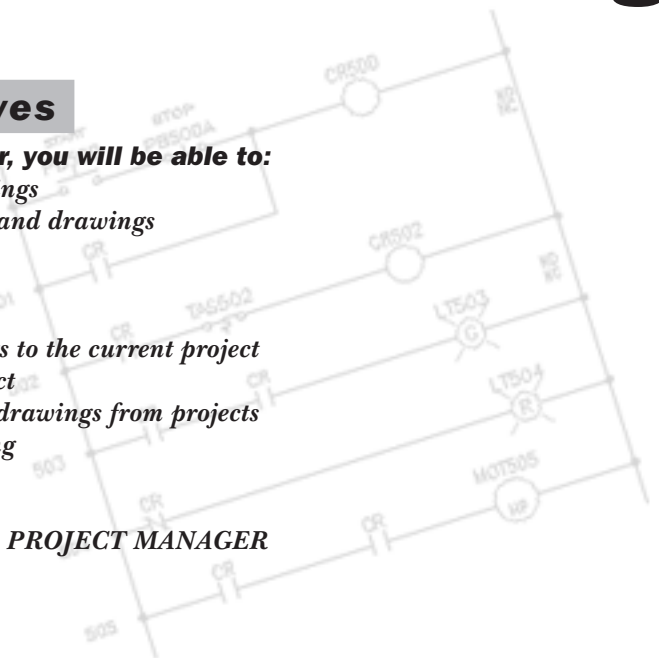
Chapter 2

Working with Projects and Drawings



Learning Objectives

After completing this chapter, you will be able to:

- *Create new projects and drawings*
- *Edit the properties of projects and drawings*
- *Add drawing descriptions*
- *Open existing projects*
- *Activate and close projects*
- *Add existing and new drawings to the current project*
- *Group the drawings of a project*
- *Rename, replace, and remove drawings from projects*
- *Assign description to a drawing*
- *Open project drawings*
- *Copy existing projects*
- *Understand the working of the PROJECT MANAGER*

INTRODUCTION

AutoCAD Electrical is a project-based software in which wiring diagrams related to each other are grouped under a project. A project is a set of electrical wiring diagrams that form a project file <*project_name*>*.wdp*. Each project is defined by an ASCII text file with *.wdp* extension. These project files contain a list of project information such as project settings, project or drawing properties, names and descriptions of drawing files, symbol library paths, and so on. You can have an unlimited number of projects. However, only one project can be active at a time. The list of these projects is displayed in the **PROJECT MANAGER**.

PROJECT MANAGER

Ribbon:	Project > Project Tools > Manager
Toolbar:	ACE:Main Electrical 2 > Project Manager or ACE:Project > Project Manager
Menu:	Projects > Project > Project Manager
Command:	AEPROJECT



The **PROJECT MANAGER** is used to create new projects, open existing projects, add new drawings to a project, re-order drawing files, access existing projects, and modify existing information in a project. By default, the **PROJECT MANAGER** is displayed and docked on the left of the screen, refer to Figure 2-1. If the **PROJECT MANAGER** is not displayed by default, choose the **Manager** tool from the **Project Tools** panel of the **Project** tab; the **PROJECT MANAGER** will be displayed.

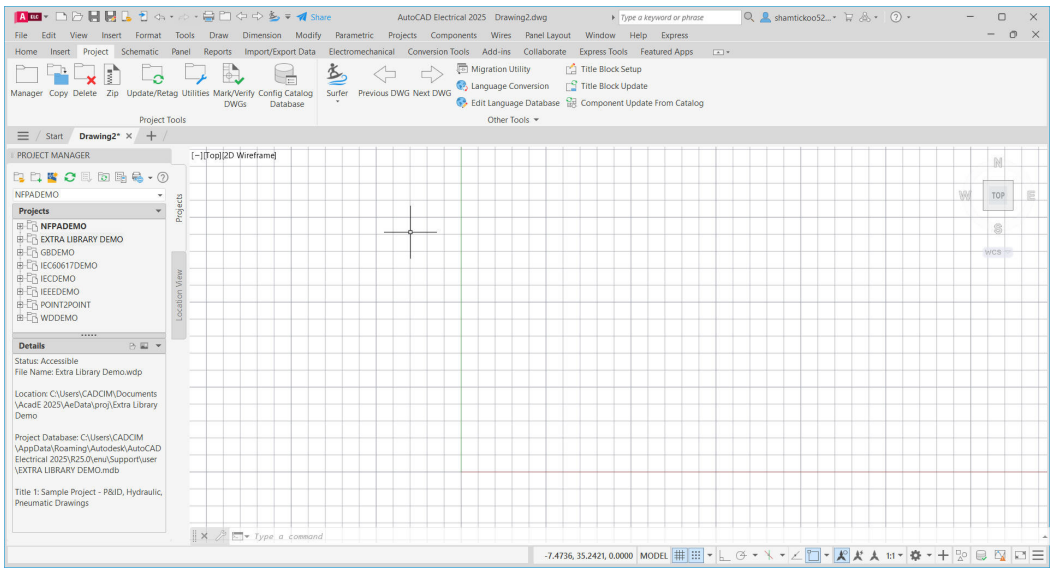


Figure 2-1 *AutoCAD Electrical screen with the* ***PROJECT MANAGER***

Alternatively, choose the **Project Manager** tool from the **ACE:Main Electrical 2** toolbar to display the **PROJECT MANAGER**. The **PROJECT MANAGER** is similar to other AutoCAD Electrical tool palettes. You can dock the **PROJECT MANAGER** at a specific location on the screen. Also, if you do not want to use the project tools, you can hide the **PROJECT MANAGER**.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

The **PROJECT MANAGER** is divided into two tabs: **Projects** and **Location View**. The options in both these tabs are discussed in detail in this chapter.

Projects TAB

In the **PROJECT MANAGER**, the **Projects** tab is chosen by default, refer to Figure 2-1. When you double-click on the title bar of the **PROJECT MANAGER**, it gets undocked and is displayed separately on the screen, refer to Figure 2-2. When you right-click on the title bar of the undocked **PROJECT MANAGER**, a shortcut menu is displayed, as shown in Figure 2-3. You can change the appearance, location, and display settings of the **PROJECT MANAGER** by choosing the respective options from the shortcut menu.

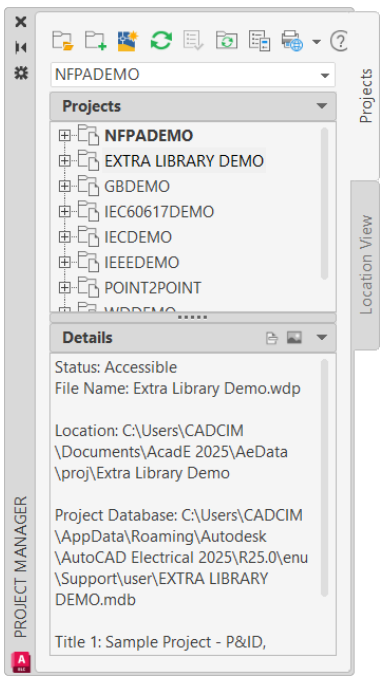


Figure 2-2 *The undocked* ***PROJECT MANAGER***

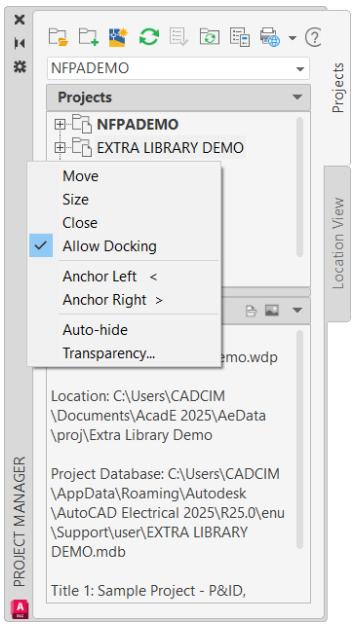


Figure 2-3 *The shortcut menu displayed by right-clicking on the title bar of the* ***PROJECT MANAGER***

When you double-click on a project name in the **Projects** rollout, a list of drawings associated with the project will be displayed. Also, the details of the selected project will be displayed in the **Details** rollout of the **PROJECT MANAGER**, as shown in Figure 2-4. The name of the active project will appear in bold text in the **Projects** rollout of the **PROJECT MANAGER**.

In AutoCAD Electrical, projects in the Project Manager are based on various drafting standards such as **IECDEMO**, **JISDEMO, GBDEMO**, **IEC60617DEMO**, **IEEEDEMO**, and so on. These projects are based on respective symbol libraries. For example, in the IECDEMO project the drawings are based on IEC symbol library whereas in the IEEEDEMO project, the drawings are based on IEEE symbol library. Detailed information about various drafting standards is provided later in this chapter.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

When you right-click on a drawing name, a shortcut menu will be displayed. You can use the options in this shortcut menu to open, close, copy, remove, replace, rename, or access other editing options to modify the drawing file. These options are discussed in detail later in the chapter. You can open a drawing file by double-clicking on it in the **PROJECT MANAGER** and the corresponding drawing file name will appear in bold text in the **Projects** rollout of the **PROJECT MANAGER**.

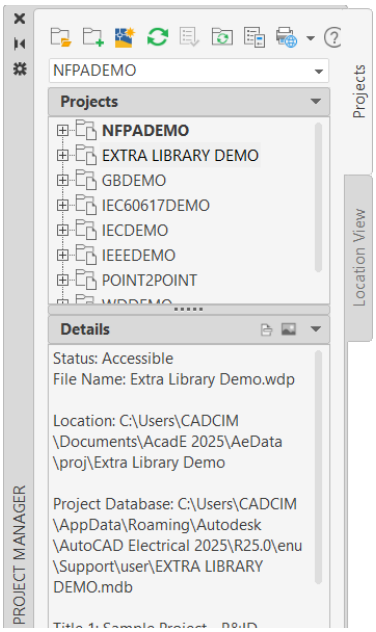


Figure 2-4 *The* ***PROJECT MANAGER*** *displaying the* ***Details*** *rollout*

Note

You cannot create two projects with the same name in the ***PROJECT MANAGER****. Moreover by using the* ***PROJECT MANAGER****, you can switch to different projects and change their settings.*

Opening a Project

You can open an existing project by using the **Open Project** button from the **PROJECT MANAGER**. On doing so, the **Select Project File** dialog box will be displayed. Next, select the existing project from this dialog box and choose the **Open** button; the selected project's name will be automatically displayed in the **Projects** rollout in bold text and it will become an active project.

In the **Select Project File** dialog box, by default, the **nfpaDemo** folder is displayed in the **Look in** drop-down list, refer to Figure 2-5. You can customize this folder. To customize the default folder, open the windows explorer. Next, navigate to the location c:*Users**User Name**Documents**Acade 2025**AeData*\ and open the *wd.env* file in Notepad. Now, scroll down in the file and go to the line highlighted in blue, refer to Figure 2-6. In this line, remove the '*' symbol available at the start of the line. Also, select the text 'x:/some path/' and

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

enter the desired path; the selected text is replaced by the desired path, refer to Figure 2-7. Next, save the *wd.env* file.



Note

If you put any extra space at the start, in between, or at the end, the desired path will not change.

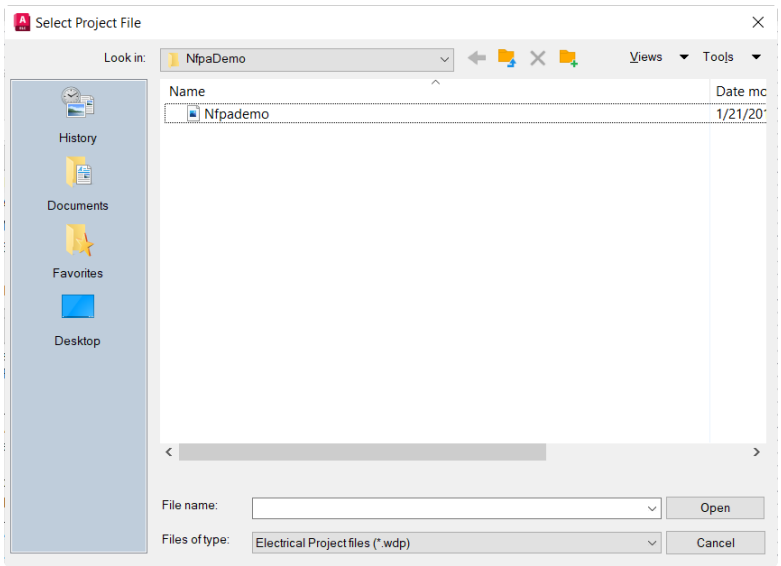


Figure 2-5 *The* ***Select Project File*** *dialog box*

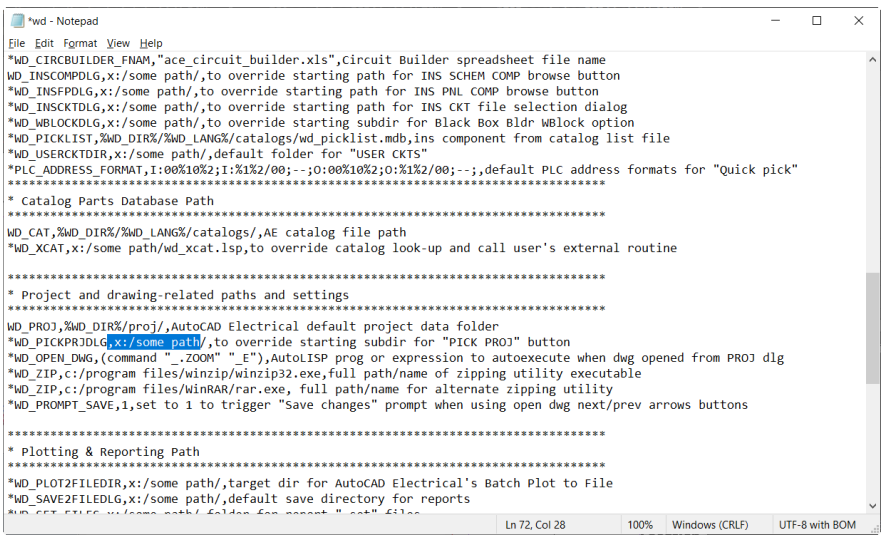


Figure 2-6 *The line highlighted in the wd.env file*

Now, choose the **Open Project** button from the **PROJECT MANAGER**; the modified **Select Project File** dialog box will be displayed, as shown in Figure 2-8. You will notice that the name of the folder is changed.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

