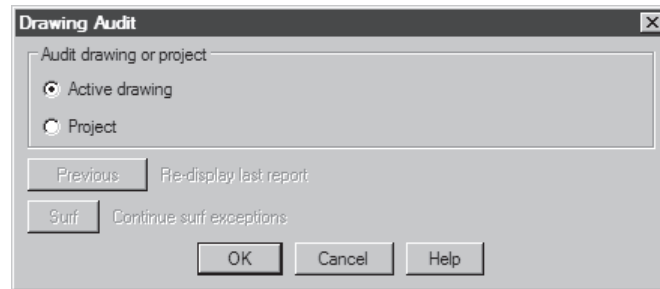


Figure 6-20 The **Drawing Audit** dialog box

The **Audit drawing or project** area consists of two radio buttons: **Active drawing** and **Project**. These radio buttons are discussed next.

Auditing an Active Drawing

The **Active drawing** radio button is selected by default. This radio button is used to audit an active drawing only. Choose the **OK** button from the **Drawing Audit** dialog box; the modified **Drawing Audit** dialog box will be displayed, as shown in Figure 6-21. The options in the modified **Drawing Audit** dialog box are discussed next.

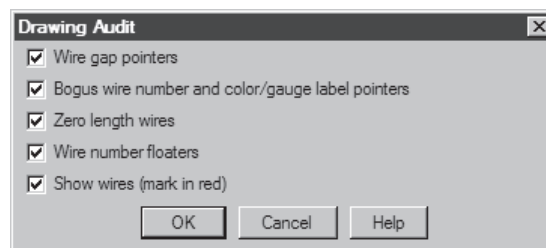


Figure 6-21 The modified **Drawing Audit** dialog box

Wire gap pointers

The **Wire gap pointers** check box is selected by default. This check box is used to find out the problems related to the missing wires that were connected through gap pointers.

Bogus wire number and color/gauge label pointers

The **Bogus wire number and color/gauge label pointers** check box is selected by default. This check box is used to check for and clean up the wires that have non-existent wire numbers. Also, this check box is used to check for bad color/gauge label pointers.

Zero length wires

The **Zero length wires** check box is selected by default and is used to check and erase the zero length line entities present on a wire layer.

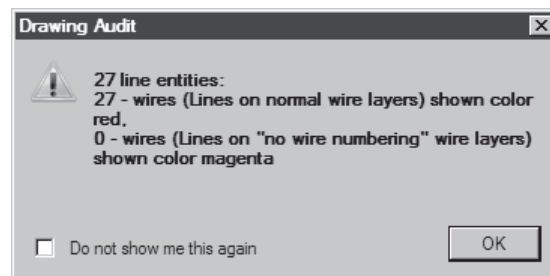
Wire number floaters

The **Wire number floaters** check box is selected by default and is used to check and erase the wire numbers that are not linked to a wire network.

Show wires (mark in red)

The **Show wires (mark in red)** check box is selected by default and is used to display a red outline around each wire entity and magenta outline around the wires defined on the no wire numbering layers. This check box will be available only if you select the **Active drawing** radio button from the **Drawing Audit** dialog box.

After specifying the options from the modified **Drawing Audit** dialog box, choose the **OK** button in this dialog box; the **Drawing Audit** message box will be displayed, as shown in Figure 6-22. Choose the **OK** button in this message box; the **Report: Audit for this drawing** dialog box will be displayed, as shown in Figure 6-23. You can save and print a report by choosing the **Save As** button and the **Print** button from this dialog box, respectively.



*Figure 6-22 The **Drawing Audit** message box*

Previous

The **Previous** button of the **Drawing Audit** dialog box is used to display the previous audit report. To do so, choose the **Previous** button; the **Report: Audit** dialog box will be displayed. This dialog box re-displays the last run audit report.

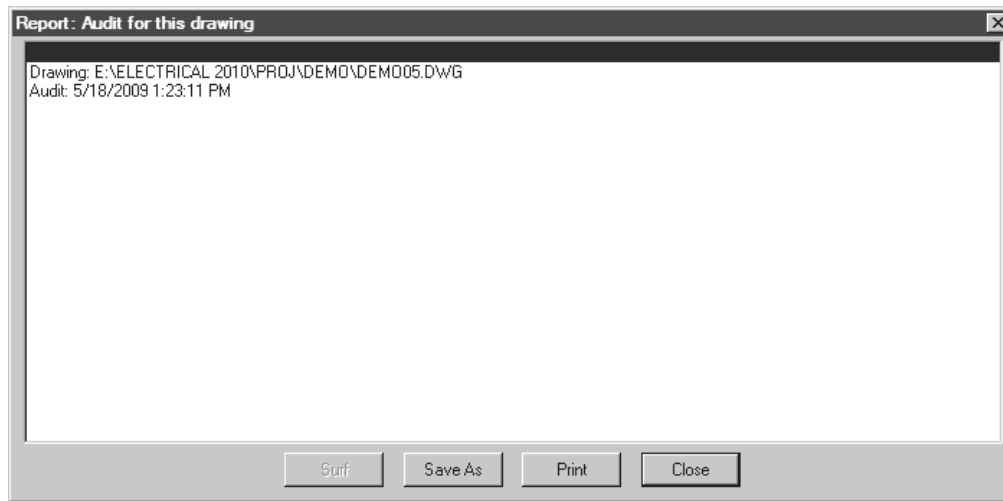


Figure 6-23 The **Report: Audit for this drawing** dialog box

Auditing an Active Project

The **Project** radio button is used to audit an active project. To do so, select the **Project** radio button from the **Active drawing or project** area of the **Drawing Audit** dialog box, refer to Figure 6-20, and then choose the **OK** button from this dialog box; the modified **Drawing Audit** dialog box will be displayed, as shown in Figure 6-24. The options in the modified **Drawing Audit** dialog box have already been discussed.

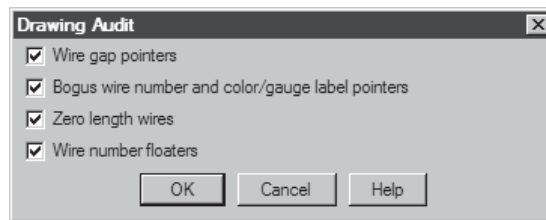


Figure 6-24 The modified **Drawing Audit** dialog box

Next, choose the **OK** button from the modified **Drawing Audit** dialog box; the **Select Drawings to Process** dialog box will be displayed, as shown in Figure 6-25. Select the drawings that you want to process and choose the **Process v** button from this dialog box; the selected drawings will be moved from the top list to the bottom list. Next, choose the **OK** button in this dialog box; the **QSAVE** dialog box will be displayed, as shown in Figure 6-26. Next, choose the **OK** button in the **QSAVE** dialog box; the changes made in the current drawing will be saved and the auditing report of drawings will be displayed in the **Report: Audit** dialog box, as shown in the Figure 6-27.



Note

The **QSAVE** dialog box will be displayed only if you have made any changes in the current drawing and have not saved them.

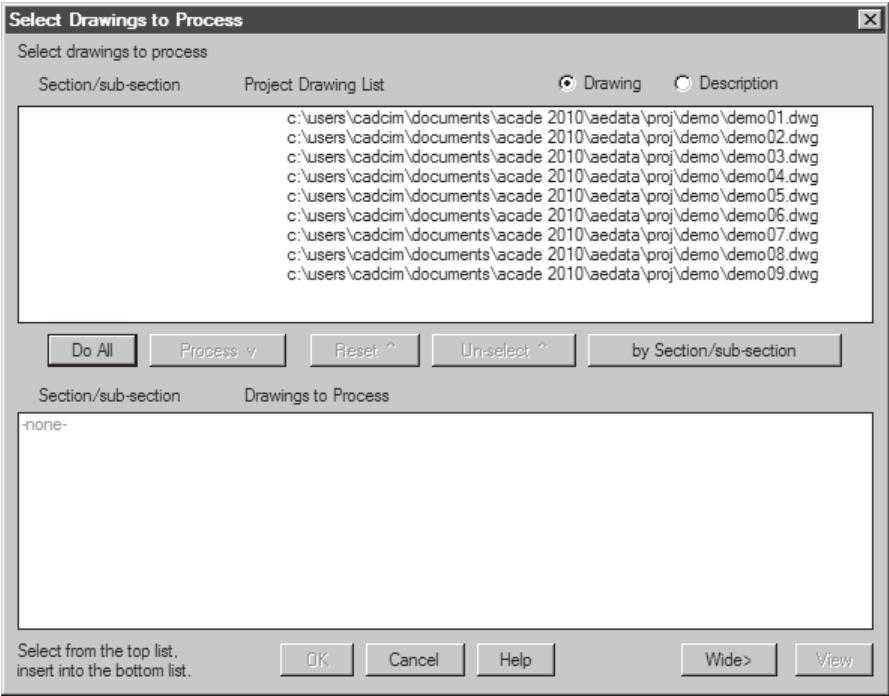


Figure 6-25 The Select Drawings to Process dialog box

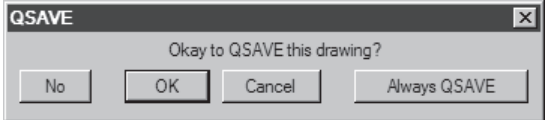


Figure 6-26 The QSAVE dialog box

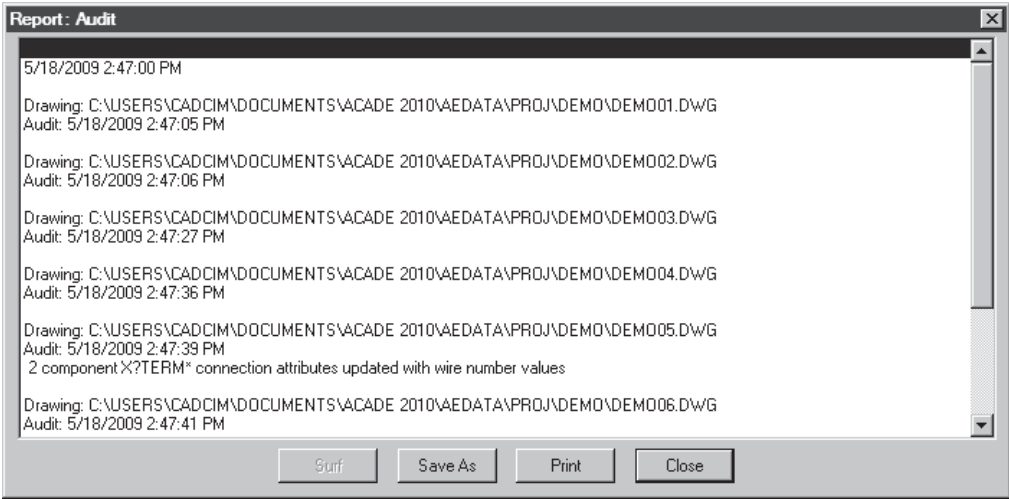


Figure 6-27 The Report: Audit dialog box

RETAGGING DRAWINGS

Ribbon:	Project > Project Tools > Update/Retag
Toolbar:	ACE:Main Electrical 2 > Project Manager > Project-Wide Update/Retag ACE:Project > Project-Wide Update/Retag
Menu:	Projects > Project-Wide Update/Retag
Command:	AEPROJUPDATE



The **Project-Wide Update/Retag** tool is used to update the selected drawings in a project. Also, this tool is used to retag components, update the cross-reference of component, retag wire numbers and signal references, change ladder reference, change drawing settings, update title blocks, and so on. To update or retag the drawings of a project, choose the **Update/Retag** button from the **Project Tools** panel of the **Project** tab or choose **Projects > Project-Wide Update/Retag** from the menu bar; the **Project-Wide Update or Retag** dialog box will be displayed, as shown in Figure 6-28. Different options in this dialog box are discussed next.