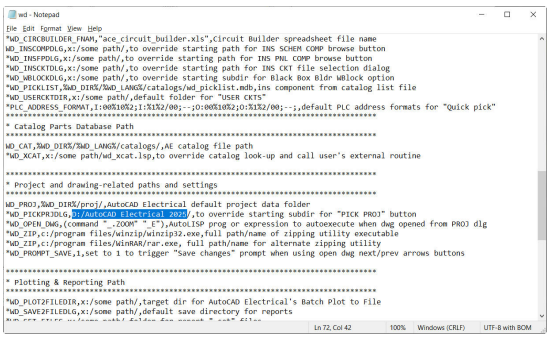


Figure 2-7 *The selected text replaced by the desired path*

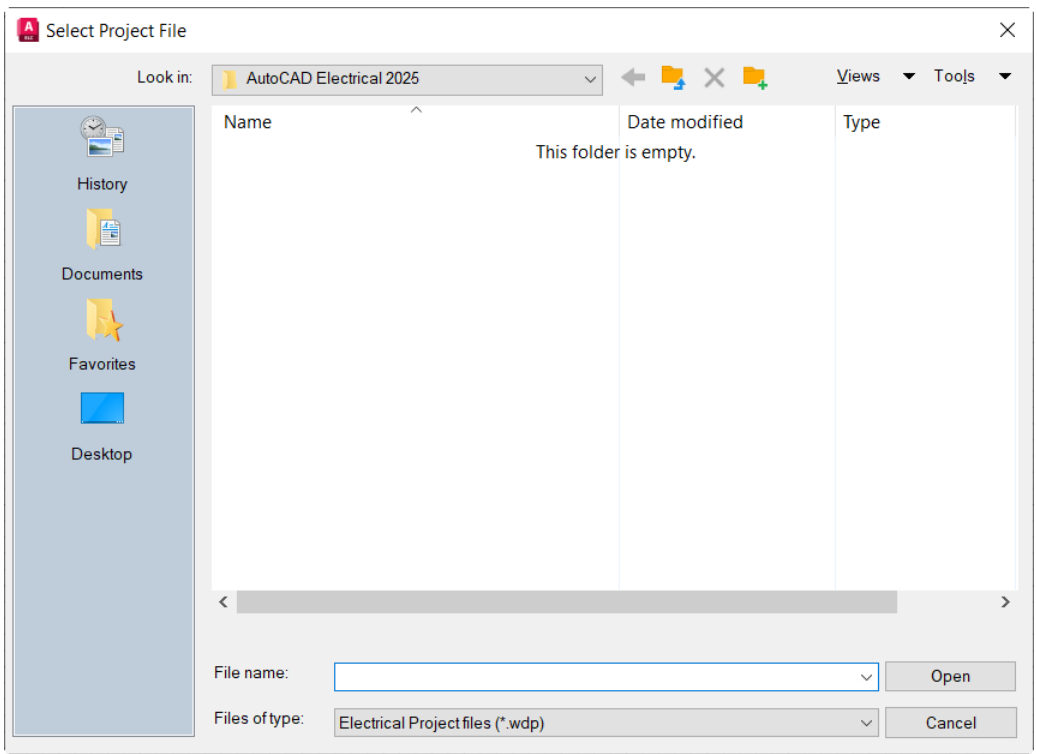


Figure 2-8 *Modified* ***Select Project File*** *dialog box*

Creating a New Project

Command: ACENEWPROJECT

You can create a new project by choosing the **New Project** button from the **PROJECT MANAGER**. On doing so, the **Create New Project** dialog box will be displayed, as shown in Figure 2-9. Alternatively, use the **ACENEWPROJECT** command to create a new project. Different options in the **Create New Project** dialog box are discussed next.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

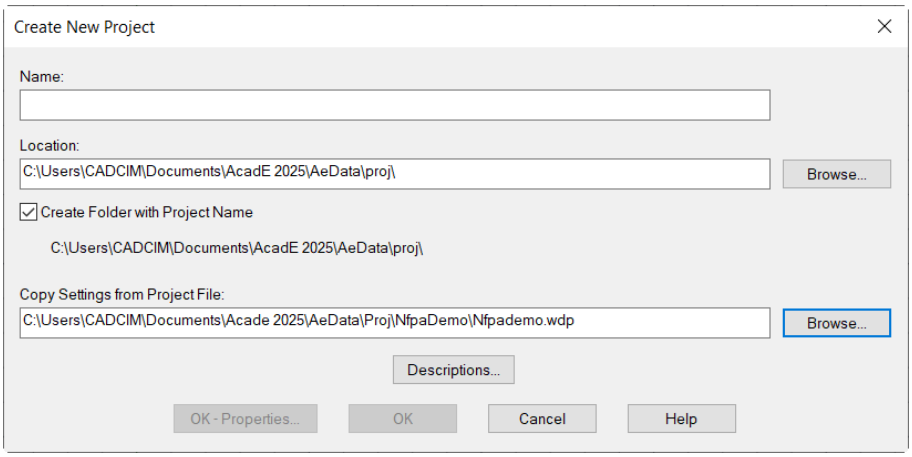


Figure 2-9 *The* ***Create New Project*** *dialog box*

Name

The **Name** edit box is used to enter a name for the project. On doing so, the *.wdp* extension will be automatically added to the file name and displayed under the **Create Folder with Project Name** check box.

Location

The **Location** edit box is used to specify the location for saving the project. You can also choose the **Browse** button on the right of the **Location** edit box to specify the location for saving the project file (*.wdp file*). By default, *C:\Users\User Name\Documents\Acade 2025\AeData\proj* is displayed in the **Location** edit box, refer to Figure 2-9.

Create Folder with Project Name

The **Create Folder with Project Name** check box is selected by default in this dialog box. As a result, a folder with a name identical to the project name will be created. Also, the new project will be saved in that folder. The path of the folder will be the same as that defined in the **Location** edit box. If you clear the **Create Folder with Project Name** check box, a separate folder will not be created for the project.

Copy Settings from Project File

You can also copy the settings from an existing project and apply it to a new project. To copy the settings from the existing project file, specify the name and location of the existing project file in the **Copy Settings from Project File** edit box. Alternatively, choose the **Browse** button on the right of this edit box; the **Select Project File** dialog box will be displayed. Select an existing project file and then choose the **Open** button; the settings of the specified project will be applied to the new project and the location of the specified project will be displayed in the **Copy Settings from Project File** edit box. By default, *C:\Users\User Name\Documents\Acade 2025\AeData\proj\NFPAdemo\NFPAdemo.wdp* is displayed in the **Copy Settings from Project File** edit box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Descriptions

The **Descriptions** button is used to add a description to the project. Choose the **Descriptions** button; the **Project Description** dialog box will be displayed, as shown in Figure 2-10. In this dialog box, you can enter a description of the new project up to 12 lines per page. If you want to add more than 12 descriptions, click on the **>** button at the bottom of this dialog box. To include the description in the reports, select the **in reports** check box on the right of each description line. Note that the descriptions thus added will be included in the report headers of the report file and the title blocks of the drawing. Now, choose the **OK** button from this dialog box to save the changes made and exit the dialog box. The concept of report generation, title blocks, and **Project Description** dialog box is discussed in detail in the later chapters.

OK

The **OK** button of the **Create New Project** dialog box will be activated only if you enter the name of the project in the **Name** edit box. Choose the **OK** button; the project that you created will be added to the current project lists in the **PROJECT MANAGER**. Also, it will become an active project and its name will appear in bold text in the **PROJECT MANAGER**.

OK-Properties

The **OK-Properties** button will be activated only if you enter the project name in the **Name** edit box. Choose the **OK-Properties** button; a new project will be created and the **Project Properties** dialog box will be displayed. This dialog box is used to modify project settings, components, wire numbering, cross-references, styles, and drawing formats. All information defined in the **Project Properties** dialog box is saved to the project definition file (*.wdp*) as project and drawing defaults. Next, choose the **OK** button from this dialog box to exit from it. The options in the **Project Properties** dialog box are discussed later.

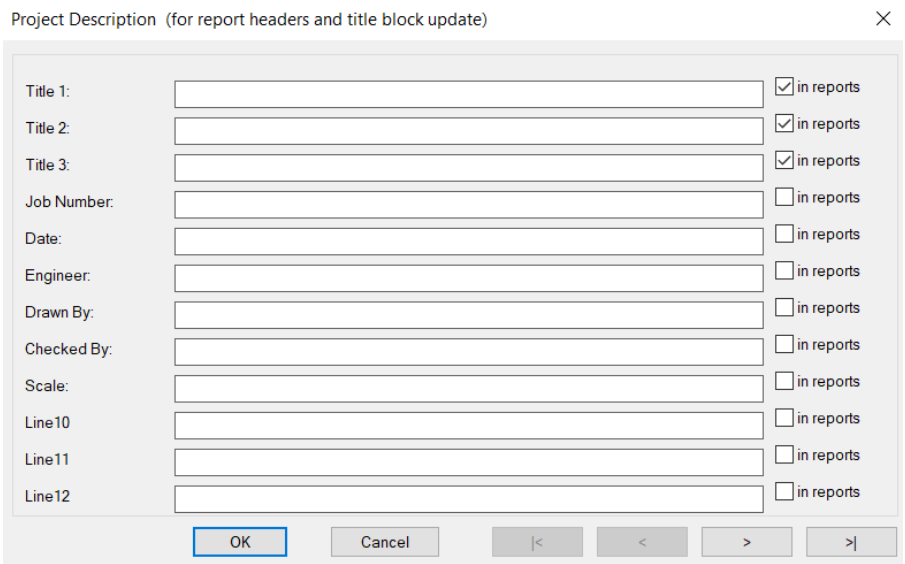


Figure 2-10 *The* ***Project Description*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Note

*You can also create a new project by right-clicking inside the **Projects** rollout and then choosing the **New Project** option from the shortcut menu displayed or by selecting the **New Project** option from the Project selection drop-down list in the **PROJECT MANAGER**. You will learn more about it later in this chapter.*

Working with Drawings

In the previous topic, you learned to create new projects using the **PROJECT MANAGER**. Now, you will learn to create new drawings within an active project.

You can add any number of drawings to your project at any time. A project file can have drawings located in different directories. But, it is recommended that you save your drawings and project file (*.wdp*) in the same folder. When you create a new drawing, an invisible smart block, WD_M, will automatically be added to your drawing at location 0,0. The WD_M block defines drawing settings. These settings may be different from the project settings. Thus, you can have different settings for different drawings in a single project. Also, each AutoCAD Electrical drawing should contain only one copy of invisible WD_M block. If multiple WD_M blocks are present in the drawing, the settings may not be stored and read consistently. Also, note that the drawing to be created will automatically get added to the active project.

Creating a New Drawing

Command: ACENEWDRAWING



The **New Drawing** button is used to create a new drawing file. To do so, choose the **New Drawing** button from the **PROJECT MANAGER**; the **Create New Drawing** dialog box will be displayed, as shown in Figure 2-11.

Alternatively, invoke this dialog box by using the **ACENEWDRAWING** command or by right-clicking on the **Projects** rollout of the **PROJECT MANAGER** and then choosing the **New Drawing** option from the shortcut menu displayed. The different areas and options in this dialog box are discussed next.

Drawing File Area

The options in the **Drawing File** area are used to specify the name of a drawing file, the template file to be used in the drawing, the location to save the drawing, and the description for the new drawing file. The options in this area are discussed next.

Name: Enter a name for the new drawing in the **Name** edit box; the *.dwg* extension will automatically be added to the drawing name. Note that the **OK** and **OK - Properties** buttons will be activated only after entering the name of the new drawing in this edit box.

Template: This edit box is used to specify the path and name of a template drawing (*.dwt*) for creating a drawing file. Alternatively, you can choose the **Browse** button; the **Select template** dialog box will be displayed. Select the template drawing from this dialog box and choose the **Open** button; the location of the template drawing will be displayed in the **Template** edit box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

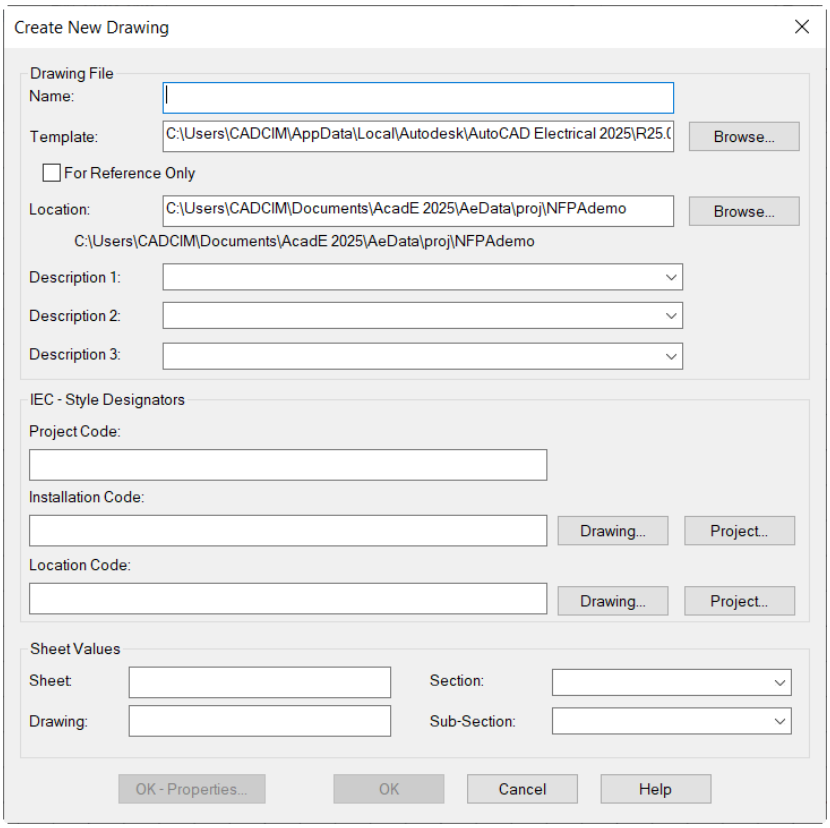


Figure 2-11 *The* ***Create New Drawing*** *dialog box*

Note

After specifying the template in the ***Template*** *edit box, next time when you create a new drawing, the template entered previously will automatically get displayed in the* ***Template*** *edit box. If this edit box is left blank, AutoCAD Electrical will use the default acad.dwt file.*

For Reference Only: The reference drawings are used for coversheets, terminal plans, and other non-electrical layouts in a project. These drawings save the processing time of AutoCAD Electrical functions. To create a reference drawing, you need to select the **For Reference Only** check box. On doing so, the reference drawing will be included in the project-wide plotting and title block update operations. All other electrical smart functions such as cross-referencing, automatic tagging, reporting, and so on will be non-functional. By default, the **For Reference Only** check box is cleared.

Note

The color of the reference drawing icon displayed on the left of the drawing name in the ***PROJECT MANAGER*** *is gray.*

Location: The **Location** edit box is used to specify the location of a new drawing. The directory of the active project file is specified by default in the **Location** edit box. Alternatively, choose the **Browse** button on its right to specify different location for the new drawing. If you

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

leave the **Location** edit box blank, the drawing file will be created at the same location as that of the active project. Note that you cannot create duplicate drawings at the same location.

Description 1-3: The **Description 1**, **Description 2**, and **Description 3** edit boxes are used to enter description for a drawing. In these edit boxes, you can enter up to 3 description lines for a drawing file. You can also specify the description for a drawing by selecting it from the description drop-down lists. But this is possible only if you have entered description in the earlier drawings. The description thus entered will be displayed in the title block updates and custom drawing properties.

OK

The **OK** button will be activated only if you enter a drawing file name in the **Name** edit box. When you choose the **OK** button, the new drawing gets automatically added at the bottom of the list in the active project and it appears in bold text. To view the drawing file, expand the active project by double-clicking on the project name.

OK - Properties

The **OK - Properties** button will be activated only if you enter a drawing name in the **Name** edit box of the **Drawing File** area. Choose the **OK - Properties** button; a drawing file will be created at the specified location and also the **Drawing Properties** dialog box will be displayed, as shown in Figure 2-12.

This dialog box is used to define settings and options for a drawing. The options in the **Drawing Properties** dialog box will be discussed in the later chapters. The changes that you make using this dialog box will be saved as attribute values on the drawing's invisible WD_M block. Choose the **OK** button from the **Drawing Properties** dialog box; the drawing will be created and will appear in bold text at the bottom of the project list of the active project in the **PROJECT MANAGER**.



Note

*Mentioning information in the **Description** edit boxes, the **IEC-Style Designators** area, the **Sheet values** area as well as choosing the **OK - Properties** button is optional. You can also edit any of these fields later in the **Drawing Properties** dialog box.*



Tip

The IEC style designators are widely used when you are using the IEC drafting standard. You must be aware of the drafting standards used in AutoCAD Electrical. The software has been designed to use seven International Drafting Standards which are listed below:

1. Joint Industrial Council ---- JIC1/JIC125 (USA)
2. International Electrotechnical Commission ---- IEC2/IEC 60617 (Europe, UK)
3. Japanese International Standard ---- JIS2 (Japan)
4. Guobiao Standard ---- GB2 (Chinese)
5. Australian Standard ---- AS2 (Australia)
6. Institute of Electrical and Electronics Engineers Standard ---- IEEE 315/315A
7. National Fire Protection Association Standard ---- NFPA (USA)

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

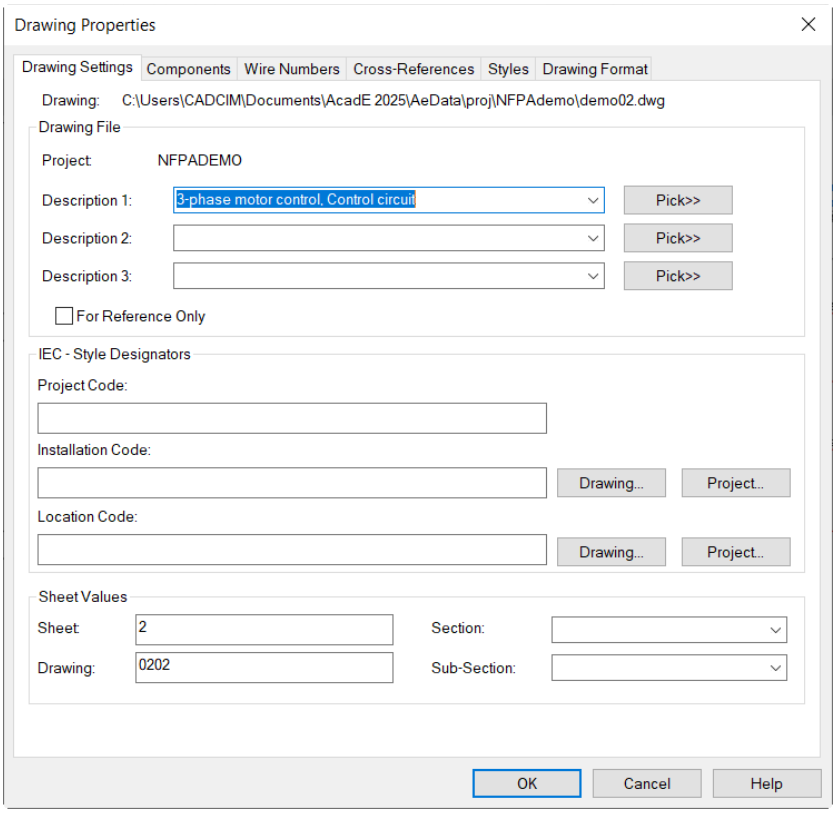


*Figure 2-12 The **Drawing Properties** dialog box*

Adding a New Drawing to the Inactive Project

You can add a new drawing to the inactive project using the **PROJECT MANAGER**. To do so, choose the **New Drawing** button from the **PROJECT MANAGER**; the **Create New Drawing** dialog box will be displayed. Enter the required information in this dialog box, as explained earlier and then choose the **OK** button; a new drawing will be created. Also, this drawing will be automatically added to the active project and will appear in bold text at the end of the Project Drawing list. Next, to add this drawing to an inactive project, right-click on the inactive project name and choose the **Add Active Drawing** option from the shortcut menu displayed; the **Apply Project Defaults to Drawing Settings** message box will be displayed, as shown in Figure 2-13.

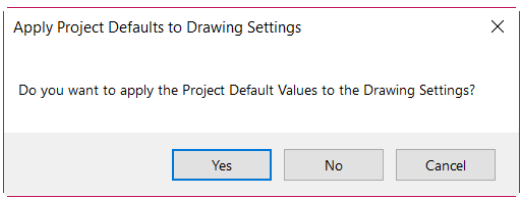


*Figure 2-13 The **Apply Project Defaults to Drawing Settings** message box*

Choose the **Yes** button from the **Apply Project Defaults to Drawing Settings** message box; the default values of the project will be added to the WD_M block definition of the newly added

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

drawing. If you choose the **No** button, the new drawing will retain its existing settings. Choose the **Cancel** button to exit the command.

Note
You can have a single drawing in multiple projects.

Adding Existing Drawings to the Current Project

You can also add the existing drawings to the current project. To do so, right-click on the project name in the Projects rollout of the **PROJECT MANAGER**; a shortcut menu will be displayed, as shown in Figure 2-14. Choose the **Add Drawings** option from the shortcut menu; the **Select Files to Add** dialog box will be displayed, as shown in Figure 2-15. In this dialog box, select the desired project folder from the **Look in** drop-down list and then select drawings to be added to the project. Next, choose the **Add** button; the **Apply Project Defaults to Drawing Settings** message box will be displayed, refer to Figure 2-13.

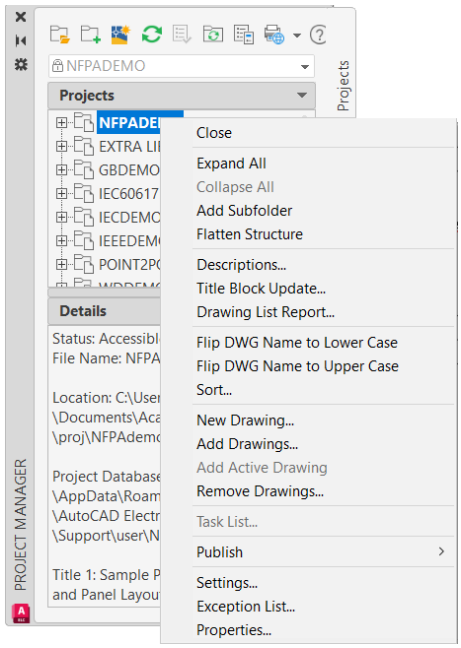


Figure 2-14 *The shortcut menu displayed by right-clicking on the active project name in the* ***PROJECT MANAGER***

If you choose the **Yes** button, the default values of the project will be added to the WD_M block definition of the newly added drawing. But if you choose the **No** button, the new drawing will retain its existing settings. Also, the selected drawings will be added to your project and will appear at the end of the Project Drawing list.

Note
You can select multiple drawing files at a time from the ***Select Files to Add*** *dialog box by pressing the SHIFT or CTRL key.*

Working with Project Drawings

You can group drawings, remove drawings, assign description to drawings, preview drawings, configure the drawing list display, and so on by using the **PROJECT MANAGER**. Also, you can access the existing project and modify its related information using the **PROJECT MANAGER**.

Grouping Drawings within a Project

You can group the drawings in a project in two ways: grouping using section/subsection and grouping using subfolders. These methods are discussed next.

Grouping Drawings Using Section/ Subsection

You can create a group of drawings within a project list by assigning the section and sub-section codes to each drawing. To do so, expand the project by double-clicking on its name, if it is not already expanded; the drawings in that project will be displayed. Next,

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

right-click on the drawing file; a shortcut menu will be displayed. Choose **Properties > Drawing Properties** from the shortcut menu; the **Drawing Properties** dialog box will be displayed, refer to Figure 2-12. In this dialog box, the **Drawing Settings** tab is chosen by default. Enter the required section and sub-section codes for the drawing in the **Sheet Values** area and choose the **OK** button; the section and sub-section codes will get assigned to the drawing file. Similarly, you can repeat the same procedure for each drawing that you want to group together, but ensure that the same section and sub-section codes are assigned to all of them.

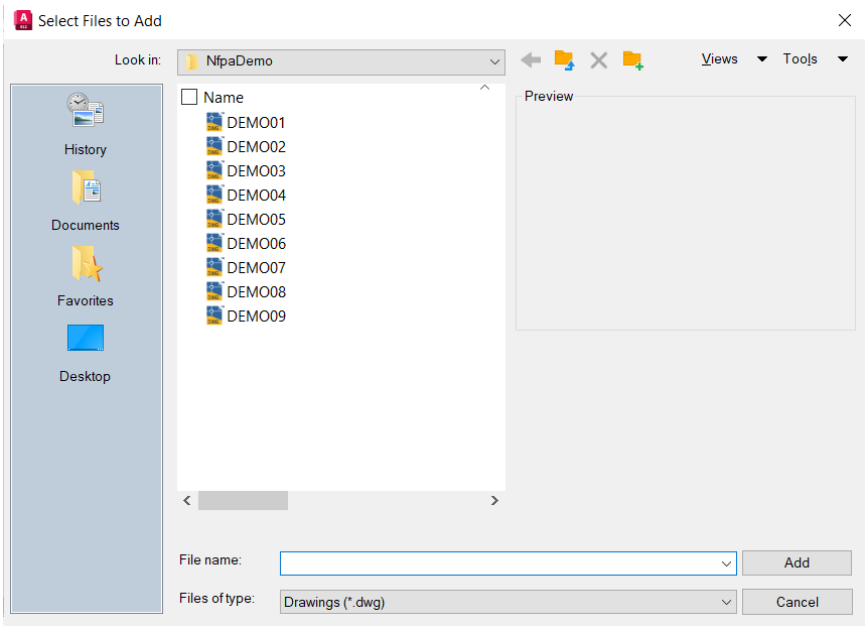


***Figure 2-15** The **Select Files to Add** dialog box*

Grouping Drawings Using Subfolders

You can create subfolders within a project to organize the drawings. To create a subfolder, right-click on the project; a shortcut menu will be displayed. Choose **Add Subfolder** from it; a subfolder with the name **NEW FOLDER** will be created at the bottom of the drawing list. Rename it with a suitable name. Now, you can drag and drop the drawings available in the project into the subfolder. When you right-click on the subfolder; a shortcut menu will be displayed. The options in the shortcut menu are used to create and add drawings, rename the subfolder, and so on. You can add any number of subfolders in a project. Also, you can add subfolders within a subfolder.

You can also remove all the subfolders from an active project. To do so, right-click on an active project and choose **Flatten Structure** from the shortcut menu displayed; the **Flatten Project Structure** message box will be displayed warning you that if the process of flattening is continued, you will not be able to revert back the changes again. Note that when you use this option only subfolders are removed and the drawings are retained in the project.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

After creating a group of drawings and subfolders, the project-wide tagging, cross-referencing, and reporting functions can be performed on the whole project or a part of the drawing set, using the section, sub-section coding, and subfolders.

Changing the Order of Drawings in a Project

In the Project Drawing list, the drawings created are arranged in the same sequence as they are processed by AutoCAD Electrical during the project-wide tagging and cross-referencing operations. However, you can change the order of the drawings present in a project by dragging and dropping the drawings at the required place in a sequence. You can also reorder the drawings by moving them between two subfolders of a project. Note that you cannot reorder the drawings by moving them between two projects. You can press the CTRL or SHIFT key to select multiple drawings.

You can also sort the drawings in a project to change their order. To do so, right-click on the project name and choose the **Sort** option from the shortcut menu displayed. If the project contains subfolder(s), the **Sort Options** message box will be displayed showing four different combinations of options to sort the drawings in the project folder and its subfolder(s), refer to Figure 2-16. Choose one of these options as per your requirement; the **Sort** dialog box will be displayed, as shown in Figure 2-17.

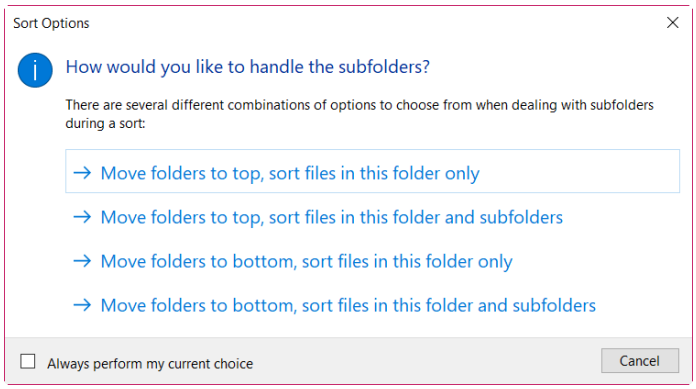


***Figure 2-16** The **Sort Options** message box*

You can sort out the drawings in different ways depending upon the options selected from the **Sort** dialog box. You can select the required option from the **Primary sort**, **Secondary sort**, **Third sort**, and **Fourth sort** drop-down lists. These drop-down lists include **REF** (reference status), **SEC** (section code), **SUBSEC** (sub-section), and **FILEPATHANDNAME** (file path and name) options.

Some projects do not contain subfolder(s). If you right-click on such a project and choose the **Sort** option from the shortcut menu displayed; the **Sort** dialog box will be displayed, refer to Figure 2-17.

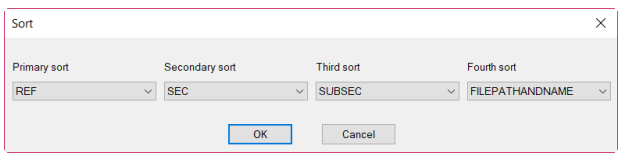


***Figure 2-17** The **Sort** dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

After setting the required parameters, choose the **OK** button from the **Sort** dialog box; the dialog box is closed and the drawing displayed in the list will be arranged according to the options selected from the drop-down lists.