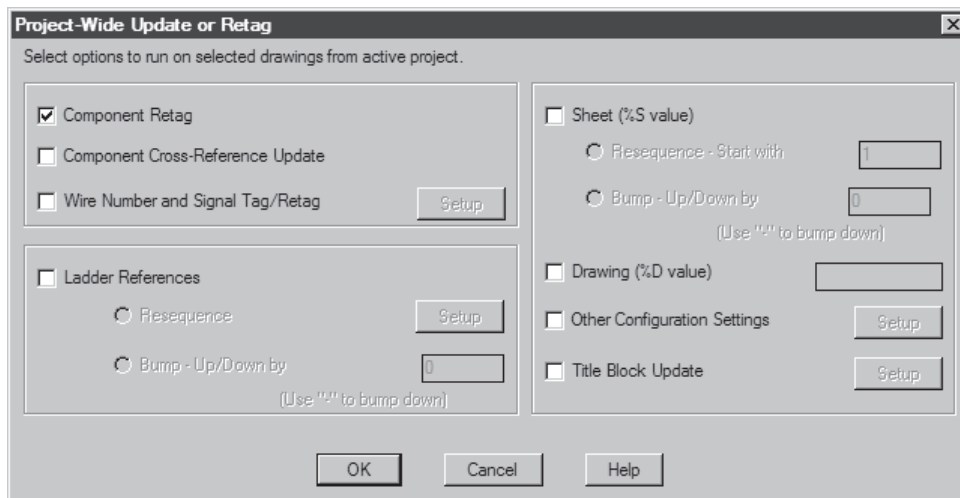


Figure 6-28 The Project-Wide Update or Retag dialog box

Component Retag

The **Component Retag** check box is selected by default and is used to retag all the non-fixed components of the selected drawings.

Component Cross-Reference Update

The **Component Cross-Reference Update** check box is used to update the cross-reference of components of the selected drawings. By default, this check box is cleared.

Wire Number and Signal Tag/Retag

The **Wire Number and Signal Tag/Retag** check box is used to update signal symbols and the wire numbers that are not fixed. By default, the **Wire Number and Signal Tag/Retag** check

box is selected. As a result, the **Setup** button will be available. Using this button, you can insert or update wire numbers that are linked to wire networks. To do so, choose the **Setup** button; the **Wire Tagging (Project-wide)** dialog box will be displayed, as shown in Figure 6-29. The options in this dialog box have been discussed in the previous chapters. Specify the required options in this dialog box and then choose the **OK** button to return to the **Project-Wide Update or Retag** dialog box.

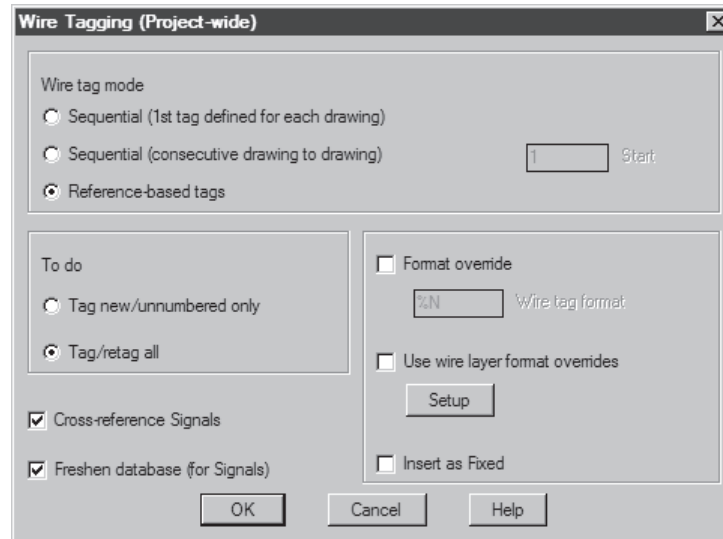


Figure 6-29 The Wire Tagging (Project-wide) dialog box

Ladder References

The **Ladder References** check box is used to renumber the ladders sequentially. To renumber each ladder of drawings sequentially, select this check box; the options below the **Ladder References** check box will become available. These options are discussed next.

Resequence

By default, the **Resequence** radio button is selected and is used to define the starting reference number for a ladder. Also, it is used to define the sequence of ladders in different drawings. The **Setup** button located on the right of the **Resequence** radio button is used to renumber ladder reference. To do so, choose the **Setup** button; the **Renumber Ladders** dialog box will be displayed, as shown in Figure 6-30. Using this dialog box, you can renumber the reference numbers of a ladder of the selected drawings in an active project. The options in this dialog box have already been discussed in detail in the previous chapter.

Enter the ladder reference number for the 1st ladder of the 1st drawing in the **1st drawing, 1st ladder** edit box.

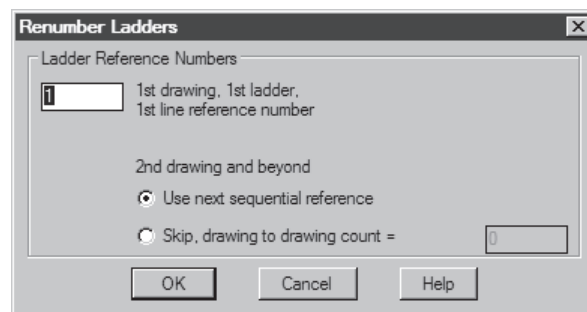


Figure 6-30 The Renumber Ladders dialog box

By default, the **Use next sequential reference** radio button is selected. As a result, next sequential reference number will be used for all the drawings starting from 2nd drawing onwards.

Select the **Skip, drawing to drawing count** radio button; the edit box located to the right of this radio button will be available. In this edit box, enter an amount that you want to skip for the next drawing's first ladder reference. Specify the options in this dialog box and choose the **OK** button to return to the **Project-Wide Update or Retag** dialog box.

Bump-Up/Down by

The **Bump-Up/Down by** radio button is used to increase or decrease the existing ladder line reference numbers by a specified amount. To do so, select this radio button; the edit box on the right of this radio button will be available. Specify a value in this edit box; the reference number of the ladder will change accordingly. Also, if you add drawings in the middle of a project, the ladder references will move up and if you remove drawings from the project, the ladder references will move down. If you enter a negative value in the edit box, the line reference number of the ladder will move down.

Sheet (%S value)

If you select the **Sheet (%S value)** check box, the options below this check box will be available. This check box is used to resequence the sheet value automatically in the successive drawings.

Resequence - Start with

The **Resequence - Start with** radio button is selected by default and it enables you to enter a number to start resequencing.

Bump - Up/Down by

Select the **Bump - Up/Down by** radio button; the edit box adjacent to this radio button will be available. Next, enter the required value in the edit box; the current sheet value will move up or down by the specified value.

Drawing (%D value)

The **Drawing (%D value)** check box is used to update drawing's %D (drawing name) parameter project-wide. Select this check box; the edit box adjacent to this check box will be available.

Other Configuration Settings

The **Other Configuration Settings** check box is used to update the drawing settings of the drawing related to component, cross-reference, wire numbers, and format project-wide. Select this check box; the **Setup** button will be available. Choose the **Setup** button; the **Change Each Drawing's Settings -- Project-wide** dialog box will be displayed, as shown in Figure 6-31. The options in this dialog box are used to change the settings of the drawings. These options are discussed next.

Component Tagging Settings

The **Component Tagging Settings** check box is used to specify the settings for the component tag. If you select the **Component Tagging Settings** check box, the **Sequential**

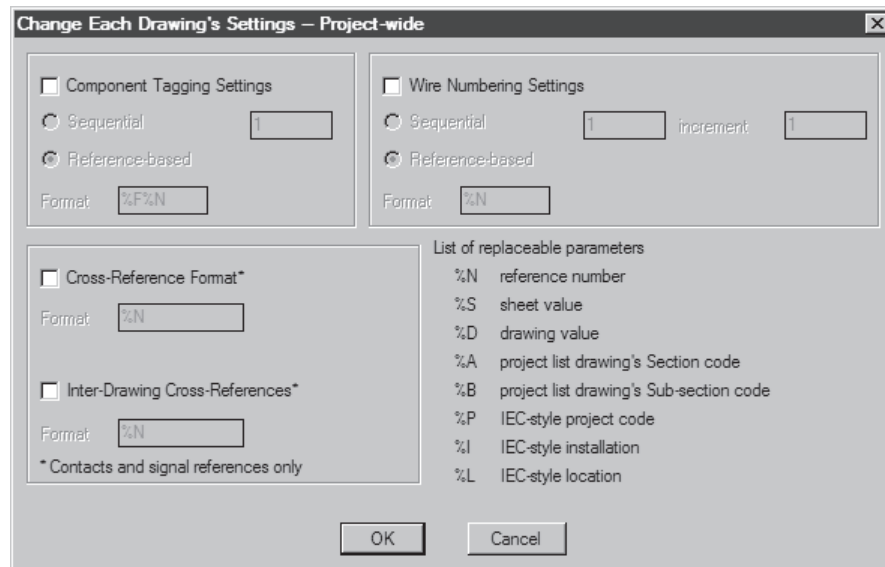


Figure 6-31 The *Change Each Drawing's Settings -- Project-wide* dialog box

radio button, the **Reference-based** radio button, and the **Format** edit box will be available. Select the **Sequential** radio button for tagging the components sequentially. The **Reference-based** radio button is selected by default. As a result, the components are tagged based on reference of the ladder. Enter the format for component tagging in the **Format** edit box. By default, %F%N is displayed in this edit box.

Cross-Reference Format

The **Cross-Reference Format** check box is used to specify the format for cross-reference of the component. Select this check box; the **Format** edit box will be available. Enter the format for cross-reference in the **Format** edit box. By default, %N is displayed in this edit box.

Inter-Drawing Cross-References

The **Inter-Drawing Cross-Reference** check box is used to specify the format for the inter-drawing cross-references. Select this check box; the **Format** edit box will be available. Enter the format of cross-reference for inter-drawing cross-reference in the **Format** edit box. By default, %N is displayed in this edit box.

Wire Numbering Settings

The **Wire Numbering Settings** check box is used to specify the settings for the wire numbers. Select this check box; the other options will be available. These options are discussed next.

Select the **Sequential** radio button for numbering the wires sequentially. You can enter the increment value in the **increment** edit box for wire numbering.

Select the **Reference-based** radio button for numbering the wires based on reference number of the ladder. Enter the format of wire numbers in the **Format** edit box. By default, %N is displayed in this edit box.

Title Block Update

Select the **Title Block Update** check box to update the title block information of the active drawing or the entire project. Also, the **Setup** button will be available. Choose the **Setup** button; the **Update Title Block** dialog box will be displayed. Using this dialog box, you can update the attributes of the title block. The options in this dialog box will be discussed in the later chapters. Specify the required options in the **Update Title Block** dialog box and choose the **OK** button, you will return to the **Project-Wide Update or Retag** dialog box.

You can use any one of the options in the **Project-Wide Update or Retag** dialog box or use all of them simultaneously by selecting the options displayed in this dialog box. After selecting the options, choose the **OK** button from the **Project-Wide Update or Retag** dialog box; the **Select Drawings to Process** dialog box will be displayed. Select the required drawings and choose the **Process** button; the selected drawings will be displayed in the bottom list. Next, choose the **OK** button from this dialog box; the selected drawings will be processed and updated accordingly.

USING TOOLS FOR EDITING ATTRIBUTES

In AutoCAD Electrical, there are number of tools that can be used to modify attributes. These tools are used to modify the attributes of selected symbols but they are not used to modify the block. In this section, you will learn about different editing tools of attributes.

Moving Attributes

Ribbon:	Schematic > Edit Components > Move/Show Attribute
Toolbar:	ACE:Main Electrical > Move/Show Attribute or ACE>Edit Attributes > Move/Show Attribute
Menu:	Components > Attributes > Move/Show Attribute
Command:	AEATTSHOW



The **Move/Show Attribute** tool is used to move attributes of a component. To do so, choose the **Move/Show Attribute** button from the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Move/Show Attribute** from the menu bar; you will be prompted to select the attribute to move or pick on block graphics for list. Select the attributes that you want to move and press ENTER; you will be prompted to specify the base point of the attribute. Specify the base point for the attribute and press ENTER or click on the screen; you will be prompted again to select the attributes to move. Press ENTER to exit the command. If you enter 'W' at the Command prompt, you will be able to move multiple attributes at a time by selecting attributes using the crossing window.

Editing Attributes

Ribbon:	Schematic > Edit Components > Move/Show Attribute > Edit Selected Attribute
Toolbar:	ACE:Main Electrical > Move/Show Attribute > Edit Selected Attribute or ACE>Edit Attributes > Edit Selected Attribute
Menu:	Components > Attributes > Edit Selected Attribute
Command:	AEEDITATT



The **Edit Selected Attribute** tool is used to edit the selected attribute. To do so, choose the **Edit Selected Attribute** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Edit Selected Attribute** from the menu bar; you will be prompted to select an attribute. Select the attribute; the **Edit Attribute** dialog box will be displayed, as shown in Figure 6-32.

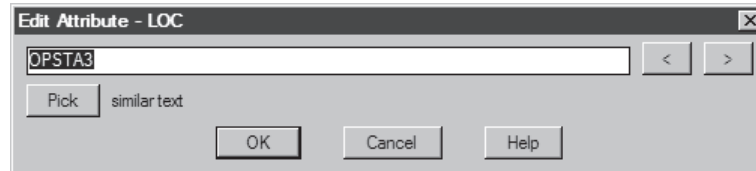


Figure 6-32 The **Edit Attribute** dialog box

Enter the new name for the attribute in the edit box. Alternatively, choose the **Pick** button from the **Edit Attribute** dialog box for selecting the attribute from the current drawing. You can increment or decrement the selected attribute value by choosing the **>** and **<** buttons. Next, choose the **OK** button; the name of the selected attribute will be changed to the specified name. Press ENTER to exit the command. You can also edit components using the **Edit** tool. To do so, choose the **Edit** button from the **Edit Components** panel of the **Schematic** tab; the **Insert / Edit Component** dialog box will be displayed. Specify the required description in the **Description** area of this dialog box and choose the **OK** button; the attribute of the component will be changed. Another method of editing the attribute of a component is by using the **Move/Show Attribute** button. To edit the attribute text of a component, choose the **Move/Show Attribute** button from the **Edit Components** panel of the **Schematic** tab; you will be prompted to select an attribute to move or pick on block graphics for list. Select the graphics of the block; the **SHOW / HIDE Attributes** dialog box will be displayed, as shown in Figure 6-33. Next, select the **Edit Attributes** check box located at the top right corner of this dialog box; the dialog box will be modified to the **EDIT Attributes** dialog box. Next, select the name of attribute from the list displayed in the **EDIT Attributes** dialog box; the **Edit Attribute** dialog box will be displayed. Specify the required options and choose the **OK** button in the **Edit Attribute** dialog box; the attribute value will be changed to the specified value.

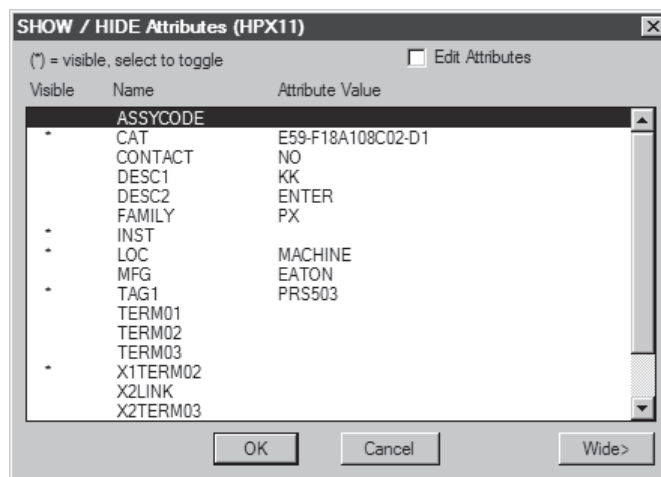


Figure 6-33 The **SHOW / HIDE Attributes** dialog box

Hiding Attributes

Ribbon:	Schematic > Edit Components > Move/Show Attribute > Hide Attribute (Single Pick)
Toolbar:	ACE:Main Electrical > Move/Show Attribute > Hide Attribute (Single Picks) or ACE>Edit Attributes > Hide Attribute (Single Picks)
Menu:	Components > Attributes > Hide/Unhide Attribute > Hide Attribute (Single Picks)
Command:	AEHIDEATT



The **Hide Attribute (Single Picks)** tool is used to hide the selected attribute. To do so, choose the **Hide Attribute (Single Pick)** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Hide/Unhide Attribute > Hide Attribute (Single Picks)** from the menu bar; you will be prompted to select the attribute to hide or pick on block graphics for list. Select the attribute; it will be hidden. Press ENTER to exit the command. Note that if you select the graphics of a block, the **SHOW / HIDE Attributes** dialog box will be displayed, refer to Figure 6-33. The attributes that have an asterisk (*) displayed in the **Visible** column of the **SHOW / HIDE Attributes** dialog box are visible in the drawing. To make these attributes invisible in the drawing, select these attributes in the dialog box; the asterisk will disappear from the **Visible** column indicating that the attribute is now invisible in the drawing.

You can hide multiple attributes at a time. To do so, enter 'W' at the Command prompt; you will be prompted to select the attributes. Select the attributes by using the crossing window and press ENTER; the attributes will hide. Press ENTER to exit the command. Also, using the **Hide Attribute (Window/Multiple)** button, you can hide the multiple attributes. To do so, choose the **Hide Attribute (Window/Multiple)** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab; you will be prompted to select the attributes. Select the attribute and press ENTER; the **Flip Attribute to Invisible** dialog box will be displayed. Select the required attributes to flip to invisible and choose the **OK** button; the attributes will hide. Note that you can select multiple attributes from the **Flip Attribute to Invisible** dialog box by using the CTRL or SHIFT keys.

Unhiding Attributes

Ribbon:	Schematic > Edit Components > Move/Show Attribute > Unhide Attribute (Window/Multiple)
Menu:	Components > Attributes > Hide/Unhide Attribute > Unhide Attribute (Window/Multiple)
Command:	AESHOWATTRIB

The **Unhide Attribute (Window/Multiple)** tool is used to unhide the hidden attributes. To do so, choose the **Unhide Attribute (Window/Multiple)** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Hide/Unhide Attribute > Unhide Attribute (Window/Multiple)** from the menu bar; you will be prompted to select objects. Select graphics of the symbol block and press ENTER; the **Flip Attributes to Visible** dialog box will be displayed. Select the attributes that you want to display and choose the **OK** button in the **Flip Attributes to Visible** dialog box;

the attributes will be visible on the screen. Note that multiple attributes can be selected from the **Flip Attributes to Visible** dialog box by pressing CTRL or SHIFT.

If you enter 'W' at the Command prompt, you will be prompted to specify the first corner. Specify the first corner; you will be prompted to specify the opposite corner. Specify the opposite corner and then press ENTER; the **Flip Attributes to Visible** dialog box will be displayed. Select one or more attributes and choose the **OK** button; the attributes will be flipped to visible.

Adding Attributes

Ribbon:	Schematic > Edit Components > Move/Show Attribute > Add Attribute
Toolbar:	ACE:Main Electrical > Move/Show Attribute > Add Attribute or ACE>Edit Attributes > Add Attribute
Menu:	Components > Attributes > Add Attribute
Command:	AEATTRIBUTE



The **Add Attribute** tool is used to add a new attribute to the existing AutoCAD Electrical block. To do so, choose the **Add Attribute** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Add Attribute** from the menu bar; you will be prompted to select the block. Select the symbol block; the **Add Attribute** dialog box will be displayed, as shown in Figure 6-34. The options in this dialog box are discussed next.

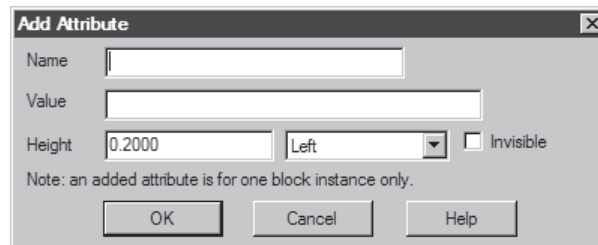


Figure 6-34 The Add Attribute dialog box

The **Name** edit box is used for identifying an attribute tag. Specify the name of the attribute in this edit box.

Specify a value for the attribute text in the **Value** edit box. The specified value will be displayed both in the drawing and reports.

Specify the height for the attribute value in the **Height** edit box. By default, 0.2000 is displayed in the **Height** edit box.

Select the justification for the attribute value from the drop-down list adjacent to the **Height** edit box. By default, Left is selected in the drop-down list.

Select the **Invisible** check box to make the attribute text invisible in the drawing. By default, this check box is cleared.

Specify the required options in the **Add Attribute** dialog box and choose the **OK** button in this dialog box; you will be prompted to specify the location for attribute. By default, the first location point is already selected by AutoCAD Electrical and it is 0, 0. Next, specify the second location point and click on the screen; the attribute value will be inserted into the drawing.

Squeezing an Attribute/Text

Ribbon:	Schematic > Edit Components > Move/Show Attribute > Squeeze Attribute/Text
Toolbar:	ACE:Main Electrical > Move/Show Attribute > Squeeze Attribute/Text or ACE>Edit Attributes > Squeeze Attribute/Text
Menu:	Components > Attributes > Squeeze Attribute/Text
Command:	AEATTSQUEEZE



The **Squeeze Attribute/Text** tool is used to reduce the attribute or text size to make it suitable for tight places. After each click on the attribute or text, the width of the attribute or text reduces by 5%. To squeeze the attribute or text size, choose the **Squeeze Attribute/Text** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Squeeze Attribute/Text** from the menu bar; you will be prompted to select the attribute or text to be squeezed. Select the attribute or text; you will be prompted again to select the attribute or text. This selection will continue till it gets reduced to minimum width. Next, press ENTER to exit the command.

Stretching an Attribute/Text

Ribbon:	Schematic > Edit Components > Move/Show Attribute > Stretch Attribute/Text
Toolbar:	ACE:Main Electrical > Move/Show Attribute > Stretch Attribute/Text or ACE>Edit Attributes > Stretch Attribute/Text
Menu:	Components > Attributes > Stretch Attribute/Text
Command:	AEATTSTRETCH



The **Stretch Attribute/Text** tool is used to expand the attribute or text size. After each click on the attribute or text, the width of the attribute or text increases by 5%. To stretch an attribute, choose the **Stretch Attribute/Text** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Stretch Attribute/Text** from the menu bar; you will be prompted to select the attribute or text that you want to stretch. Select the attribute or text; the attribute or text will be stretched. The command will continue till you press ENTER.

Changing the Attribute Size

Ribbon:	Schematic > Edit Components > Move/Show Attribute > Change Attribute Size
Toolbar:	ACE:Main Electrical > Move/Show Attribute > Change Attribute Size or ACE>Edit Attributes > Change Attribute Size
Menu:	Components > Attributes > Change Attribute Size
Command:	AEATTSIZE



The **Change Attribute Size** tool is used to change the height and width of the attribute text that has already been inserted into the drawing. To change the text size of the attribute, choose the **Change Attribute Size** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Change Attribute Size** from the menu bar; the **Change Attribute Size** dialog box will be displayed, as shown in Figure 6-35. The options in this dialog box are discussed next.

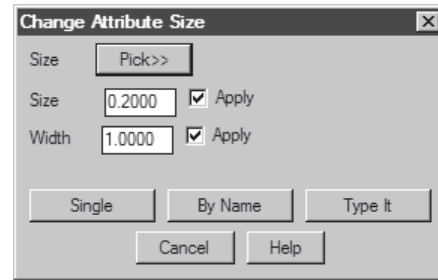


Figure 6-35 The **Change Attribute Size** dialog box

Specify the size for the attribute in the **Size** edit box. Specify the width for the attribute in the **Width** edit box. Alternatively, choose the **Pick >>** button; you will be prompted to select the attribute. Select the required attribute; the **Change Attribute Size** dialog box will appear again on the screen and the values of the attribute size and width will be displayed in the **Size** and **Width** edit boxes.

By default, the **Apply** check boxes are selected. As a result, the new size and width is applied to the attribute that you select. If you clear the **Apply** check boxes in the **Change Attribute Size** dialog box, the **Single**, **By Name**, and **Type It** buttons will not be available.

The **Single** button is used to select one attribute at a time.

The **By Name** button is used to change the size and width of the same type of attributes. For example, if you select DESC2 attribute of a symbol, then the size and width of only DESC2 attribute of all the components present in the drawing will get changed. Choose the **By Name** button; you will be prompted to select the attribute/name. Select the attribute/name and press ENTER; the size and width of the attributes of same type will be changed according to the specified size and width. You can also select multiple attributes at a time. To do so, enter **W** at the Command prompt; you will be prompted to specify the first corner. Specify the first corner; you will be prompted to specify the opposite corner. Specify the opposite corner and then press ENTER; the size and width of the selected attributes will get changed according to the specified size. Also, if you enter ALL at the Command prompt, all attributes by the same name that are present in the drawing will be selected and changed to the new attribute size. Press ESC to exit the command.