Removing a Drawing from a Project

You can remove a drawing from a project using two methods. These methods are discussed next.

First Method

Right-click on the drawing name in the **PROJECT MANAGER**; a shortcut menu will be displayed. Choose the **Remove** option from the shortcut menu; the **PROJECT MANAGER - Remove Files** message box will be displayed. Choose the **Yes** button from the message box; the drawing will be removed from the project, but it will not be deleted from the folder where it is stored.

Second Method

Right-click on a project name; a shortcut menu will be displayed. Next, choose the **Remove Drawings** option from the shortcut menu; the **Select Drawings to Process** dialog box will be displayed, as shown in Figure 2-18. The options in this dialog box are discussed next.

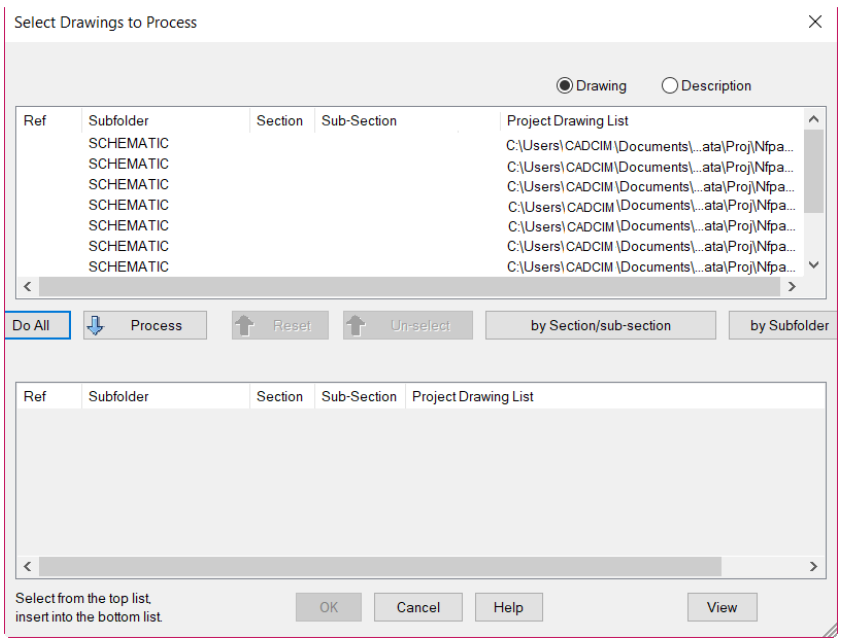


Figure 2-18 *The* ***Select Drawings to Process*** *dialog box*

Drawing: The **Drawing** radio button is used to display the path and the name of all drawings in a project. This radio button is selected by default.

Description: Select the **Description** radio button; the description of all drawings will be displayed in the top list of the **Select Drawings to Process** dialog box. Remember that the description will be displayed only if you have entered the description in the **Description**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

1/2/3 edit boxes in the **Create New Drawing** dialog box or in the **Description 1/2/3** edit boxes of the **Drawing Settings** tab in the **Drawing Properties** dialog box.

Do All: Choose the **Do All** button to transfer all drawings from the top list to the bottom list of the **Select Drawings to Process** dialog box.

Process: The **Process** button will be activated only if you select a drawing(s) from the top list. This button is used to transfer the selected drawing from the top list to the bottom list. To do so, select a drawing(s) and then choose the **Process** button; the selected drawing(s) will be transferred from the top list to the bottom list of the **Select Drawings to Process** dialog box. You can use the SHIFT or CTRL key to select more than one drawing at a time.

Reset: The **Reset** button will be activated only if you have transferred drawing(s) from the top list to the bottom list. Choose the **Reset** button to reset the drawings to their original positions in the dialog box.

Un-select: The **Un-select** button will be activated only if you select drawing(s) from the bottom list. The **Un-select** button is used to transfer drawing from the bottom list to the top list. To do so, select a drawing(s) from the bottom list and then choose the **Un-select** button; the drawings will be moved from the bottom list to the top list. You can unselect multiple drawings at a time by using the SHIFT or CTRL key.

by Section/sub-section: The **by Section/sub-section** button is used to select the drawings that have been grouped by specifying their sections and sub-sections. To do so, choose the **by Section/sub-section** button from the **Select Drawings to Process** dialog box; the **Select Drawings by Section/sub-section** dialog box will be displayed. Specify the section and sub-section codes in the **Section** and **Sub-section** edit boxes, respectively. Alternatively, select the section and sub-section codes from the **Section** and **Sub-section** drop-down lists, respectively. Next, choose the **OK** button from the **Select Drawings by Section/sub-section** dialog box; the corresponding drawing(s) will be transferred from the top list to the bottom list. Note that the **Section** and **Sub-section** drop-down lists display the section and sub-section codes only if you have entered the section and sub-section values in the **Sheet Values** area of the **Create New Drawing** dialog box or in the **Drawing Settings** tab of the **Drawing Properties** dialog box.

by Subfolder: The **by Subfolder** button is used to select the drawings that have been grouped in subfolders. To do so, choose the **by Subfolder** button from the **Select Drawings to Process** dialog box; the **Select Drawings by Subfolder** dialog box will be displayed. Select the subfolder(s) from this dialog box and choose the **OK** button; the drawings from the selected subfolder(s) will be transferred from the top list to the bottom list. Note that the **Select Drawings by Subfolder** dialog box displays the subfolders only if you have added subfolders in the project.

After selecting drawings from the **Select Drawings to Process** dialog box, choose the **OK** button; the **Remove Drawing(s) from Project List** message box will be displayed. In this message box, choose the **OK** button; the selected drawings will be removed from the project list. Note that the drawings will instantly be removed from the project list, but they will not be deleted permanently from the folder where they are stored.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Assigning a Description to a Drawing

You can assign a three-line description to each drawing listed in your project. To do so, right-click on the name of a drawing in the project; a shortcut menu will be displayed. Next, choose **Properties > Drawing Properties** from the shortcut menu; the **Drawing Properties** dialog box will be displayed.

By default, the **Drawing Settings** tab is chosen in the **Drawing Properties** dialog box. Enter a description for the drawing in the **Description 1/2/3** edit boxes. You can also select the predefined description for your drawing from the **Description** drop-down list, refer to Figure 2-19. Note that the predefined description will be available only if you have entered the description for any of the drawings of a project. This figure shows the predefined descriptions listed in the **Description 1** drop-down list of the **Drawing Properties** dialog box. Next, choose the **OK** button; the drawing file will be updated and the changes will be saved.

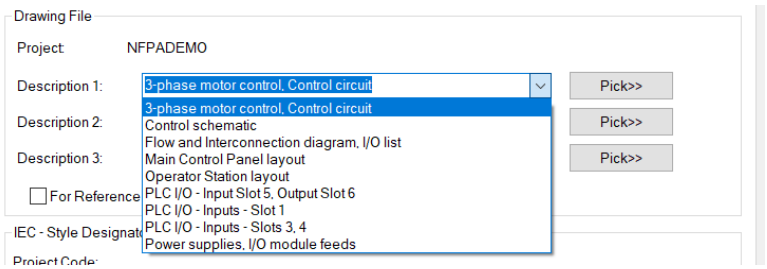


Figure 2-19 *Partial view of the* ***Drawing Properties*** *dialog box displaying predefined description*



Note

The description specified in the ***Description 1****,* ***Description 2****, and* ***Description 3*** *edit boxes of the* ***Create New Drawing*** *or* ***Drawing Properties*** *dialog box can be linked to the attribute in the title block for automatic update and will be discussed in the later chapters.*

After adding description to a drawing, you can preview the description in the **Details** rollout of the **PROJECT MANAGER**. These drawing details give a unique identity to the drawings based on users requirement and helps users to search a particular drawing among multiple drawings in a project file.

Switching between the Drawings of an Active Project

Ribbon:	Project > Other Tools > Previous DWG
	Project > Other Tools > Next DWG
Toolbar:	ACE:Main Electrical 2 > Previous Project Drawing
	ACE:Main Electrical 2 > Next Project Drawing
	ACE:Quick Pick > Previous Project Drawing
	ACE:Quick Pick > Next Project Drawing
Command:	AEPREV and AENEXT



The **Previous DWG** and **Next DWG** tools are used to open the previous and next drawings of an active project. Using these tools, you can view and switch between various drawings of a project. To do so, open any drawing of the active project and

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

then choose the **Previous DWG** or **Next DWG** tool from the **Other Tools** panel of the **Project** tab. Alternatively, you can choose the **Previous Project Drawing** or **Next Project Drawing** tool from the **ACE:Main Electrical 2** or the **ACE:Quick Pick** toolbar. On doing so, the currently opened drawing will be closed and all changes made to it will be saved. Also, the requested drawing will open. The currently opened drawing will appear in bold text in the project list.



Tip

*To open all drawings of an active project in a new window without closing the original drawing window, hold the SHIFT key while choosing the **Previous Project Drawing** or the **Next Project Drawing** button.*



Note

1. *You cannot view and switch among the drawings that are not associated with the active project.*
2. *You can use the **PROJECT MANAGER** to preview the drawings easily using the **Preview** button.*
3. *If you move among various drawings using the up and down arrow keys, the selected drawing will not open.*

Configuring the Drawing List Display



The **Drawing List Display Configuration** button is used to configure the information related to various drawings in the **Projects** rollout of the **PROJECT MANAGER**. This button is used to display the required information such as drawing number, description, and so on. By default, only the name of the drawing is displayed in the **Projects** rollout of the **PROJECT MANAGER**, as shown in Figure 2-20. Choose the **Drawing List Display Configuration** button from the **PROJECT MANAGER**; the **Drawing List Display Configuration** dialog box will be displayed, as shown in Figure 2-21. The options in this dialog box are discussed next.

Display Options Area

The **Display Options** area displays the predefined values that can be associated to a drawing.

Current Display Order

The **Current Display Order** area lists the order in which the required display options will be displayed in the **Projects** rollout in the **PROJECT MANAGER**. The **File Name** option is displayed by default in this area. You can add other display options in this area by selecting them from the **Display Options** area and then choose **>>** button.

>>

The **>>** button is used to move only the selected display option from the **Display Options** area to the **Current Display Order** area.

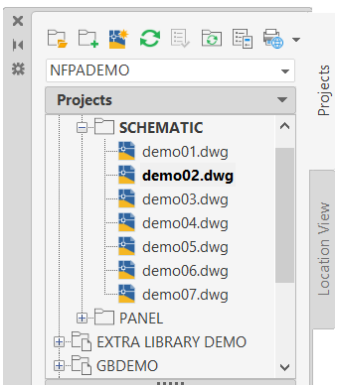


***Figure 2-20** Partial view of the **PROJECT MANAGER** displaying the drawing files of the project*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

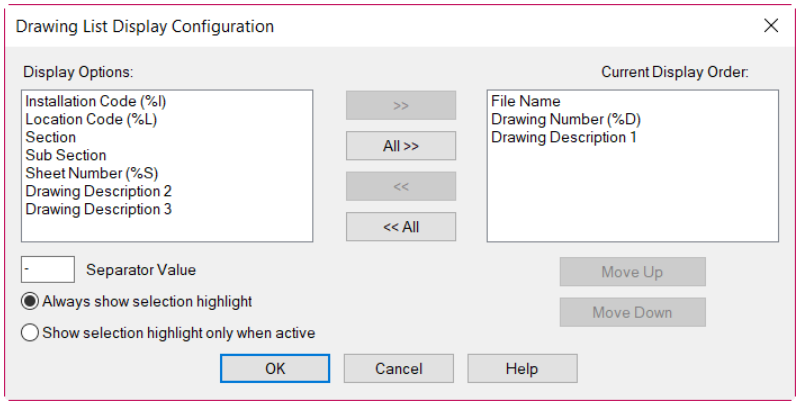


*Figure 2-21 The **Drawing List Display Configuration** dialog box*

All>>

The **All>>** button is used to move all display options from the **Display Options** area to the **Current Display Order** area.

<<

The **<<** button is used to move only the selected display option from the **Current Display Order** area to the **Display Options** area.

<<All

The **<<All** button is used to move all display options from the **Current Display Order** area to the **Display Options** area.

Separator Value

The **Separator Value** edit box is used to specify the character to be used as a separator between the values in the listing. You can enter a character in the **Separator Value** edit box or use the default character (-) in this edit box.

Move Up

Choose the **Move Up** button to move the selected display option one step up in the **Current Display Order** area.

Move Down

Choose the **Move Down** button to move the selected display option one step down in the **Current Display Order** area.

Choose the **OK** button in the **Drawing List Display Configuration** dialog box; the entire information related to drawings will be displayed in the **Projects** rollout, refer to Figure 2-22.

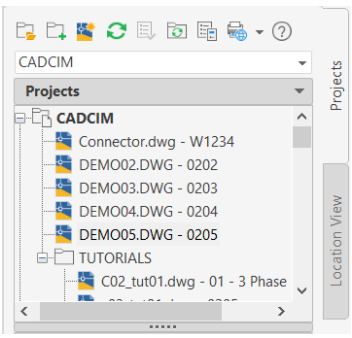


*Figure 2-22 Partial view of the **PROJECT MANAGER** displaying the drawing information*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Note

*The drawing options are displayed in the **Current Display Order** area in a particular sequence. The same sequence is followed for their display in the **Projects** rollout of the **PROJECT MANAGER**. If you want to change the sequence, choose the **Move Up** or **Move Down** button.*

Copying a Project



Ribbon:	Project > Project Tools > Copy
Toolbar:	ACE:Main Electrical 2 > Project Manager drop-down > Copy Project or ACE:Project > Copy Project
Menu:	Projects > Project > Copy Project
Command:	AECOPYPROJECT

The **Copy** tool is used to copy the entire project as well as copy the drawings present within that project. You can copy the project in a single step. To copy the project, choose the **Copy** tool from the **Project Tools** panel of the **Project** tab; the **Copy Project: Select Existing Project to Copy** dialog box will be displayed, as shown in Figure 2-23.