The **Type It** button is used to specify the name of the attribute to be matched to the selected attributes. To do so, choose the **Type It** button; the **Enter Attribute Name** dialog box will be displayed. Enter the attribute name in the edit box that you want to match to the attributes that you will select. For example, DESC1 for description of the component, TAG1 for tag of the component, and so on. Next, choose the **OK** button in the **Enter Attribute Name** dialog box; you will be prompted to select blocks to process. Select the blocks and press ENTER; the size of attributes will be changed accordingly. Alternatively, select multiple attributes by the window selection. Also, if you enter ALL at the Command prompt, all attributes present in the active drawing will be selected. Next, press ENTER to change the attributes to the new

attribute size and width. Note that you can also enter wildcards in the edit box and include a series of attribute names to match by separating each attribute name using semi-colons.

If you do not enter any attribute name in the edit box of the **Enter Attribute Name** dialog box and choose **OK**, then it will again return to the **Change Attribute Size** dialog box. Choose the **Cancel** button to exit from this dialog box.

Rotating an Attribute

Ribbon:	Schematic > Edit Components > Move/Show Attribute > Rotate Attribute
Toolbar:	ACE:Main Electrical > Move/Show Attribute > Rotate Attribute or ACE>Edit Attributes > Rotate Attribute
Menu:	Components > Attributes > Rotate Attribute
Command:	AEATTROTATE



The **Rotate Attribute** tool is used to rotate the selected attribute text by 90 degrees. To do so, choose the **Rotate Attribute** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Rotate Attribute** from the menu bar; you will be prompted to select the attribute text to rotate. Select the attribute text; the text will be rotated by 90 degrees in counterclockwise direction. Press ENTER to exit the command. Now if you want to move the attribute text, enter 'M' at the Command prompt and press ENTER; you will be prompted to specify the base point. Specify the base point and press ENTER; you will be prompted to specify the destination point. Specify the destination point and press ENTER; the attribute text will be moved to the specified location.

Changing the Justification of an Attribute

Ribbon:	Schematic > Edit Components > Move/Show Attribute > Change Attribute Justification
Toolbar:	ACE:Main Electrical > Move/Show Attribute > Change Attribute Justification or ACE>Edit Attributes > Change Attribute Justification
Menu:	Components > Attributes > Change Attribute Justification
Command:	AEATTJUSTIFY



The **Change Attribute Justification** tool is used to change the justification of any attribute such as wire number text, component description text, and so on. To change the justification of an attribute, choose the **Change Attribute Justification** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Change Attribute Justification** from the menu bar; the **Change Attribute/Text Justification** dialog box will be displayed, as shown in Figure 6-36. The options in this dialog box are discussed next.

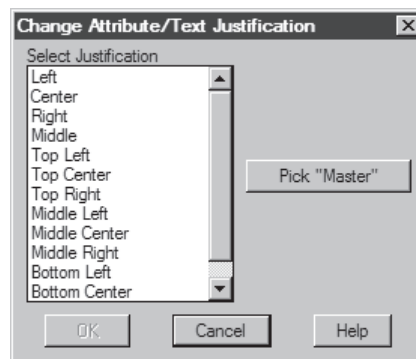


Figure 6-36 The **Change Attribute/Text Justification** dialog box

Select the required justification for the attribute text from the **Select Justification** area in this dialog box; the **OK** button will be available. Choose **OK**; you will be prompted to select the text or attribute to change the justification. Select the text or attribute; the justification of the attribute or text will be changed.

Alternatively, choose the **Pick Master** button; you will be prompted to select the master attribute or text that will be used for justification of other attributes or text. Next, select the attribute or text; the **Change Attribute/Text Justification** dialog box will be displayed on the screen again. Choose **OK**; you will be prompted to select the attribute or text for the change of justification. Select attributes or text and press ENTER; the justification of attributes or text will be changed. Alternatively, enter 'W' at the Command prompt to select the attributes or text by crossing window. Press ENTER to exit the command.

Changing an Attribute Layer

Ribbon:	Schematic > Edit Components > Move/Show Attribute > Change Attribute Layer
Toolbar:	ACE:Main Electrical > Move/Show Attribute > Change Attribute Layer or ACE>Edit Attributes > Change Attribute Layer
Menu:	Components > Attributes > Change Attribute Layer
Command:	AEATTLAYER



The **Change Attribute Layer** tool is used to change the layer of the selected attribute. To do so, choose the **Change Attribute Layer** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Attributes > Change Attribute Layer** from the menu bar; the **Force Attribute/Text to a Different Layer** dialog box will be displayed, as shown in Figure 6-37. Different options in this dialog box are discussed next.

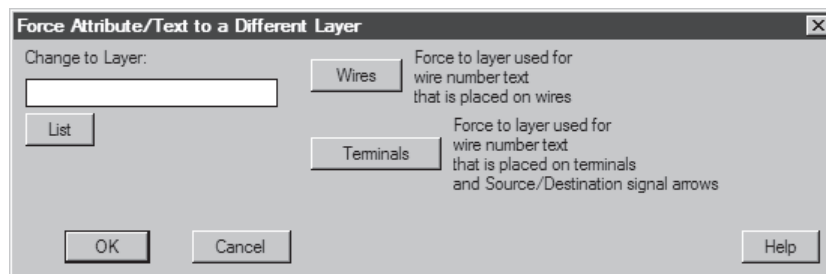


Figure 6-37 The **Force Attribute/Text to a Different Layer** dialog box

Enter the name of the target layer in the **Change to Layer** edit box. Alternatively, choose the **List** button; the **Layers in Drawing** dialog box will be displayed. Select the layer and choose the **OK** button; the name of the target layer will be displayed in the **Change to Layer** edit box.

If you choose the **Wires** button, the WIRENO will be displayed in the **Change to Layer** edit box. This button is used to change the layer of a selected attribute or text for the layer that has been used for wire number text placed on wires.

If you choose the **Terminal** button, the WIREREF will be displayed in the **Change to Layer** edit box. In this case, the layer of the attribute or text that you select will get changed to the layer that is used for wire number text placed on terminals and source/destination signal arrow.

After specifying the required options in the **Force Attribute/Text to a Different Layer** dialog box, choose the **OK** button in this dialog box; you will be prompted to select the attribute or text to move to layer. Select the attribute or text; the attribute or text layer will get changed to the layer that you have entered in the **Change to Layer** edit box. Press ENTER to exit the command.

TUTORIALS

Tutorial 1

In this tutorial, you will insert a ladder and components in a drawing and then use the **Copy Component**, **Scoot**, **Move Component**, and **Delete Component** tools for copying, scooting, moving, and deleting the inserted components respectively, as shown in Figure 6-38. Also, you will hide the attributes of components. **(Expected time: 25 min)**

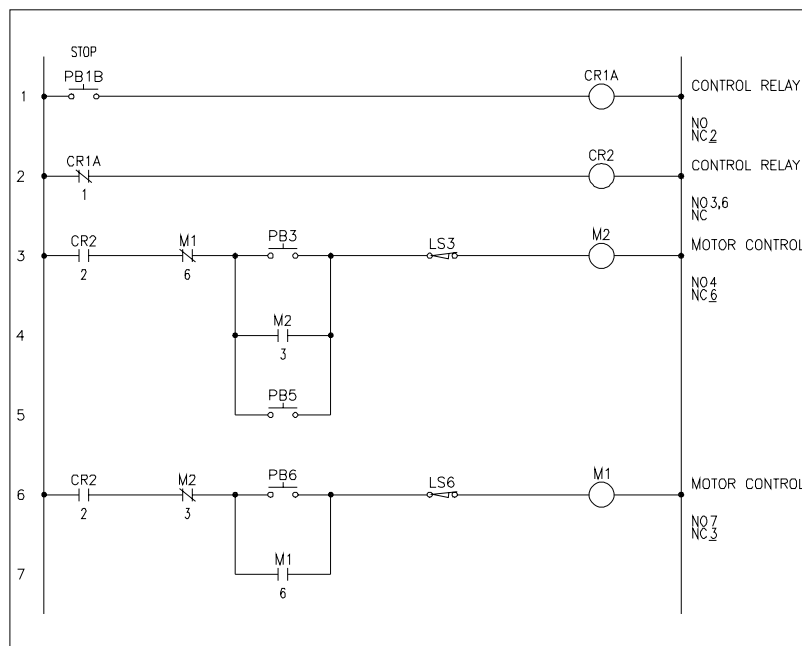



Figure 6-38 Ladder diagram for Tutorial 1

The following steps are required to complete this tutorial:


- Create a new drawing in the **CADCIM** project.
- Insert ladder in the drawing.
- Insert components and add descriptions to the component.

- d. Copy components.
- e. Hide attributes.
- f. Delete components.
- g. Move components.
- h. Add a wire between two rungs.
- i. Scoot the components.
- j. Trim rung.
- k. Align components.
- l. Save and close the drawing file.

Creating a New Drawing

1. If the **CADCIM** project is inactive, right-click on it; a shortcut menu is displayed. Choose the **Activate** option from the shortcut menu; the **CADCIM** project becomes the active project and is displayed at the top of the **Projects** rollout. 
2. Choose the **New Drawing** button from the **Project Manager**; the **Create New Drawing** dialog box is displayed. Enter **C06_tut01** in the **Name** edit box of the **Drawing File** area.
3. Next, choose the **Browse** button located on the right of the **Template** edit box; the **Select template** dialog box is displayed. Select the **ACAD_ELECTRICAL** template from the **Select template** dialog box. Choose the **Open** button; the location and path of the template file is displayed in the **Template** edit box.
4. Enter **Schematic Components** in the **Description 1** edit box.
5. Choose the **OK** button; the **C06_tut01.dwg** drawing is created in the **CADCIM** project and is displayed at the bottom of the drawing list in the **CADCIM** project.

Inserting the Ladder into the Drawing

1. Choose the **Insert Ladder** button from the **Insert Wires/Wire Numbers** panel of the **Schematic** tab or choose **Wires > Ladders > Insert Ladder** from the menu bar; the **Insert Ladder** dialog box is displayed. 
2. Set the following parameters in the **Insert Ladder** dialog box:

Width: 8.000	Spacing: 1.000
1st Reference: 1	Rungs: 7
1 Phase : Select this radio button	Yes : Select this radio button
3. On setting these parameters, click in the **Length** edit box; the length of the ladder is automatically calculated and displayed in this edit box.
4. Enter **0** in the **Skip** edit box of the **Draw Rungs** area.
5. Choose the **OK** button; you are prompted to specify the start position of the first rung. Enter **11,18** at the Command prompt and press ENTER; the ladder is inserted into the drawing, as shown in Figure 6-39.

6. Choose **View > Zoom > In** from the menu bar to zoom the drawing.

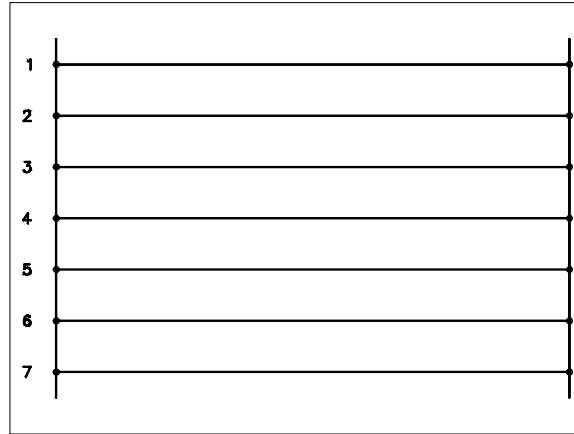



Figure 6-39 Ladder with 7 rungs

Inserting Components and Adding a Description

1. Choose the **Icon Menu** button from the **Insert Components** panel of the **Schematic** tab or choose **Components > Insert Component** from the menu bar; the **Insert Component** dialog box is displayed. 
2. Select the **Push Buttons** icon from the **JIC: Schematic Symbols** area of the **Insert Component** dialog box; the **JIC: Schematic Symbols** area gets changed to the **JIC: Push Buttons** area.
3. Next, select the **Push Button NO** icon located at the 1st row, 1st column in the **JIC: Push Buttons** area; the cursor along with the component is displayed on the screen and you are prompted to specify the insertion point for the push button. Enter **11.5,18** at the Command prompt and press ENTER; the push button is placed at rung 1 and the **Insert / Edit Component** dialog box is displayed.

By default, PB1B is displayed in the edit box of the **Component Tag** area.

4. Enter **STOP** in **Line 1** edit box in the **Description** area.
5. Choose the **OK** button from the **Insert / Edit Component** dialog box; the PB1B component is inserted into the drawing.
6. Repeat step 1. Select the **Relays/Contacts** icon displayed in the **JIC: Schematic Symbols** area of the **Insert Component** dialog box; the **JIC: Relays and Contacts** area is displayed.
7. Select the **Relay Coil** icon located at the 1st row and 1st column in the **JIC: Relays and Contacts** area of the **Insert Component** dialog box; you are prompted to specify the insertion point. Enter **13,18** at the Command prompt and press ENTER; the **Insert / Edit Component** dialog box is displayed.

By default, CR1A is displayed in the edit box of the **Component Tag** area.

8. Enter **CONTROL RELAY** in **Line 1** edit box in the **Description** area.
9. Choose the **OK** button from the **Insert / Edit Component** dialog box; the component CR1A gets added to rung 1 of the ladder.
10. Repeat step 1. Select the **Relays/Contacts** icon displayed on the right of the **Insert Component** dialog box; the **JIC: Relays and Contacts** area is displayed.
11. Select the **Relay Coil** icon located at the 1st row and 1st column in the **JIC: Relays and Contacts** area of the **Insert Component** dialog box; you are prompted to specify the insertion point. Enter **13.5,17** at the Command prompt and press ENTER. By default, CR2 is displayed in the edit box of the **Component Tag** area of the **Insert / Edit Component** dialog box.
12. Enter **CONTROL RELAY** in **Line 1** edit box in the **Description** area.
13. Choose the **OK** button from the **Insert / Edit Component** dialog box; the component CR2 gets added to rung 2 of the ladder.
14. Repeat Step 1. Select **Relay NC Contact** located at the 1st row, 3rd column in the **JIC: Relays and Contacts** area of the **Insert Component** dialog box; you are prompted to specify the insertion point. Enter **11.5,17** at the Command prompt and press ENTER; the **Insert / Edit Child Component** dialog box is displayed.
15. Choose the **Drawing** button from the **Component Tag** area of the **Insert / Edit Child Component** dialog box; the **Active Drawing list for FAMILY = "CR"** dialog box is displayed. Select **CR1A** from this dialog box and choose the **OK** button; the information is displayed in the **Insert / Edit Child Component** dialog box.

By default, CONTROL RELAY is displayed in the **Line 1** edit box of the **Description** area.

16. Enter **NORMALLY CLOSED** in the **Line 2** edit box of the **Description** area.
17. Choose the **OK** button in the **Insert / Child Edit Component** dialog box; the component CR1A is added to the ladder rung 2.
18. Repeat step 1. Select the **Relays/Contacts** icon and then select **Relay NO Contact** from the 1st row, 2nd column in the **JIC: Relays and Contacts** area; you are prompted to specify the insertion point. Enter **11.5,16** at the Command prompt and press ENTER; the **Insert / Edit Child Component** dialog is displayed. Choose the **Drawing** button from the **Component Tag** area of the **Insert / Edit Child Component** dialog box; the **Active Drawing list for FAMILY = "CR"** dialog box is displayed. Select CR2 from this dialog box and choose the **OK** button; the information is displayed in the **Insert / Edit Child Component** dialog box. Enter **NORMALLY OPEN** in the **Line 2** edit box of

the **Description** area in the **Insert / Edit Child Component** dialog box and then choose the **OK** button from this dialog box; the CR2 control relay is inserted into the drawing in the rung 3.

19. Repeat step 1. Select **Motor Control** icon from the **JIC: Schematic Symbols** area in the **Insert Component** dialog box. Next, select **Motor Starter Coil** (1st row, 5th column) from the **JIC: Motor Control** area; you are prompted to specify the insertion point for the component. Enter **17.5,16** at the Command prompt and press ENTER; the **Insert / Edit Component** dialog box is displayed. Enter M2 in the edit box of the **Component Tag** area. Also, enter its description as **MOTOR CONTROL** in the **Line 1** edit box of the **Description** area of the **Insert / Edit Component** dialog box. Choose the **OK** button from the **Insert / Edit Component** dialog box; the **Motor Starter Coil** is inserted into rung 3 of the ladder.
20. Similarly, insert **2nd + Starter Contact NO** in the rung 4. To do so, choose the **Motor Control** icon from the **Insert Component** dialog box and then select **2nd+Starter Contact NO**(2nd row, 3rd column) from the **JIC: Motor Control** area; you are prompted to specify the insertion point. Enter **11.5,15** at the Command prompt and press ENTER; the **Insert / Edit Child Component** dialog box is displayed. Choose the **Drawing** button from the **Component Tag** area of the **Insert / Edit Child Component** dialog box; the **Active Drawing list for FAMILY = "M"** dialog box is displayed. Select M2 from this dialog box and choose the **OK** button; the information is displayed in the **Insert / Edit Child Component** dialog box. Enter **NORMALLY OPEN** in the **Line 2** edit box of the **Description** area in the **Insert / Edit Child Component** dialog box and then choose the **OK** button from this dialog box; the M2 starter contact is inserted into the drawing in the rung 4.
21. Repeat step 1. Select **Motor Control** icon from the **JIC: Schematic Symbols** area in the **Insert Component** dialog box. Next, select **Motor Starter Coil** (1st row, 5th column) from the **JIC: Motor Control** area; you are prompted to specify the insertion point for the component. Enter **18,13** at the Command prompt. Enter M1 in the edit box of the **Component Tag** area. Also, enter its description as **MOTOR CONTROL** in the **Line 1** edit box of the **Description** area of the **Insert / Edit Component** dialog box. Choose the **OK** button from the **Insert / Edit Component** dialog box; the **Motor Starter Coil (M1)** is inserted into rung 6 of the ladder.
22. Again, repeat step 1. Select the **Motor Control** icon and then select **2nd+Starter Contact NC** from the 3rd row, 1st column in the **JIC: Motor Control** area of the **Insert Component** dialog box; you are prompted to specify the insertion point for the component. Enter **12.8,16** at the Command prompt and press ENTER; the **Insert / Edit Child Component** dialog box is displayed. Choose the **Drawing** button from the **Component Tag** area of the **Insert / Edit Child Component** dialog box; the **Active Drawing list for FAMILY = "M"** dialog box is displayed. Select M1 from this dialog box and choose the **OK** button; the information is displayed in the **Insert / Edit Child Component** dialog box. Enter **NORMALLY CLOSED** in the **Line 2** edit box of the **Description** area in the **Insert / Edit Child Component** dialog box and then choose the **OK** button from this dialog box; the Motor starter contact (M1) is inserted into the drawing in the rung 3.

23. Insert **Limit Switch, NC** in the rung 3. To do so, choose **Limit Switches** icon from the **Insert Component** dialog box; the **JIC: Limit Switches** area is displayed. Select **Limit Switch, NC** from this area; you are prompted to specify the insertion point. Enter **16,16** at the Command prompt and press ENTER; the **Insert / Edit Component** dialog box is displayed. By default, LS3 is displayed in the **Component Tag** edit box of the **Insert / Edit Component** dialog box. Also, enter **LIMIT SWITCH** and **NORMALLY CLOSED** in the **Line 1** and **Line 2** edit boxes of the **Description** area of the **Insert / Edit Component** dialog box. Choose the **OK** button; LS3 gets inserted into rung 3.
24. Similarly, insert **Limit Switch, NO** in the rung 1. To do so, choose the **Limit Switch, NO** from the **JIC: Limit Switches** icon of the **Insert Component** dialog box; you are prompted to specify the insertion point. Enter **18,18** at the Command prompt and press ENTER; the **Insert / Edit Component** dialog box is displayed. By default, LS1A is displayed in the edit box of the **Component Tag** area in this dialog box. Next, choose the **OK** button; LS1A is inserted into the drawing. Figure 6-40 shows the ladder with components inserted in it.

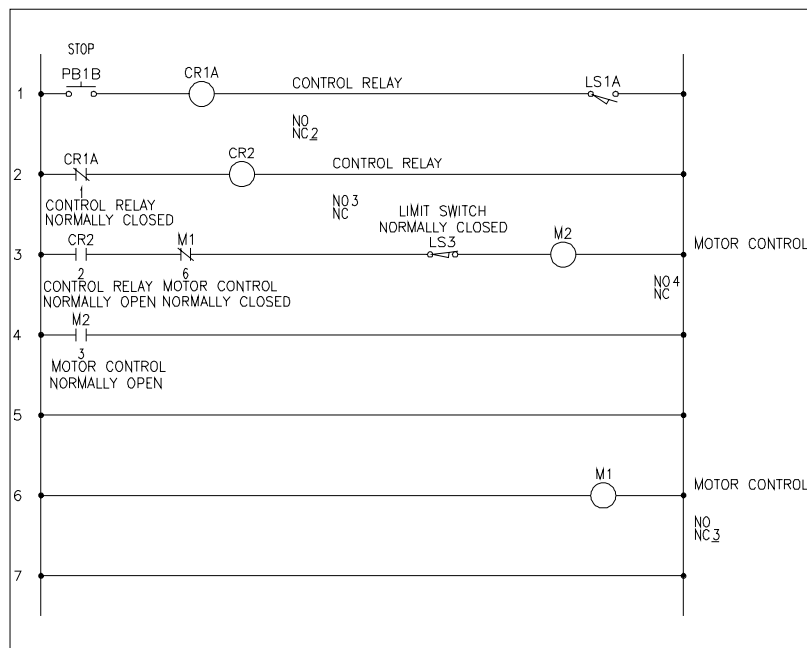


Figure 6-40 Components inserted in the ladder

Copying Components

1. Choose the **Copy Component** button from the **Edit Components** panel of the **Schematic** tab or choose **Components > Copy Component** from the menu bar; you are prompted to select a component to copy.
2. Select PB1B (push button) placed on rung 1; you are prompted to specify the insertion point for the copied component. Enter **14,16** at the Command prompt and press ENTER; the **Insert / Edit Component** dialog box is displayed.



3. By default, PB3 is displayed in the edit box of the **Component Tag** area. Enter **DOWN** in the **Line 1** edit box of the **Description** area of the **Insert / Edit Component** dialog box.
4. Choose the **OK** button from this dialog box; the component PB3 gets inserted into rung 3.
5. Similarly, right-click on CR2 placed on rung 3 and choose the **Copy Component** option from the shortcut menu; you are prompted to specify the insertion point. Enter **12,13** at the Command prompt and press ENTER; the **Insert / Edit Child Component** dialog box is displayed.
6. Choose the **Drawing** button from the **Component Tag** area of the **Insert / Edit Child Component** dialog box; the **Active Drawing list for FAMILY = "CR"** dialog box is displayed. Select **CR2** (third entry) from this dialog box and choose the **OK** button; the information is displayed in the **Insert / Edit Child Component** dialog box.

By default, CONTROL RELAY is displayed in the **Line 1** edit box of the **Description** area.

7. Enter **NORMALLY OPEN** in the **Line 2** edit box of the **Description** area.
8. Choose the **OK** button in the **Insert / Child Edit Component** dialog box; the component CR2 is inserted into rung 6 of the drawing.
9. Next, right-click on PB1B that you have placed on rung 1 and choose the **Copy Component** option from the shortcut menu; you are prompted to specify the insertion point. Enter **13,14** at the Command prompt and press ENTER; the **Insert / Edit Component** dialog box is displayed. Remove the description from the **Line 1** of the **Description** area in the **Insert / Edit Component** dialog box. Also, enter **PB5** in the edit box of the **Component Tag** area, if it is not displayed. Choose the **OK** button in this dialog box; the PB5 is inserted on rung 5. Again, copy the PB1B to rung 6. To do so, right-click on it and choose the **Copy Component** option from the shortcut menu; you are prompted to specify the insertion point. Enter **14,13** at the Command prompt and press ENTER; the **Insert / Edit Component** dialog box is displayed. In this dialog box, PB6 is displayed by default in the edit box of the **Component Tag** area. Remove the description from the **Line 1** edit box of the **Description** area and choose the **OK** button; PB6 is inserted into the rung 6 in the ladder.
10. Next, right-click on motor control M2 that is on rung 4 and choose the **Copy Component** option from the shortcut menu; you are prompted to specify the insertion point. Enter **12,12** at the Command prompt and press ENTER; the **Insert / Edit Child Component** dialog box is displayed. Choose the **Drawing** button from the **Component Tag** area of the **Insert / Edit Child Component** dialog box; the **Active Drawing list for FAMILY = "M"** dialog box is displayed. Select **M1** (first entry) from this dialog box and choose the **OK** button; the information is displayed in the **Insert / Edit Child Component** dialog box. Enter **NORMALLY OPEN** in the **Line 2** edit box of the **Description** area in the **Insert / Edit Child Component** dialog box and then choose the **OK** button from this dialog box; the Motor starter contact (M1) is inserted into the drawing in the rung 7.