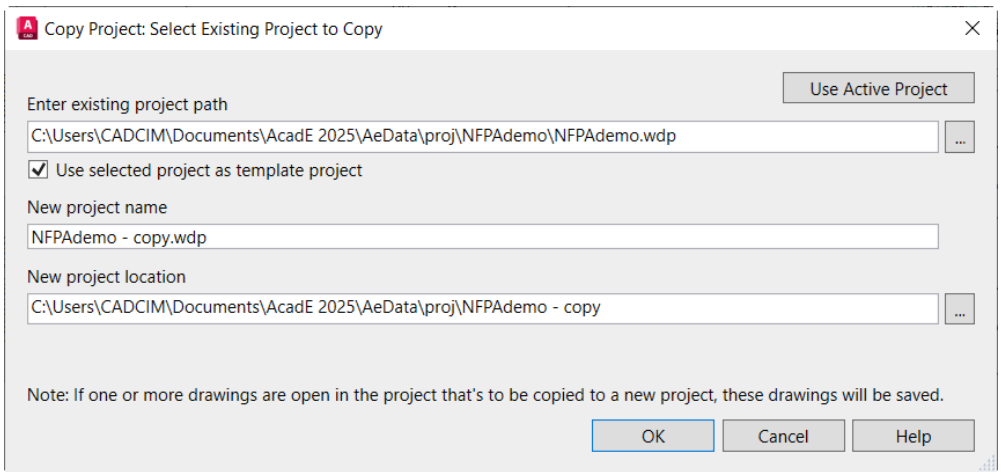


***Figure 2-23** The **Copy Project: Select Existing Project to Copy** dialog box*

Next, enter the name and path of the existing project in the **Enter existing project path** edit box. Alternatively, choose the **Browse** button next to this edit box to select the existing project; the **Open** dialog box will be displayed. Next, navigate to the project folder and double-click on it. Next, select the project's *.wdp* file and choose the **Open** button; the name and path of the existing project will be displayed in the **Enter existing project path** edit box. You can also choose the **Use Active Project** button available in this dialog box to copy the currently active project.

Enter a name for the new project in the **New project name** edit box. Next, enter the location where the new project will be saved in the **New project location** edit box. Alternatively, choose the **Browse** button next to this edit box to specify the location. Also, make sure the **Use selected project as template project** check box is selected and then choose the **OK** button; you will notice that the project will be copied and the name of the copied project will be displayed at the top in the **Projects** rollout in bold text. This will now become the active project.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Note

*The **Copy Project** dialog box now saves the user-defined destination folder location.*

You can also customize the project to be copied. To do so, you need to clear the **Use selected project as template project** check box, refer to Figure 2-24. Next, enter the information in this dialog box as discussed above and then choose the **OK** button; the **Select Drawings to Process** dialog box will be displayed, as shown in Figure 2-25.

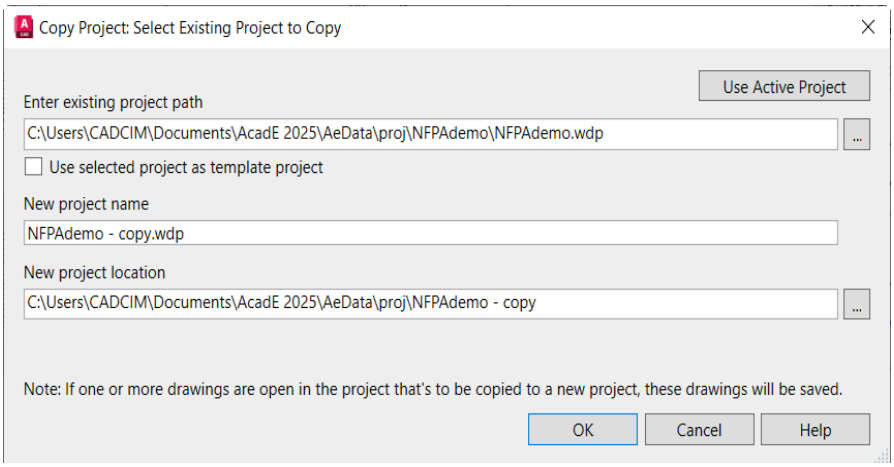


***Figure 2-24** The **Use selected project as template project** check box cleared*

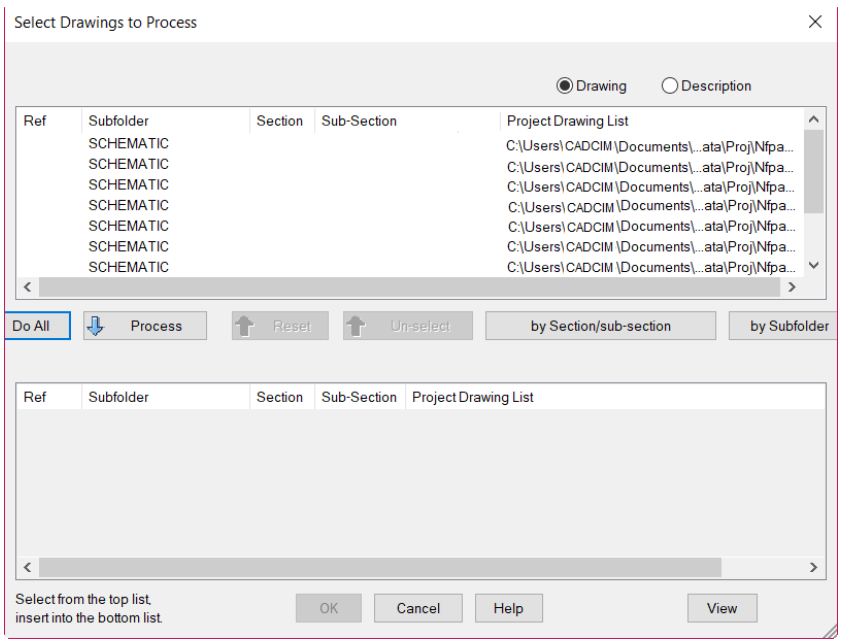


***Figure 2-25** The **Select Drawings to Process** dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Select the drawing files that you want to copy in the project from the top list of this dialog box. Next, choose the **Process** button; the selected drawings will be transferred from the top list of the **Select Drawings to Process** dialog box to the bottom list. The other options in the **Select Drawings to Process** dialog box have already been discussed. Choose the **OK** button from the **Select Drawings to Process** dialog box; the **Copy Project: Enter Base Path for Project Drawings** dialog box will be displayed, refer to Figure 2-26.



Note

*In case, there is no drawing file in the project to be copied, then the **Select Drawings to Process** dialog box will not be displayed and the **AutoCAD Message** message box will be displayed. Choose the **OK** button in this message box; the **Copy Project: Enter Base Path for Project Drawings** dialog box will be displayed.*

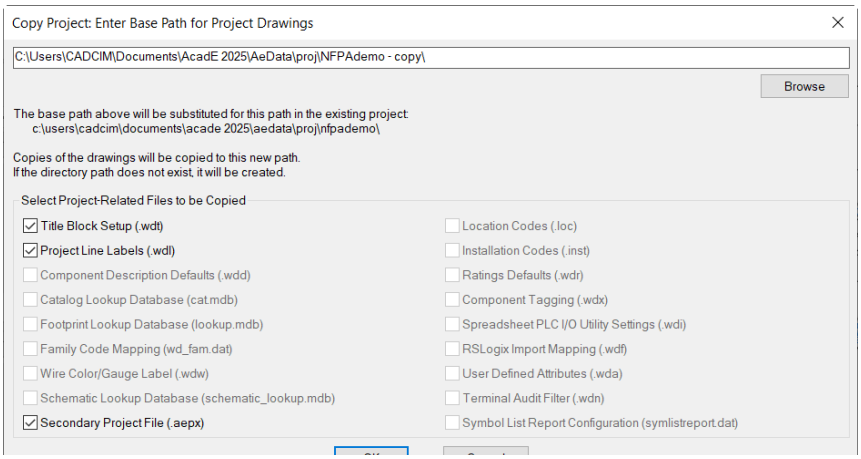


***Figure 2-26** The **Copy Project: Enter Base Path for Project Drawings** dialog box*

In this dialog box, you can specify the project related files that need to be copied to the project. Note that the project related files to be copied will be activated in the **Select Project-Related Files to be Copied** area. You need to select the respective check boxes and then choose the **OK** button; the **Copy Project: Adjust new drawing file names** dialog box will be displayed, as shown in Figure 2-27.

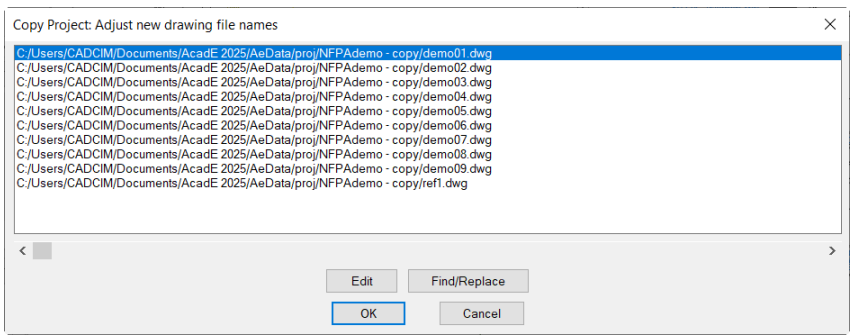


***Figure 2-27** The **Copy Project: Adjust new drawing file names** dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Choose the **Edit** button in this dialog box to edit the file name and path of the selected drawing, if needed. Similarly, choose the **Find/Replace** button to find or replace a drawing file name and its path. Next, choose the **OK** button; the name of the new project will be displayed at the top in the **Projects** rollout in bold text and it will become the active project.

Deleting a Project

Ribbon:	Project > Project Tools> Delete
Menu:	Projects > Project > Delete Project
Command:	AEDELETEPROJECT



The **Delete** tool or the **AEDELETEPROJECT** command is used to delete an existing project and its drawings permanently. To do so, choose the **Delete** tool from the **Project Tools** panel of the **Project** tab; the **Select Existing Project to Delete** dialog box will be displayed, as shown in Figure 2-28.

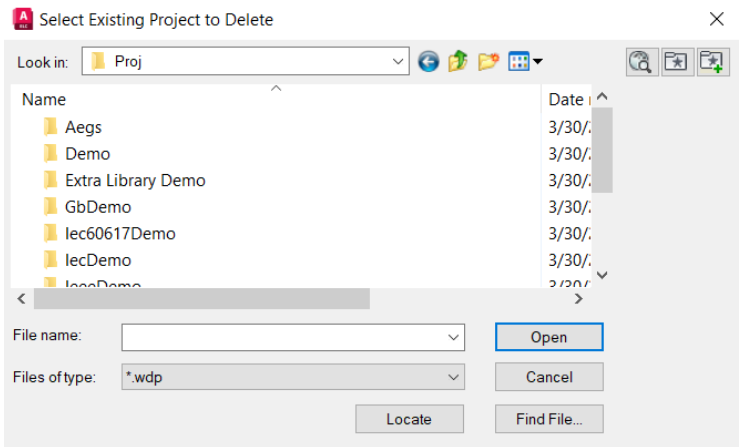


***Figure 2-28** The **Select Existing Project to Delete** dialog box*

Enter the name of the project file to be deleted in the **File name** edit box. Alternatively, select a project name from the list displayed in the **Select Existing Project to Delete** dialog box. Next, double-click on the name of the project folder, if it exists. Select the project definition file(*.wdp*) and choose the **Open** button; the **Project File Delete Utility** dialog box will be displayed, as shown in Figure 2-29. The options in this dialog box are discussed next.

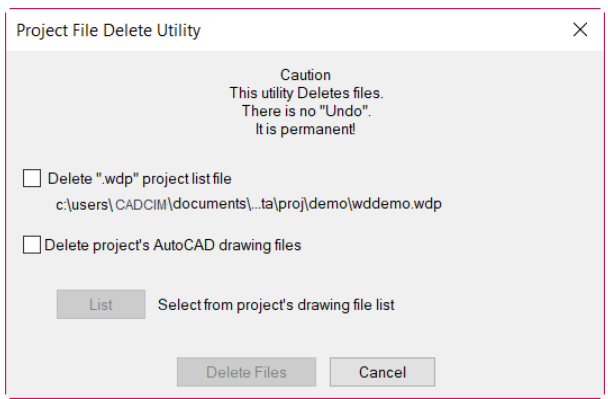


***Figure 2-29** The **Project File Delete Utility** dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Delete ".wdp" project list file

The **Delete ".wdp" project list file** check box is used to permanently delete the selected project file with *.wdp* extension.

Delete project's AutoCAD drawing files

The **Delete project's AutoCAD drawing files** check box is used to delete only the drawing files of a project. Note that this check box will be activated only if the related project consists of drawing files.

List

The **List** button will be activated only if you select the **Delete project's AutoCAD drawing files** check box. Choose the **List** button; the **Select Drawings to Process** dialog box will be displayed, refer to Figure 2-25. Select the drawings to be deleted from the drawing file list of the project. Next, choose the **OK** button from the **Select Drawings to Process** dialog box; the **Project File Delete Utility** dialog box will be displayed again. Choose the **Delete Files** button from the **Project File Delete Utility** dialog box; the selected files will be deleted permanently, and you cannot retrieve them.

Other Options in the PROJECT MANAGER

As discussed earlier, the **PROJECT MANAGER** lists the drawing files associated with each project. You can change the settings of a project by using the **PROJECT MANAGER**, refer to Figure 2-30. The major tools and rollouts in the **PROJECT MANAGER** have already been discussed. The remaining ones are discussed next.

1. Buttons
2. Project selection drop-down list
3. Projects rollout
4. Project Drawing list
5. Details/Preview rollout

Buttons

There are several buttons available in the **PROJECT MANAGER** such as **Refresh**, **Publish/Plot**, and so on, see Figure 2-31. These buttons are discussed next.

Refresh



The **Refresh** button is used to freshen the drawing list present in the **PROJECT MANAGER**.

Project task List



The **Project Task List** button will be activated only if there are pending updates for a drawing file. The **Project Task List** button is used to execute the pending updates on any drawing file modified in the active project. Choose the **Project Task List** button from the **PROJECT MANAGER** or right-click on the active project; a shortcut menu will be displayed. Next, choose the **Task List** option from the shortcut menu; the **Task List** dialog box will be displayed. Select the drawing files that you want to update and choose the **OK** button; the **QSAVE** message box will be displayed. Choose the **OK** button from the **QSAVE** message box; the drawing(s) will be updated. The process of keeping the updates

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

pending at a certain stage in the drawing and executing these updates later using the **Task List** option as well as options in the **Task List** dialog box are discussed in detail in Chapter 12.