11. Similarly, select M1 from rung 3 and copy it to rung 6 at **15,13**. Choose the **Drawing** button from the **Component Tag** area of the **Insert / Edit Child Component** dialog box; the **Active Drawing list for FAMILY = "M"** dialog box is displayed. Select M2 (fourth entry) from this dialog box and choose the **OK** button; the information is displayed in the **Insert / Edit Child Component** dialog box. Enter **NORMALLY CLOSED** in the **Line 2** edit box of the **Description** area in the **Insert / Edit Child Component** dialog box and then choose the **OK** button from this dialog box; the M2 starter contact is inserted into the drawing in the rung 6.
12. Next, select limit switch LS3 from 3rd rung and copy it at **15,15**. Again, select LS3 and copy it at **16,13**. You will notice that the limit switch is inserted in rung 4 and rung 6. Keep the values in the **Description** area intact. Figure 6-41 shows the copied components.

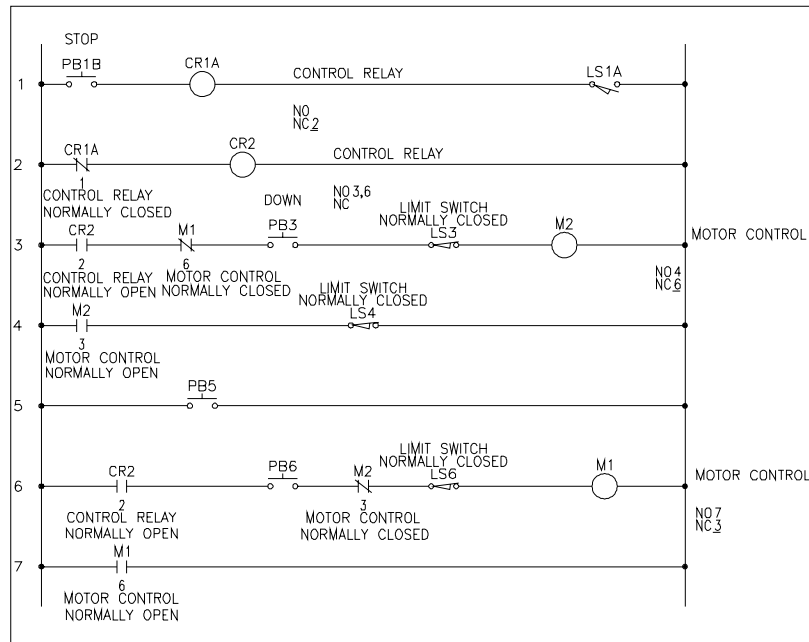


Figure 6-41 The copied components


Hiding Attributes

1. Choose the **Hide Attribute (Single Pick)** button from the **Move/Show Attribute** flyout of the **Edit Components** panel of the **Schematic** tab or **Components > Attributes > Hide/Unhide Attribute > Hide Attribute (Single Pick)** from the menu bar; you are prompted to select the attributes to hide.
2. Select the DESC1 and DESC2 attributes of following components:
 - CR1A placed on rung 2,
 - CR2, M1, PB3, and LS3 placed on rung 3,
 - M2 and LS4 placed on rung 4,
 - CR2, M2, and LS6 placed on rung 6, and
 - M1 placed on rung 7. On doing so, the description of the components is hidden.



3. Press ENTER to exit the command. Figure 6-42 shows the ladder and component with hidden attributes.

Deleting a Component

1. Choose the **Delete Component** button from the **Edit Components** panel of the **Schematic** tab or choose **Components > Delete Component** from the menu bar; you are prompted to select objects. 
2. Select LS1A and LS4, and then press ENTER; the **Search for / Surf to Children?** dialog box is displayed. Choose the **No** button from this dialog box; components are deleted from the drawing, as shown in Figure 6-43.

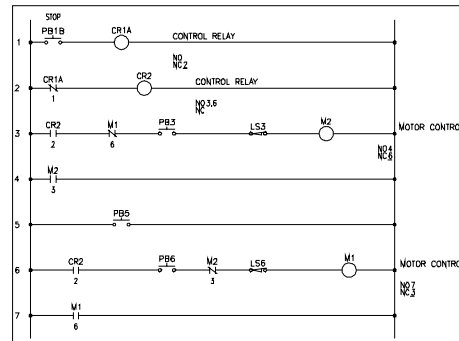
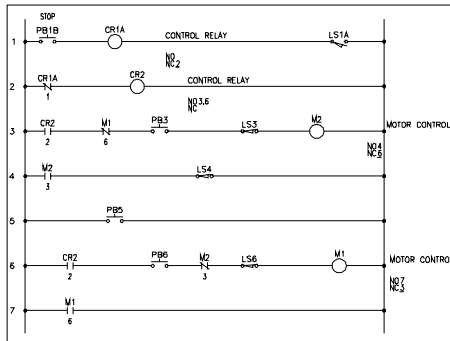




Figure 6-42 The ladder after hiding attributes Figure 6-43 The ladder after deleting components

Moving Components

1. Choose the **Move Component** button from the **Scoot** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Move Component** from the menu bar; you are prompted to select the component to move. 
2. Select M2 from rung 6; you are prompted to specify the insertion point for the component.
3. Enter **13,13** at the Command prompt and press ENTER; M2 is moved to a new location in rung 6, as shown in Figure 6-44.

Adding Wires

1. Choose the **Wire** button from the **Insert Wires/Wire Numbers** panel of the **Schematic** tab or choose **Wires > Insert Wire** from the menu bar; you are prompted to specify the starting point of the wire. 
2. Enter **13.4,16** and press ENTER; you are prompted to specify the wire endpoint. Then, enter **13.4,15** and press ENTER; the wire is inserted between rung 3 and 4 on the left side of the PB3.

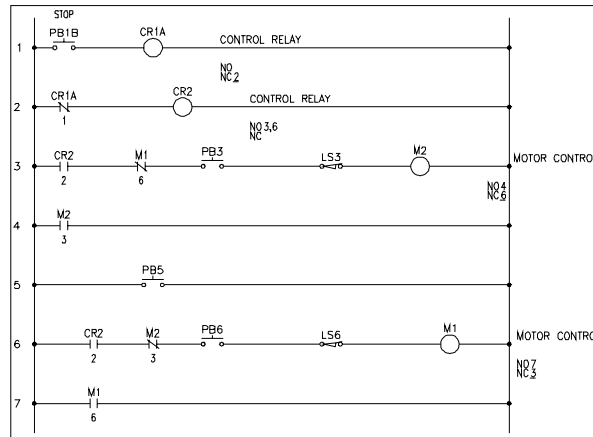



Figure 6-44 M2 moved to a different location

3. Enter **14.6,16** and press ENTER; you are prompted to specify the wire endpoint. Then, enter **14.6,15** and press ENTER; the wire is inserted between rung 3 and 4 on the right side of the PB3.
4. Enter **13.4,15** and press ENTER; you are prompted to specify the wire endpoint. Then, enter **13.4,14** and press ENTER; the wire is inserted between rung 4 and 5.
5. Enter **14.6,15** and press ENTER; you are prompted to specify the wire endpoint. Then, enter **14.6,14** and press ENTER; the wire is inserted between rung 4 and 5.
6. Enter **13.4,13** and press ENTER; you are prompted to specify the wire endpoint. Then, enter **13.4,12** and press ENTER; the wire is inserted between rung 6 and 7 on the left side of the PB6.
7. Next, enter **14.6,13** and press ENTER; you are prompted to specify the wire endpoint. Then enter **14.6,12** and press ENTER; the wire is inserted between rung 6 and 7 on the right side of the PB6.
8. Press ENTER to exit the command. Figure 6-45 shows the wires inserted in the drawing.

Scooting Components

1. Choose the **Scoot** button from the **Edit Components** panel of the **Schematic** tab or choose **Components > Scoot** from the menu bar; you are prompted to select the component. 
2. Select M2 from rung 4 and enter **14,15** at the Command prompt and then press ENTER; M2 is moved to a new location.
3. Select PB5 and enter **14,14** at the Command prompt and then press ENTER; PB5 is moved to a new location.

4. Select M1 from rung 7 and enter **14,12** at the Command prompt and then press ENTER; M1 is moved to a new location.
5. Press ENTER to exit the command. Figure 6-46 shows the schematic diagram after using the **Scoot** tool.

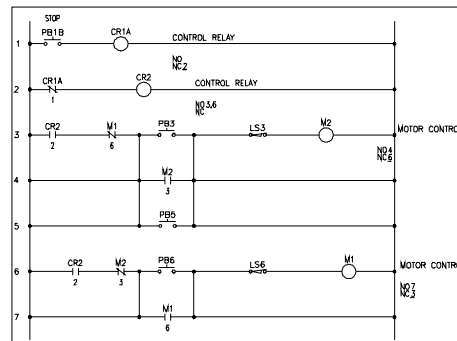
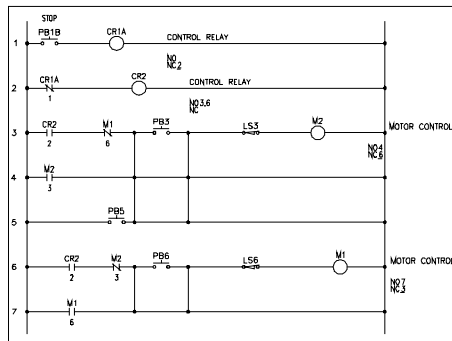




Figure 6-45 Wires inserted in the ladder Figure 6-46 The ladder after using the **Scoot** tool

Trimming the Rung

1. Choose the **Trim Wire** button from the **Edit Wires/Wire Numbers** panel of the **Schematic** tab or choose **Wires > Trim Wire** from the menu bar; you are prompted to select the wire to trim. 
2. Select the following wires to trim: right and left portions of rung 4, rung 5, and rung 7.
3. Next, press ENTER to exit the command.

Aligning Components

1. Choose the **Align** button from the **Scoot** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Align** option from the menu bar; you are prompted to select the component to align horizontally or vertically. 
2. Select PB1B on rung 1; an imaginary line passing through the center of PB1 is displayed and you are prompted to select objects.
3. Select CR1A placed on rung 2, CR2 placed on rung 3, and CR2 placed on rung 6 from the left of the ladder.
4. Press ENTER; the components are aligned toward the left.
5. Repeat step 1. Select M1 from rung 3; an imaginary line passing through the center of M1 is displayed and you are prompted to select objects. Next, select M2 placed on rung 6 and press ENTER; M1 placed on rung 3 and M2 placed on rung 6 are aligned.

6. Again, repeat step 1 and then select M1 placed on rung 6. Next, select M2 placed on rung 3, CR2 placed on rung 2, and CR1A placed on rung 1 and press ENTER; the selected components are aligned.
7. Repeat step 1 and then select PB3; you are prompted to select objects. Next, select M2 placed on rung 4, PB5, PB6, and M1 placed on rung 7 from the drawing and then press ENTER; the selected components are aligned. Figure 6-47 shows the aligned components.

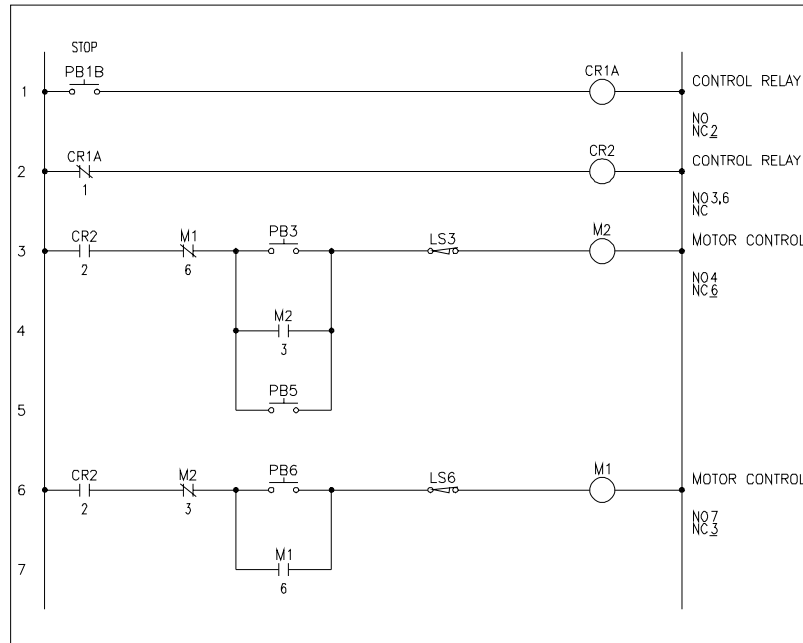


Figure 6-47 The trimmed wires and the aligned components

Saving and Closing the Drawing File

1. Choose **Save** from the **Application Menu** to save the drawing file *C06_tut01.dwg*.
2. Choose **Close > Current Drawing** from the **Application Menu** to close the drawing file.

Tutorial 2

In this tutorial, you will correct errors in *C06_tut01.dwg* drawing file of the **CADCIM** project by using the **Electrical Audit** and the **Drawing Audit** tools. Also, you will add catalog data to components.
(Expected time: 20 min)


The following steps are required to complete this tutorial:

- a. Open *C06_tut01.dwg* of Tutorial 1 from the **CADCIM** project.
- b. Audit the drawing using the **Drawing Audit** tool.
- c. Audit the drawing using the **Electrical Audit** tool.
- d. Save and close the drawing file.

Opening the Drawing File

1. Right-click on *C06_tut01.dwg* drawing file of the **CADCIM** project in the **Projects** rollout of the **Project Manager**; a shortcut menu is displayed. Choose the **Open** option from the shortcut menu; the drawing is opened.

Auditing the Drawing Using the DWG Audit Tool

1. Choose the **DWG Audit** button from the **Schematic** panel of the **Reports** tab or choose **Projects > Reports > Drawing Audit** from the menu bar; the **Drawing Audit** dialog box is displayed. 
2. Select the **Active drawing** radio button, if it is not selected by default.
3. Choose the **OK** button; the modified **Drawing Audit** dialog box is displayed.
4. Choose the **OK** button from the modified **Drawing Audit** dialog box; the **Drawing Audit** message box is displayed.
5. Choose the **OK** button in the **Drawing Audit** message box; the **Report: Audit for this drawing** dialog box is displayed.
6. Choose the **Close** button from the **Report: Audit for this drawing** dialog box.
7. The visual wire indicators in the diagram are shown in red color, as shown in Figure 6-48, which indicates the wires in the drawing.
8. Next, choose **View > Redraw** from the menu bar to eliminate the wire indicators.

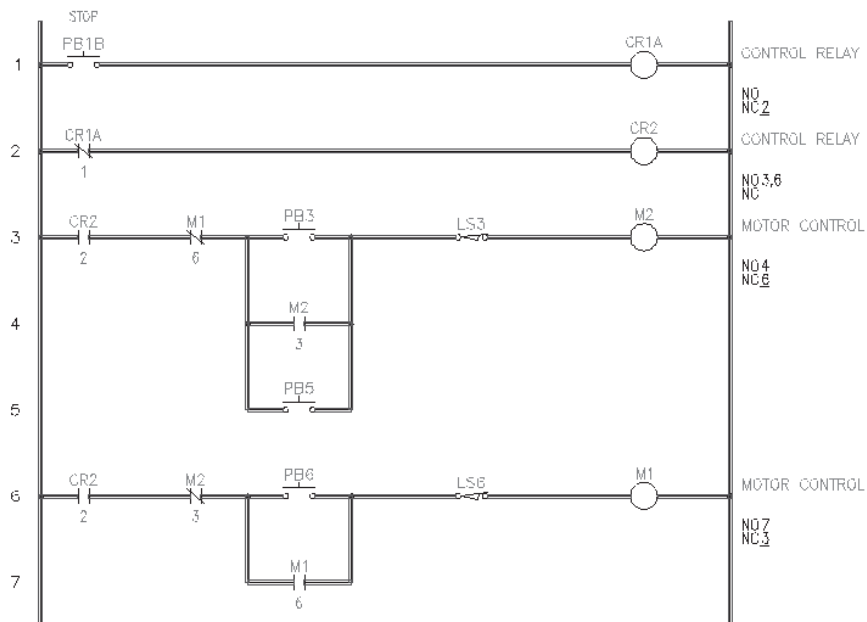



Figure 6-48 Visual wire indicators indicating wires in the drawing

Auditing the Drawing Using the Electrical Audit Tool

1. Choose the **Electrical Audit** button from the **Schematic** panel of the **Reports** tab or choose **Projects > Reports > Electrical Audit** from the menu bar; the **Electrical Audit** dialog box is displayed. 
2. Select the **Active Drawing** radio button from the **Electrical Audit** dialog box; the errors in the *C06_tut01.dwg* drawing file are displayed in the edit box located on the right of the **Active Drawing** radio button.
3. Choose the **Details** button from the **Electrical Audit** dialog box to expand it.
4. Choose the **Component - No Catalog Number** tab from the **Electrical Audit** dialog box. Next, select the PB1B push button from the **Tag Name** column of this tab.
5. Choose the **Go To** button; PB1 push button is zoomed on your screen.

**Note**

After choosing the **Go To** button, if the **QSAVE** dialog box is displayed, choose the **OK** button in this dialog box to save the changes.

6. Next, right-click on the PB1B component; a shortcut menu is displayed. Choose the **Edit Component** option from the shortcut menu; the **Insert / Edit Component** dialog box is displayed.
7. Choose the **Project** button from the **Catalog Data** area of the **Insert / Edit Component** dialog box; the **Find: Catalog Assignments** dialog box is displayed. Select the **Active Project** radio button, if it is not selected.
8. Choose the **OK** button from the **Find: Catalog Assignments** dialog box; the **HPB11 / VPB11 catalog values(this project)** dialog box is displayed.

**Note**

If the **QSAVE** dialog box is displayed, choose the **OK** button in this dialog box to save the changes.

9. Select the **800T-A2A** catalog number from the **Catalog Number** column and choose the **OK** button from the **HPB11 / VPB11 catalog values** dialog box; AB is displayed in the **Manufacturer** edit box and 800T-A2A is displayed in the **Catalog** edit box of the **Insert / Edit Component** dialog box.
10. Next, choose the **OK** button from the **Insert / Edit Component** dialog box to save the changes and exit this dialog box. Now you can notice 'x' on the left of the PB1B in the **Electrical Audit** dialog box.
11. Choose the **Close** button from the **Electrical Audit** dialog box. In this way, you can correct the errors in the drawing.

12. To check whether an error is rectified or not, choose the **Electrical Audit** button from the **Schematic** panel of the **Reports** tab; the **Electrical Audit** dialog box is displayed. Choose the **Active Drawing** button to display the errors in the active drawing.
13. Choose the **Details** button to expand this dialog box. Next, choose the **Component - No Catalog Number** tab from the **Electrical Audit** dialog box. Here you can check that the selected error is corrected and is not displayed in the **Electrical Audit** dialog box. In other words, PB1B is not displayed in the list. Choose the **Close** button to exit the **Electrical Audit** dialog box

Saving and Closing the Drawing File

1. Choose **Save** from the **Application Menu** to save the drawing file.
2. Choose **Close > Current Drawing** from the **Application Menu** to close the drawing file.


Tutorial 3

In this tutorial, you will create a schematic drawing. Then you will add catalog data to one of the components and then copy it to other components. Also, you will add location data to one of the control relay and copy it to the rest of the control relays. **(Expected time: 20 min)**

The following steps are required to complete this tutorial:

- a. Create a new drawing *C06_tut03.dwg* in the **CADCIM** project.
- b. Insert ladder in the drawing.
- c. Insert components and add descriptions to them.
- d. Copy the catalog data using the **Copy Catalog Assignment** tool.
- e. Enter the location data.
- f. Save and close the drawing file.

Creating a New Drawing

1. Choose the **New Drawing** button in the **Project Manager**; the **Create New Drawing** dialog box is displayed. Enter **C06_tut03** in the **Name** edit box of the **Drawing File** area. 
2. Next, Choose the **Browse** button located on the right of the **Template** edit box; the **Select template** dialog box is displayed. Select the **ACAD_ELECTRICAL** template and choose the **Open** button; the location and path of the template file is displayed in the **Template** edit box.
3. Enter **Schematic Components** in the **Description 1** edit box.
4. Choose the **OK** button in the **Create New Drawing** dialog box; the *C06_tut03.dwg* drawing is created and displayed at the bottom of the drawing list in the **CADCIM** project.

Inserting the Ladder

1. Choose the **Insert Ladder** button from the **Insert Wires/Wire Numbers** panel of the **Schematic** tab or choose **Wires > Ladders > Insert Ladder** from the menu bar; the **Insert Ladder** dialog box is displayed.



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

Width: **6.000**

Spacing: **1.000**

1st Reference: **101**

Rungs: **7**

1 Phase: Select this radio button

Yes: Select this radio button

Do not change values in the rest of the edit boxes.

On setting these parameters, click in the **Length** edit box; the length of the ladder is automatically calculated and displayed in this edit box.

3. Choose the **OK** button; you are prompted to specify the start position of first rung.
4. Enter **8,13** at the Command prompt and press ENTER; the ladder is inserted into the drawing.
5. To zoom the drawing, choose **View > Zoom > Extents** from the menu bar; the drawing gets zoomed.

Inserting Components in the Ladder


1. Choose the **Icon Menu** button from the **Insert Components** panel of the **Schematic** tab or **Components > Insert Component** from the menu bar; the **Insert Component** dialog box is displayed.
2. Choose components from the **Insert Component** dialog box and insert them on the ladder. Also, add description to components in the **Insert / Edit Component** or **Insert / Edit Child Component** dialog box given on next page:



Rung Number	Component to be selected from the Insert Component dialog box	Insertion point to be specified at the Command prompt	Component tag values to be specified in the Component Tag edit box of the Insert / Edit Component dialog box, if it is not already displayed	Descriptions to be specified in the Description area of the Insert / Edit Component or Insert / Edit Child Component dialog box
101	Push Button NO	8.5,13	PB101	START
101	Limit Switch, NC	9.5,13	LS101	UP
101	Relay Coil	11.5,13	CR101	CONTROL RELAY
102	Relay NO Contact	9,12	CR101	CONTROL RELAY
103	Relay NO Contact	9,11	CR101	CONTROL RELAY
103	Relay NC Contact	10.5,11	CR 101	CONTROL RELAY
103	1 Phase Motor	12.5,11	MOT103	DOWN MOTOR
104	Relay NO Contact	9,10	CR101	CONTROL RELAY
105	Relay Coil	11.5,9	CR105	CONTROL RELAY
106	Relay Coil	11.5,8	CR106	CONTROL RELAY
106	Relay NC Contact	10.5,8	CR105	CONTROL RELAY
107	Relay NO Contact	9,7	CR106	CONTROL RELAY

- After entering description and other information in the **Insert / Edit Component** or **Insert / Edit Child Component** dialog box, choose the **OK** button; the components are inserted in the drawing, as shown in Figure 6-49.

Copying the Catalog Data

- Choose the **Copy Catalog Assignment** button from the **Edit** flyout of the **Edit Components** panel of the **Schematic** tab or choose **Components > Component Miscellaneous > Copy Catalog Assignment** from the menu bar; you are prompted to select the master component. 
- Select CR101 placed on rung 101 as the master component; the **Copy Catalog Assignment** dialog box is displayed.

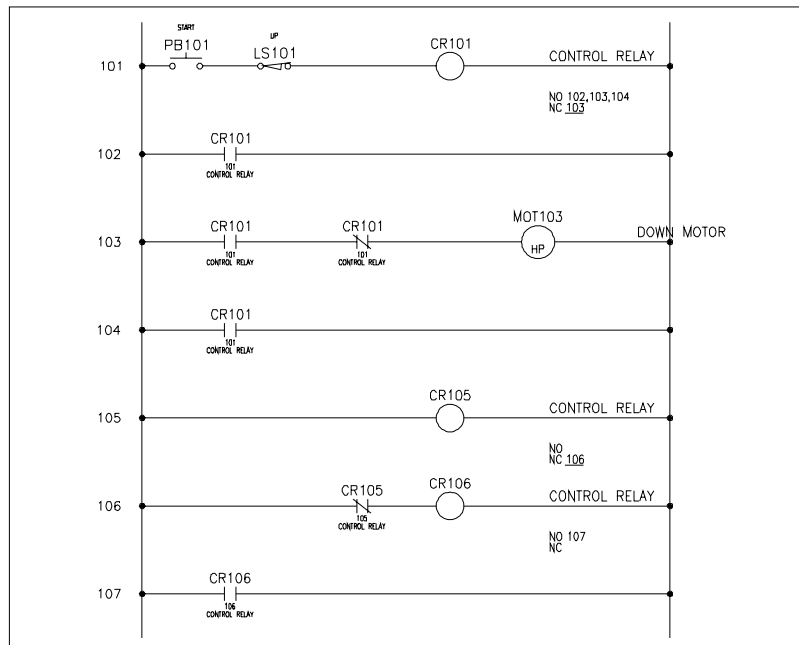


Figure 6-49 Components inserted in the drawing

3. Choose the **Catalog Lookup** button; the **Parts catalog** dialog box is displayed, as shown in Figure 6-50.
4. Select **AB** from the **MANUFACTURER** drop-down list, **TYPE P** from the **TYPE** drop-down list, and **120VAC** from the **COIL** drop-down list.
5. Select **700-P200A1** and then choose the **OK** button; **AB** is displayed in the **Manufacturer** edit box and **700-P200A1** is displayed in the **Catalog** edit box of the **Copy Catalog Assignment** dialog box.
6. Choose the **OK** button from the **Copy Catalog Assignment** dialog box; you are prompted to select the target components.
7. Select CR105 and CR106 placed on rung 105 and rung 106, respectively and press ENTER; the **Update Related Components?** dialog box is displayed. Choose the **Skip** button; the catalog data is copied to the selected components.