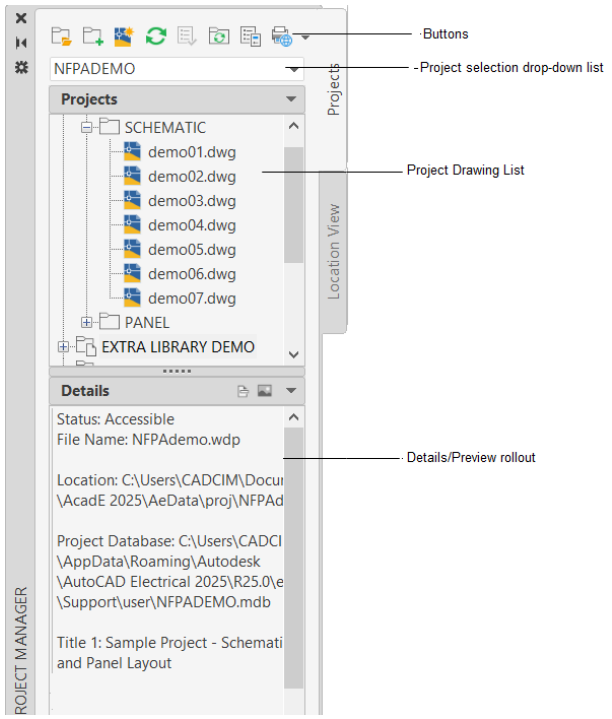


Figure 2-30 *Various components of the* ***PROJECT MANAGER***

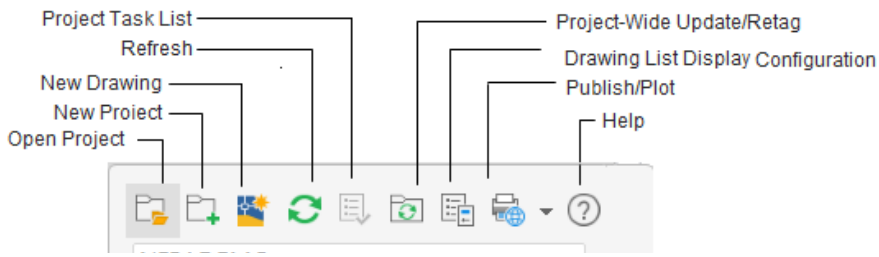


Figure 2-31 *Various buttons in the* ***PROJECT MANAGER***

Project-Wide Update/Retag



The **Project-Wide Update/Retag** button is used to update the related line reference numbers, device tagging, cross-reference text, and signal reference updates on the selected drawing files in an active project. Choose the **Project-Wide Update/Retag** button from the **PROJECT MANAGER**; the **Project-Wide Update or Retag** dialog box will be displayed. Specify the required options in this dialog box and choose the **OK** button; the **Select Drawings to Process** dialog box will be displayed. Now, select the drawings that you want to process and choose the **Process** button. Next, choose the **OK** button; the selected drawings will be updated. The options in the **Project-Wide Update or**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Retag dialog box will be discussed in detail in Chapter 6. You can also use the **(WD_BUMP)** command to invoke the **Project-Wide Update or Retag** dialog box.

Publish / Plot



Choose the **Publish / Plot** button from the **Project Manager**; the **Publish/Plot** drop-down will be displayed, as shown in Figure 2-32. The options in the drop-down of the **Publish / Plot** button are used to plot active drawings, publish the drawings of the active project to web, DWF, PDF, DWFx and zip the active project. Choose the **Plot Project** option from the drop-down to batch plot one or more drawings in the active project. The Plotting of drawings will be discussed in detail in Chapter 12.

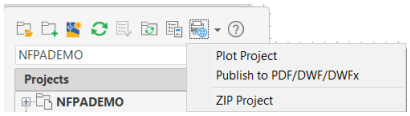


Figure 2-32 *The drop-down displayed on choosing the* ***Publish / Plot*** *button*

Help



When you choose the **Help** button from the **PROJECT MANAGER**, the **AutoCAD Electrical 2025 - Help** window will be displayed. Alternatively, choose **Help** from the **Help** menu to display the **AutoCAD Electrical 2025 - Help** window. This window helps you understand different options, commands, and tools of AutoCAD Electrical.



Note

If you press F1 or use the ***HELP*** *command, it will also display the* ***AutoCAD Electrical 2025 - Help*** *window.*

Project selection Drop-down List

The Project selection drop-down list is available at the top of the **PROJECT MANAGER**, as shown in Figure 2-33, and it consists of names of all open projects and following options:

1. Recent
2. New Project
3. Open Project

You cannot close any open project using the Project selection drop-down list. The other options in this drop-down list are discussed next.

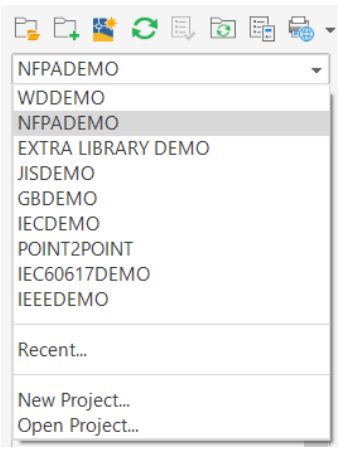


Figure 2-33 *The* ***PROJECT MANAGER*** *displaying the Project selection drop-down list*

Recent

When you select the **Recent** option from the Project selection drop-down list, the **Recent Projects** dialog box will be displayed. In this dialog box, you can view the recently opened projects, view the drawings of the selected project, remove the selected project, and find the drawings from the recent projects list displayed in the **Recent Projects** dialog box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

New Project

The **New Project** option is used to create a new project. To do so, select the **New Project** option from the Project selection drop-down list; the **Create New Project** dialog box will be displayed. The options in this dialog box have already been discussed. Specify the required options and choose the **OK** button; the new project will be created and will appear in bold text on the top of the list in the **Projects** rollout. Also, the newly created project will automatically become an active project.

Open Project

The **Open Project** option is used to open an existing project. To do so, select the **Open Project** option from the Project selection drop-down list; the **Select Project File** dialog box will be displayed. Next, select the required project from this dialog box and choose the **Open** button; the selected project's name will automatically be displayed in the **Projects** rollout in bold text and the project will become an active project.

Projects Rollout

The **Projects** rollout displays a list of all opened projects. You can open as many projects as you want, but only one project can be active at a time. The active project appears in bold text and is always displayed at the top of the list in the **Projects** rollout. When you right-click on the name of the active project or on the project that is in bold text, a shortcut menu will be displayed, as shown in Figure 2-34. The options in the shortcut menu are discussed next.

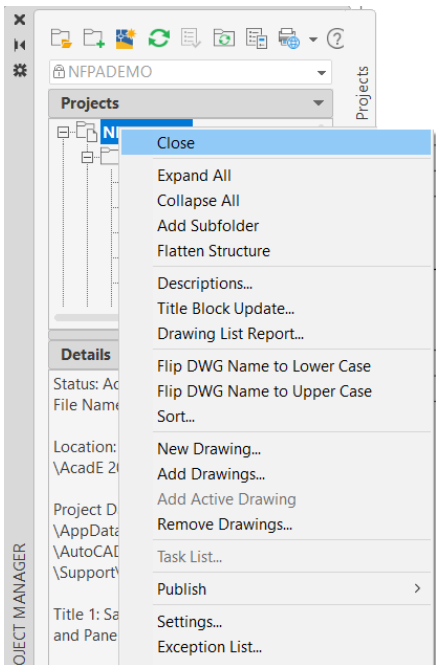


Figure 2-34 *The shortcut menu displayed by right-clicking on the active project in the* ***PROJECT MANAGER***

Close

The **Close** option is used to remove the project which is displayed in the **Projects** rollout.

Expand All

The **Expand All** option is used to expand a project and subfolder(s) in a project, if any.

Collapse All

The **Collapse All** option is used to collapse a project and subfolder(s) in a project, if any.

Add Subfolder

The **Add Subfolder** option is used to add a subfolder to a project.

Flatten Structure

The **Flatten Structure** option is used to remove subfolder(s) from the project to get a flat drawing list in a project.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Descriptions

The **Descriptions** option is used to edit the description of an existing project. It is also used to add description to a new project. To do so, choose the **Descriptions** option from the shortcut menu; the **Project Description** dialog box will be displayed, refer to Figure 2-10.

As the **Project Description** dialog box displays unlimited lines, you can enter the description as per your requirement. The information that you enter in these lines can be re-used in the Component, Bill of Material, and Wire reports that are generated for the project or mapped to title block of the drawing. Select the **in reports** check box from the **Project Description** dialog box to include the project description line information in report headers and title blocks, which will be discussed in detail in the later chapters.

Title Block Update

The **Title Block Update** option is used to update the information of the title block for the entire project drawing set or for the active drawing. To update the title block information, right-click on the active project; a shortcut menu is displayed. Choose the **Title Block Update** option from the shortcut menu; the **Update Title Block** dialog box will be displayed, as shown in Figure 2-35. The options in this dialog box will be discussed in detail in Chapter 12. You can also update a title block by using the **WD_TB** command.

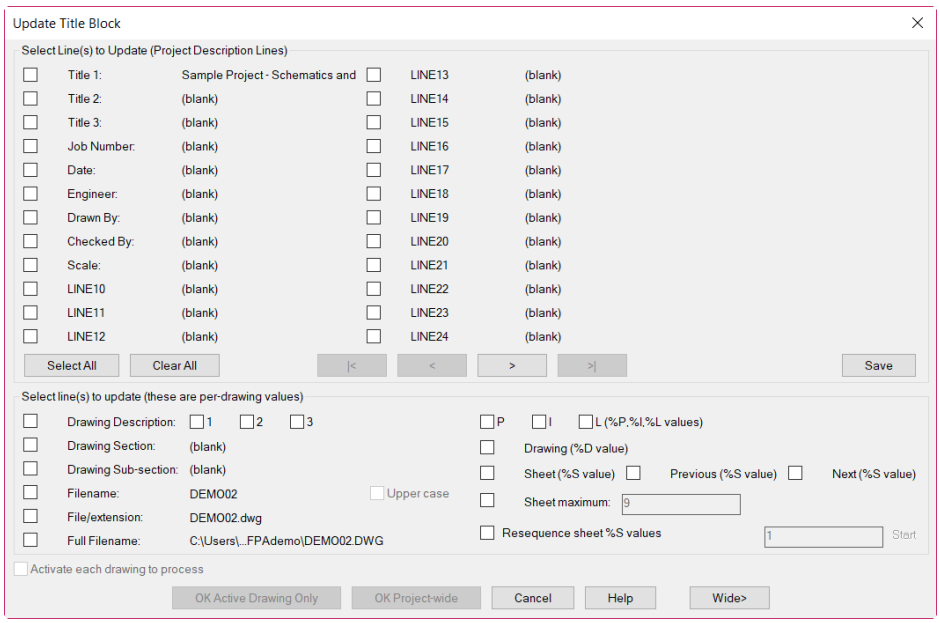


*Figure 2-35 The **Update Title Block** dialog box*

Drawing List Report

The **Drawing List Report** option is used to generate a report that lists the project drawing information of title block such as file names, file date, time, sheet number, drawing descriptions, sections, and so on. To generate a report, choose the **Drawing List Report** option from the shortcut menu or use the **WD_DWGLST_PROJ** command; the **Drawing List Report** dialog box will be displayed. The options in this dialog box are used to extract

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

the new drawing list report, display previous drawing list report, and select format file for report. You will learn more about the generation of reports in Chapter 9.

Task List

The **Task List** option will be activated only if an active project has pending updates on any drawing file which is present within the active project and has been modified. Choose the **Task List** option from the shortcut menu; the **Task List** dialog box will be displayed. The options in the **Task List** dialog box will be discussed in Chapter 12.

Publish

When you right-click on an active project and move the cursor to the **Publish** option of the shortcut menu, a cascading menu will be displayed, as shown in Figure 2-36. This cascading menu consists of various options such as **Plot Project**, **Publish To PDF/DWF/DWFx**, and **Zip Project**. Choose the **Plot Project** option to plot one or more drawings in the active project. The **Publish To PDF/DWF/DWFx**, and **Zip Project** options are used to publish the project to pdf, dwf, dwfx and to create the zipped file of a project, respectively.

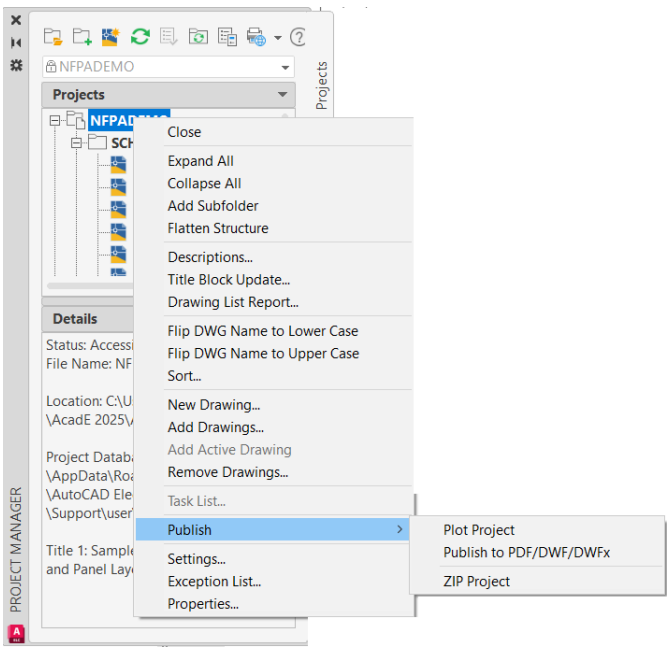


Figure 2-36 *Cascading menu displayed on selecting the* ***Publish*** *option*

Settings

When you choose the **Settings** option from the shortcut menu, the **Current Settings** dialog box is displayed. This dialog box displays settings of a project and information about AutoCAD Electrical environment.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Exception List

When you choose the **Exception List** option from the shortcut menu, the **Properties Exception List** dialog box is displayed, as shown in Figure 2-37. This dialog box displays the list of drawing files that possess the properties different from the project definition file (*.*WDP*). Figure 2-38 shows the **Properties Exception List** dialog box that will be displayed if the settings of all drawing files of a project match with the settings of the project definition file. Also, this dialog box displays that there are no exceptions.