Note

Choose the **OK** button from the **QSAVE** dialog box, if it is displayed.

8. To check the catalog data of CR105 and CR106, right-click on these components; a shortcut menu is displayed. Next, choose the **Edit Component** option; the **Insert / Edit Component** dialog box is displayed. Now, you can notice that the catalog data information is displayed in the **Catalog Data** area of the **Insert / Edit Component** dialog box. Choose the **OK** button in this dialog box to save the changes made and exit from this dialog box.

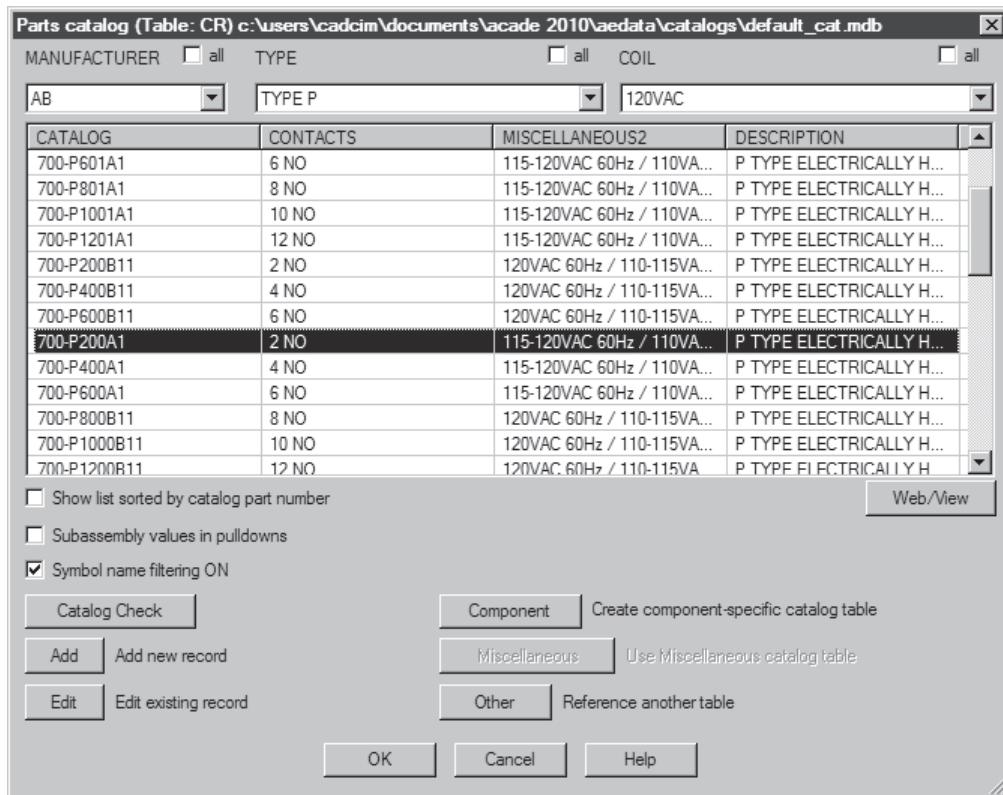


Figure 6-50 The Parts catalog dialog box

Entering the Location Data

1. Click on the down arrow given on the right of the **Edit Components** panel of the **Schematic** tab; a flyout will be displayed. Choose the **Copy Installation/Location Code Values** button from the flyout or choose **Components > Component Tagging > Copy Installation/Location Code Values** from the menu bar; the **Copy Installation/Location to Components** dialog box is displayed.
2. The **Location** check box is selected by default. If it is not selected, then select it. Enter **MCAB5** in the **Location** edit box.
3. Choose the **OK** button; you are prompted to select objects. Next, select CR105 and CR106 placed on rung 105 and rung 106, respectively, and then press ENTER; the **Update Related Components?** dialog box is displayed.
4. Choose the **Skip** button from the **Update Related Components?** dialog box; the location codes are assigned to the selected components. If the **QSAVE** dialog box is displayed, choose the **OK** button in this dialog box; the drawings get updated and MCAB5 is displayed on the CR105 and CR106, as shown in Figure 6-51.



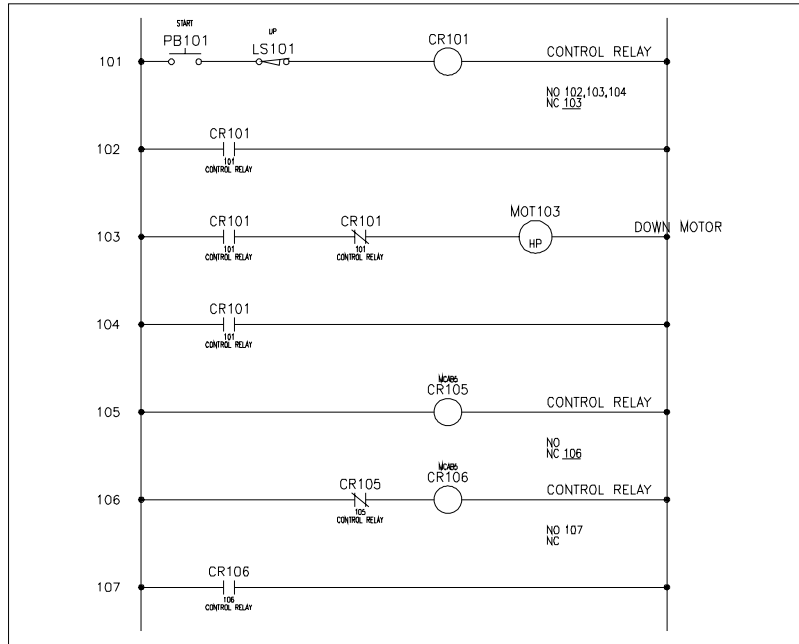


Figure 6-51 The updated drawing

Saving and Closing the Drawing File

1. Choose **Save** from the **Application Menu** to save the drawing file.
2. Choose **Close > Current Drawing** from the **Application Menu** to close the drawing file.

Self-Evaluation Test

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

1. You can scoot component in any direction. (T/F)
2. You can align components vertically only. (T/F)
3. The **Electrical Audit** button is used to correct potential errors in a project. (T/F)
4. You cannot print an auditing report. (T/F)
5. Auditing of a drawing is done to check for discrepancies in wire that can affect your design wire connectivity. (T/F)
6. The default format for component tagging is _____.

7. The _____ tool is used to change the text size and width of attributes.
8. The _____ tool is used to update or retag components.
9. The _____ tool is used to copy the selected components.
10. The _____ button of the **Electrical Auditing** dialog box is used to display the detailed information of the errors found in a project.

Review Questions

Answer the following questions:

1. Which of the following commands is used to copy components?
 - (a) **AECOMPONENT**
 - (b) **AECOPYCOMP**
 - (c) **AEWIRE**
 - (d) **AEAUDIT**
2. Which of the following tabs of the **Electrical Audit** dialog box displays the missing or duplicated wire numbers in a project?
 - (a) **Cable Exception**
 - (b) **Component - No Connection**
 - (c) **Wire Exception**
 - (d) **Wire - No Connection**
3. Which of the following buttons is used to align components vertically or horizontally?
 - (a) **Move**
 - (b) **Pick Master**
 - (c) **Scoot**
 - (d) **Align**
4. Which of the following buttons is used to change the state of a normally open (NO) component to a normally closed (NC) and vice-versa?
 - (a) **Toggle NO/NC**
 - (b) **Project-Wide Update/Retag**
 - (c) **Setup**
 - (d) **Copy Component**
5. Which of the following check boxes is selected to retag all non-fixed components of drawing (s)?
 - (a) **Component Cross-Reference Update**
 - (b) **Wire gap pointers**
 - (c) **Wire Number and Signal Tag/Retag**
 - (d) **Component Retag**
6. When you choose the **Go To** button, AutoCAD Electrical takes you to the error location in the project and rectify the error. (T/F)
7. If you select the **Title Block Update** check box, the title block information of only the active drawing will be updated. (T/F)

8. By choosing the **Move Component** button, you can reposition a selected component from its current location or wire location and insert it in a new location. (T/F)
9. The auditing tools find out errors in project drawings. (T/F)
10. The **Move/Show Attribute** tool is used to move attributes. (T/F)

Exercises

Exercise 1

In this exercise, you will create a new drawing named *C06_exer01.dwg* and insert a ladder with Width = 5, Spacing = 1, Rungs = 6, and 1st reference = 500. Also, you will insert components in the ladder, as shown in Figure 6-52. In addition, you will use the basic editing tools such as **Copy Component** and **Align Component** to copy and align the components, respectively. You will use the **Wire** tool to insert wire between rung 500 and rung 501 and the **Trim Wire** tool to trim the wire.

(Expected time: 25 min)

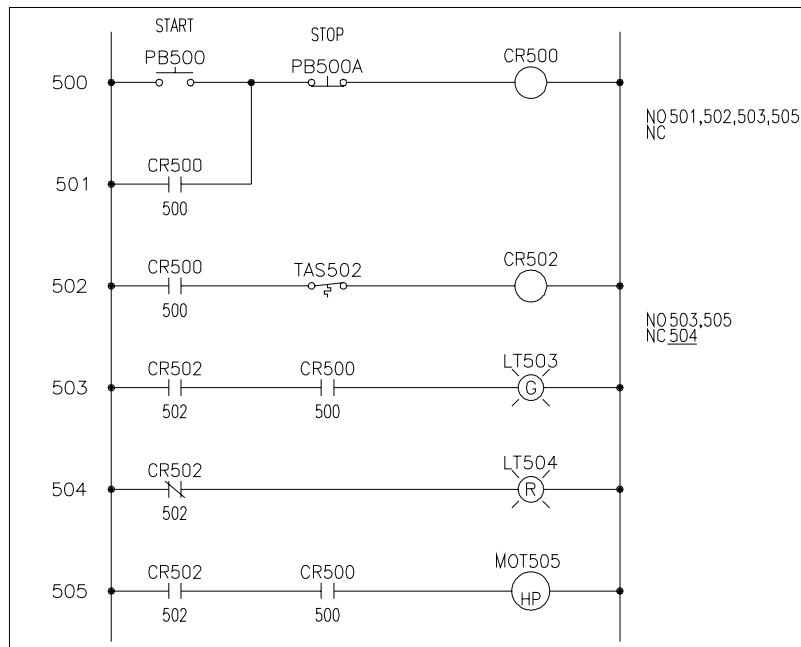


Figure 6-52 Components inserted in the ladder

Exercise 2

In this exercise, you will audit the **NEW_PROJECT** project using the **DWG Audit** and **Electrical Audit** tools. Also, you will save the report on the desktop as *drawing_audit.txt* and *electrical_audit.txt* (Expected time: 15 min)

Exercise 3

In this exercise, you will open the *C06_exer01.dwg* drawing file and save it as *C06_exer03.dwg* in the **NEW_PROJECT** project. Then, you will change component tags from reference-based to sequential using the **Update/Retag** tool. (Expected time: 20 min)

Hint: Enter **1** in the edit box on the right of the **Sequential** radio button in the **Change Each Drawing's Settings - Project-wide** dialog box.

Answers to Self-Evaluation Test

1. F, 2. F, 3. T, 4. F, 5. T, 6. %F%N, 7. Change Attribute Size, 8. Update/Retag, 9. Copy Component, 10. Details