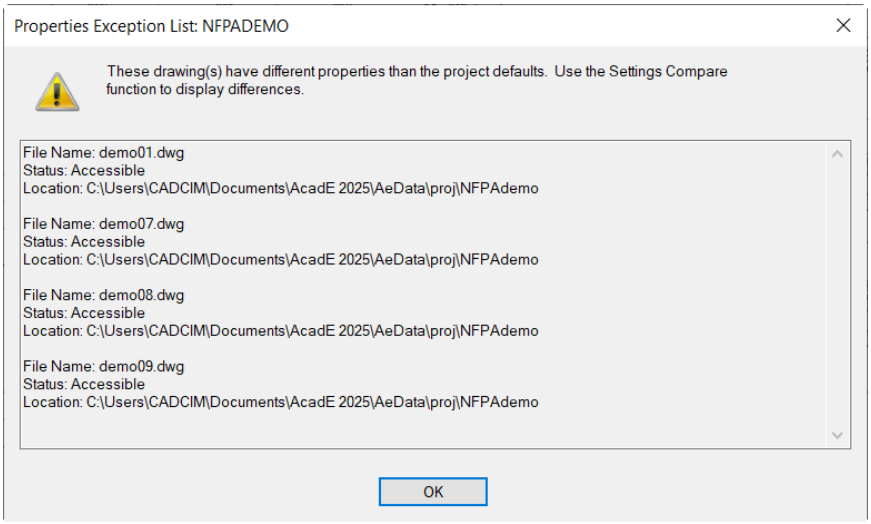


Figure 2-37 *The* ***Properties Exception List*** *dialog box*

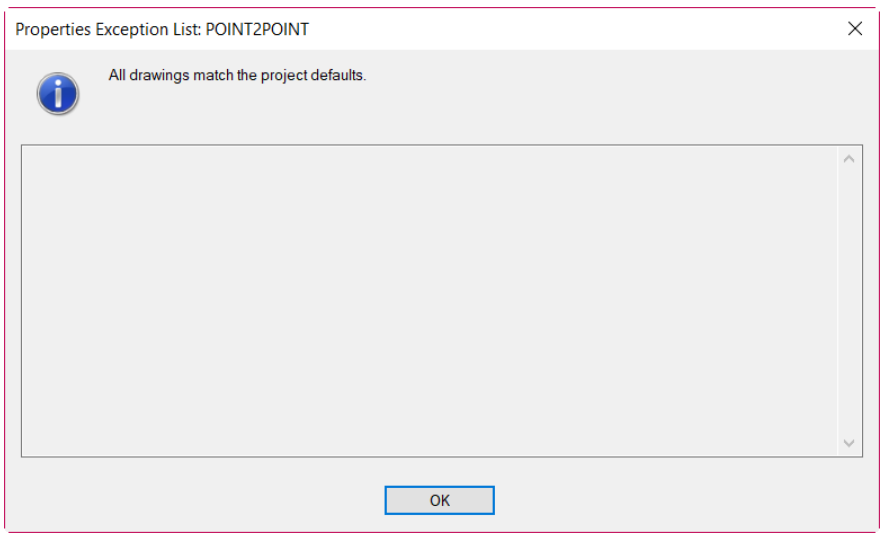


Figure 2-38 *The* ***Properties Exception List*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Note

You can view the difference between drawing properties and project defaults. To do so, choose ***Settings Compare*** *from the* ***Projects*** *menu or choose the* ***Settings Compare*** *button from the* ***Drawing Properties*** *drop-down in the* ***ACE:Main Electrical 2*** *toolbar; the* ***Compare Drawing and Project Settings*** *dialog box will be displayed. This dialog box is used to compare the drawing and project default settings.*

Properties

On choosing the **Properties** option from the shortcut menu, the **Project Properties** dialog box will be displayed, as shown in Figure 2-39. You can use this dialog box to edit and modify the properties for project settings, components, wire numbers, cross-references, styles, and drawing format. The editing of project properties will be discussed in detail in Chapter 12.

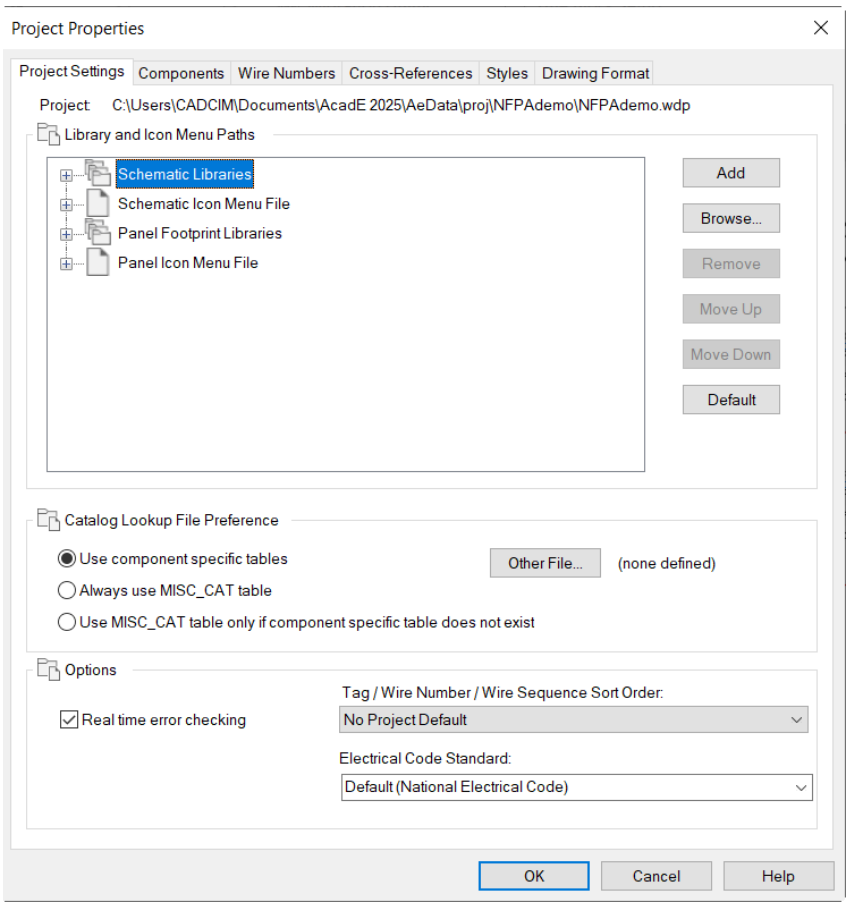


Figure 2-39 *The* ***Project Properties*** *dialog box*



Note

The options discussed above will be displayed if you right-click on an active project in the ***Projects*** *rollout. But if you right-click on an inactive project, a shortcut menu with the* ***Activate*** *option*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

will be displayed, as shown in Figure 2-40. If you choose this option from the shortcut menu, the selected project will become active and will be displayed at the top of the projects list in bold text.

Project Drawing List

The Project Drawing list displays the drawings available in a project. If you double-click on a project name in the **Projects** rollout of the **PROJECT MANAGER**, the drawings associated with that project will be displayed. Right-click on a drawing file name; the editing options will be displayed, as shown in Figure 2-41. Using these options, you can open, close, remove, rename, and replace a drawing file. You can also edit the properties of a drawing by choosing **Drawing Properties** from the shortcut menu, refer to Figure 2-41. The options in this shortcut menu are discussed next.

Note

*You can convert a drawing file into a reference drawing. To do so, right-click on the drawing name and choose **Properties > Drawing Properties** from the shortcut menu; the **Drawing Properties** dialog box will be displayed. Select the **For Reference Only** check box from the **Drawing Settings** tab; the drawing file will be converted into a reference drawing. However, its extension (.dwg) will remain the same.*

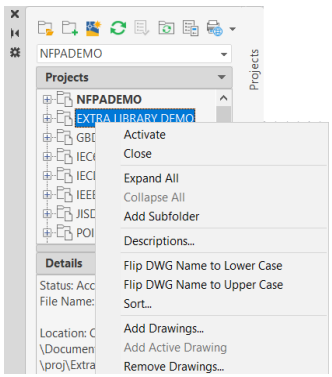


Figure 2-40 *The shortcut menu displaying the **Activate** option*

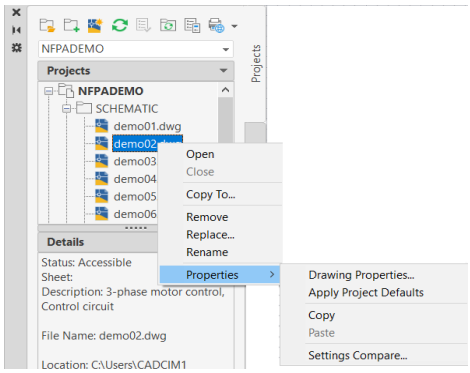


Figure 2-41 *The editing options available in the shortcut menu*

Open

On choosing the **Open** option from the shortcut menu, the selected drawing will open in a new window and its name will appear in bold text in the Project Drawing list. Alternatively, you can select a drawing file name and press ENTER to open the corresponding drawing. You can also open a drawing file by double-clicking on it. The **OPEN** command is also used to open a drawing file.

Close

Choose the **Close** option from the shortcut menu to close the current drawing file. This option will be available only if the drawing file is open. You can also close a drawing file by using the **CLOSE** command.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Copy To

The **Copy To** option is used to copy the selected drawing to the same or another open project. To do so, choose the **Copy To** option from the shortcut menu; the **Copy To** dialog box will be displayed, as shown in Figure 2-42. Next, from the **Save in** drop-down list, select the location where you want to copy the drawings. If you want to change the name of the drawing file, enter the drawing file name in the **File name** edit box. Next, select the project name from the **Project** drop-down list. Choose the **Save** button; the **Apply Project Defaults to Drawing Settings** message box will be displayed. Choose the **Yes** button in the message box to apply the project default values to the newly added drawing's WD_M block definition; the selected drawing(s) will be copied to the specified project. If you want the new drawing to retain its existing settings, choose the **No** button.

Remove

Choose the **Remove** option from the shortcut menu to remove the selected drawing from the current project. This option does not remove the drawings permanently.

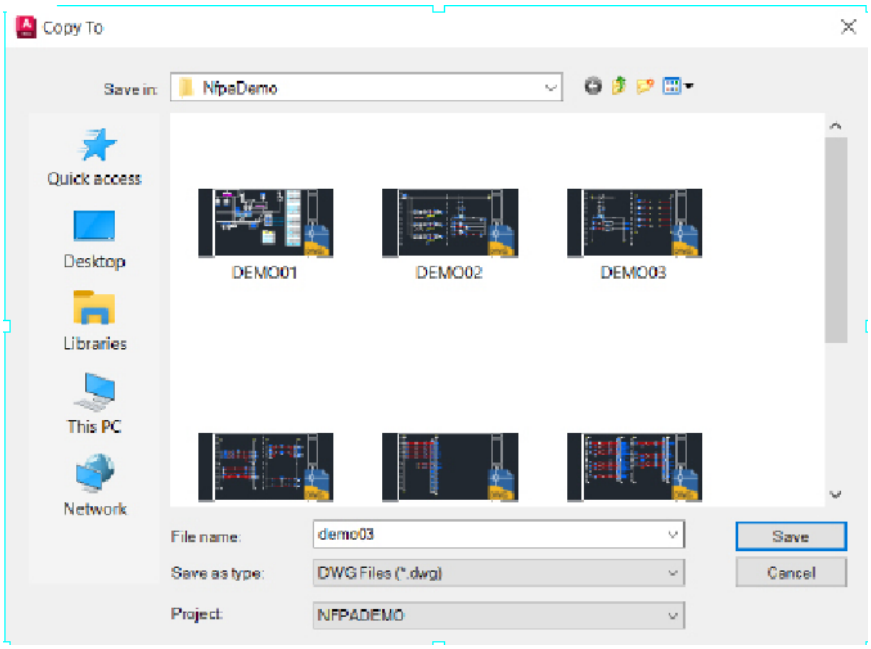


Figure 2-42 *The* ***Copy To*** *dialog box*

Replace

The **Replace** option is used to replace the selected drawing with the desired one. To do so, choose the **Replace** option from the shortcut menu; the **Select Replacement Drawing** dialog box will be displayed, as shown in Figure 2-43. Next, select the drawing file and choose the **Select** button; the **Apply Project Defaults to Drawing Settings** message box will be displayed, refer to Figure 2-13. Choose the **Yes** button to apply the project default values to the newly added drawing's WD_M block definition. If you want the new drawing to retain its existing settings, choose the **No** button; the drawing file will get replaced with the selected drawings. This drawing will be displayed in the Project Drawing list. Also, note

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

that if the drawing is already present in the project, it will display the message that drawing is already present in the project.

Rename

The **Rename** option is used to rename the selected drawing. To do so, choose the **Rename** option from the Project Drawing list; the name of the selected drawing will be replaced by an edit box. Enter a new name in the edit box to rename the drawing.

Drawing Properties

The **Drawing Properties** option is used to change the drawing settings, component tag format, wire number format, cross-reference format, styles, and drawing format. Also, you can edit, assign, and remove the section and sub-section codes of a drawing. To do so, choose **Properties > Drawing Properties** from the shortcut menu; the **Drawing Properties** dialog box will be displayed. In the **Drawing Properties** dialog box, you can assign descriptions to the drawing files. Note that each individual drawing can have its own drawing settings for designing purpose.

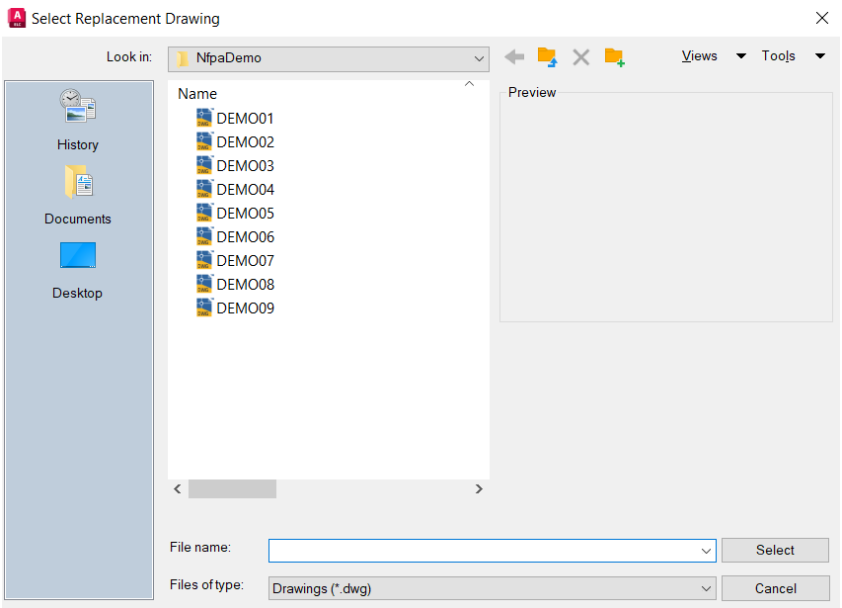


*Figure 2-43 The **Select Replacement Drawing** dialog box*

Apply Project Defaults

Choose the **Apply Project Defaults** option from the shortcut menu to apply the project default settings of the project to the new drawing files, if it was not done while creating the drawing files.

Copy

You can copy drawing settings and options from one drawing to one or more drawings by choosing the **Copy** option from the shortcut menu.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Note

The drawing-specific information that is displayed in the ***Drawing Settings*** *tab of the* ***Drawing Properties*** *dialog box cannot be copied from one drawing to another, refer to Figure 2-12.*

Paste

Choose the **Paste** option from the shortcut menu to apply the copied drawing settings as well as other options from one drawing to other selected drawing(s).

Settings Compare

The **Settings Compare** option is used to compare the drawing with its project settings. To do so, choose the **Settings Compare** option from the shortcut menu; the **Compare Drawing and Project Settings** dialog box will be displayed, as shown in Figure 2-44. This dialog box displays the differences between the drawing settings and their associated default values in the project definition file (*.wdp*). See Figure 2-44, where ***demo01.dwg*** is the drawing name and ***NFPADEMO*** is the project name.

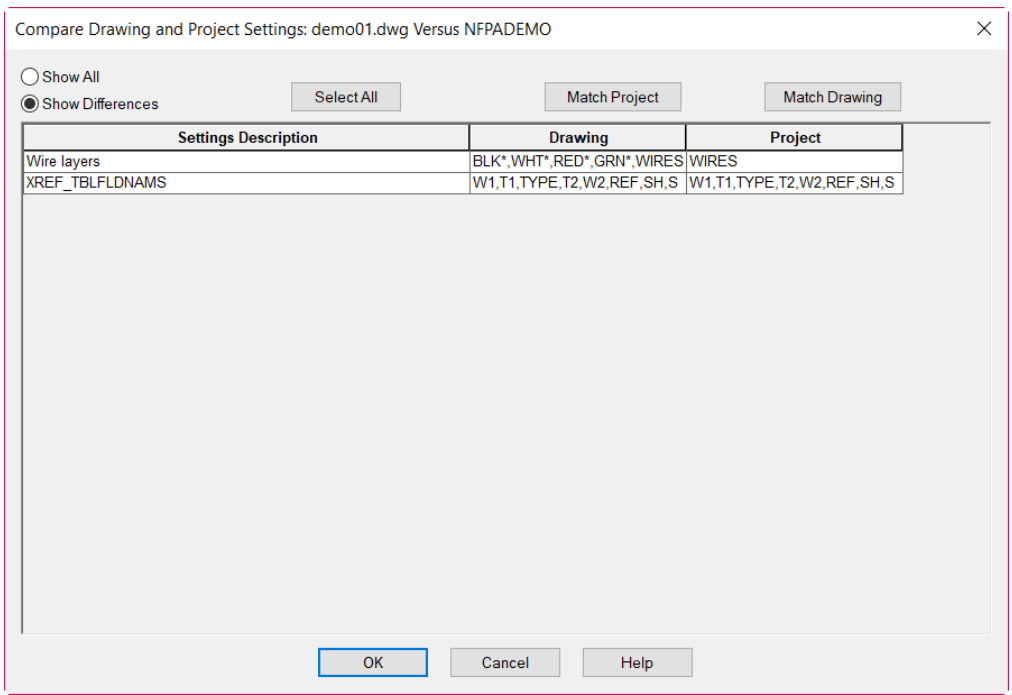


Figure 2-44 *The* ***Compare Drawing and Project Settings*** *dialog box*

Details/Preview Rollout

The **Details/Preview** rollout displays the details of the selected project or drawing as well as the preview of the selected drawing. The **Details** and **Preview** rollouts shown in Figure 2-45 and Figure 2-46 respectively are discussed next.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

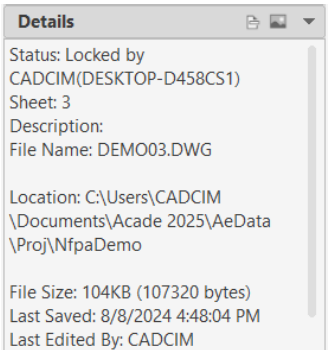


*Figure 2-45 The **PROJECT MANAGER** displaying the **Details** rollout*

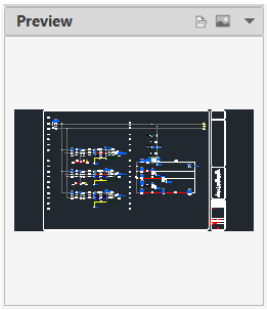


*Figure 2-46 The **PROJECT MANAGER** displaying the **Preview** rollout*

Details

The **Details** button is used to view details of the selected project and drawing. To do so, select a project or drawing from the **Projects** rollout and then choose the **Details** button from the **Details/Preview** rollout in the **PROJECT MANAGER**; details of the selected project or drawing will be displayed in the **Details** rollout. Whenever you select a drawing file, its details get updated and remain visible till you select a new drawing file. The information that will be displayed in the **Details** rollout includes status, description if added, file name, file location, file size, date when the file was last saved, the name of the user who modified the file last, and so on, refer to Figure 2-45.

Note

You can switch/move from one drawing to another by using the up and down arrow keys.

Preview

The **Preview** button is used to display preview of the selected drawing in the **Preview** rollout. To preview a selected drawing, select the drawing from the **Projects** rollout and then choose the **Preview** button from the **Details/Preview** rollout; the image of the selected drawing will be displayed in the **Preview** rollout, refer to Figure 2-46. The image of the selected drawing will be visible till you select another drawing from the **Projects** rollout. You can use up and down ARROW keys to view all drawings of a project.

Note

*As discussed in the previous section, you can use the **PROJECT MANAGER** to preview drawings. When you scroll through the drawings in the **PROJECT MANAGER** using the up and down ARROW keys, the preview or the details of the drawing get displayed in the **PROJECT MANAGER**.*

Location View TAB

The options in the **Location View** tab of the **PROJECT MANAGER** are used to display and filter the components based on the installation and location codes, display details and connections of the components, and so on. Figure 2-47 shows the **PROJECT MANAGER** with the **Location View** tab chosen. The options in this tab are discussed next.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Filter by Installation and Location



This button is used to filter components in the Component Tree as per the installation and location codes. When you click on the down arrow of this button, a flyout is displayed, refer to Figure 2-48. This flyout consists of various combinations of installation and location codes available in the active project as options. You can click on any of these options to view or hide components in the Component Tree based on the requirement.

Refresh Tree for Local Changes



This button is used to refresh the Component Tree, the **Details** pane, and the **Connections** pane based on the changes made in the current session.

Search Field

This field is used to specify the search text such as component name, installation code, or location code.

Go

This button is used to search the text entered in the **Search field**.

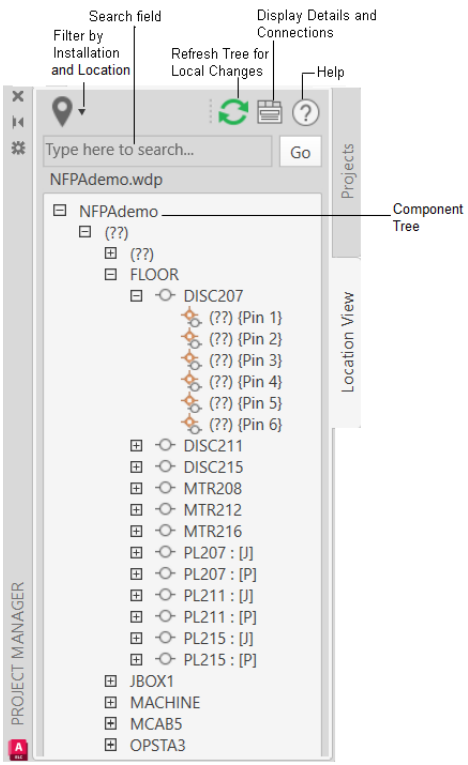


*Figure 2-47 The **PROJECT MANAGER** with the **Location View** tab chosen*

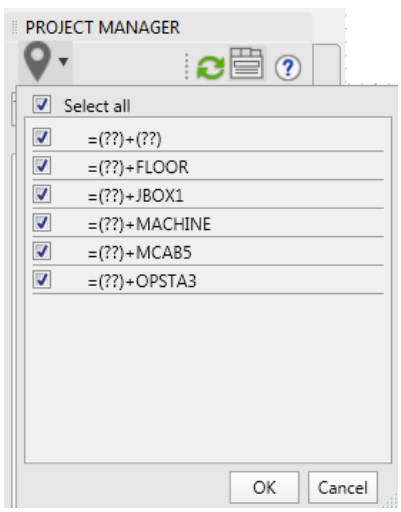


*Figure 2-48 Flyout displayed on choosing the **Filter by Installation and Location** button*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Component Tree

The Component Tree displays a list of all the components available in the active project. It has various nodes. The main node is the active project node. When you expand this node, a node representing installation code will be displayed. When you expand this node, all location code nodes will be listed as separate nodes. You need to expand these nodes to view the components with these installation/location codes as separate nodes. This list also includes PLC modules, connectors, cable markers, and terminals. You need to expand these component nodes in the Component Tree to view pin information of the components, refer to Figure 2-49. When you hover the cursor over a component, the detailed information about the component such as catalog information, assembly code, description is displayed, refer to Figure 2-50.

You will notice that there are different icons on the left of each component. The significance of these icons is explained in Table 2-1.

To expand or collapse all the nodes in the Component Tree, right-click on any of the installation or location nodes and choose the respective option from the shortcut menu displayed, refer to Figure 2-51.

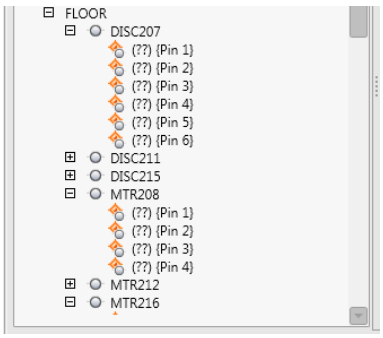


Figure 2-49 *The pin information of the components*

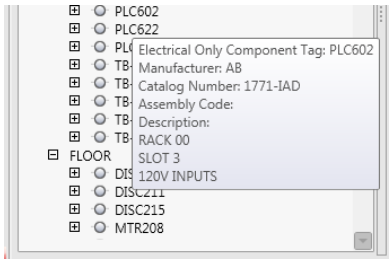


Figure 2-50 *The detailed information of the component*

Table 2-1 *The significance of icons*

	AutoCAD Electrical component not linked to an Inventor component
	Inventor component not linked to an AutoCAD Electrical component
	Linked AutoCAD Electrical and Inventor components with no mismatches in data
	Linked AutoCAD Electrical and Inventor components with some mismatch between the data
	AutoCAD Electrical cable not linked to Inventor cable
	Inventor cable not linked to AutoCAD Electrical cable
	Linked AutoCAD Electrical and Inventor Cable with no mismatches in data
	Linked AutoCAD Electrical Inventor cable with some mismatch between the data

When you right-click on an individual component node, a shortcut menu will be displayed, refer to Figure 2-52. Choose the **Surf** option; the **Surf** dialog box will be displayed. In this dialog box,

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

related references of the component are displayed. The options in this dialog box are discussed in detail in Chapter 6.

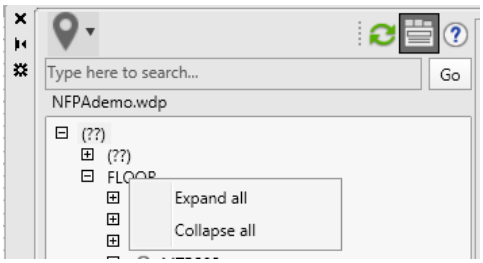


Figure 2-51 *The shortcut menu displayed on right-clicking on an installation/location node*



Figure 2-52 *The shortcut menu displayed on right-clicking on a component node*

Display Details and Connections



This button is used to expand the **PROJECT MANAGER** with two additional tabs: **Details** and **Connections**. These two tabs are discussed next.

Details Tab

When you choose this tab, the **Details** pane is displayed. Choose the installation or location node in the **PROJECT MANAGER**; a list of components in this node will be displayed in a grid in the **Details** pane, refer to Figure 2-53. You need to scroll horizontally in the pane or increase the size of the pane by stretching it horizontally to view all the columns in the grid. If you select an individual component node in the **PROJECT MANAGER**, the tabular information of only that component will be displayed in a grid.

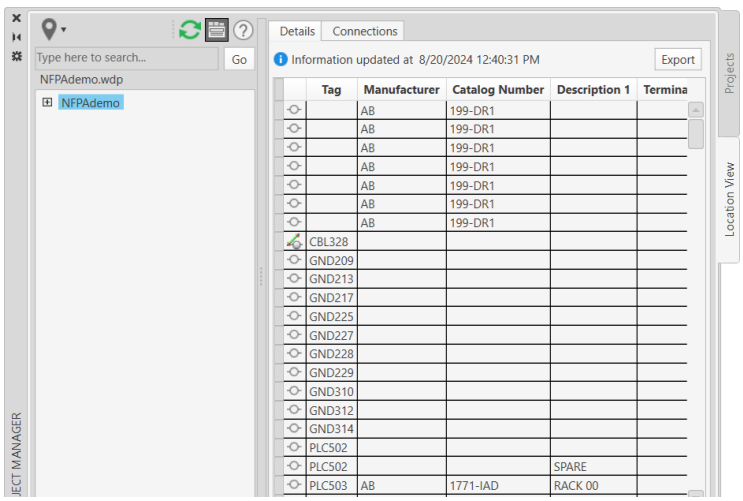


Figure 2-53 *The **Details** pane with tabular information*

If you right-click on the column name of the grid, a shortcut menu will be displayed, refer to Figure 2-54. The options in this shortcut menu are used to add or remove columns from the

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Details pane and to restore default number of columns. Similarly, if you right-click on the cell(s) or row(s) in the grid, a shortcut menu will be displayed, refer to Figure 2-55. This shortcut menu consists of two options: **Copy** and **Surf**. The **Surf** option is already discussed in the Component Tree. The **Copy** option will be activated only when you select cell(s) or row(s) in the grid. Using this option, you can copy the content of the selected cell(s) or row(s).

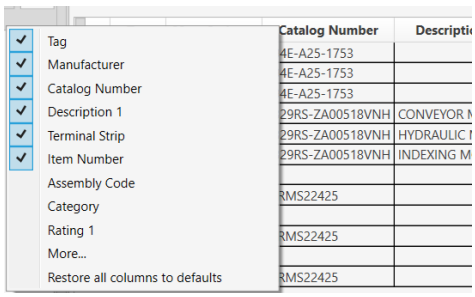


Figure 2-54 The shortcut menu displayed on right-clicking on the column name

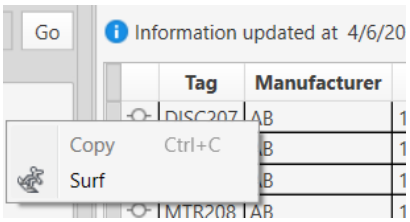


Figure 2-55 The shortcut menu displayed on right-clicking on the cell(s) or row(s) in the grid

The **Export** button located at the top right corner of the **Details** pane is used to export the data in the grid to .xls or .csv file. Note that in the export process only the data that is currently displayed in the grid will be exported.

Connections Tab

When you choose this tab, the **Connections** pane is displayed. Choose the installation or location node in the **PROJECT MANAGER**; information about wiring of the components in this node will be displayed in a grid in the **Connections** pane, refer to Figure 2-56. You need to scroll horizontally in the pane or increase the size of the pane by stretching it horizontally to view all the columns in the grid. If you select an individual component node in the **PROJECT MANAGER**, the information regarding that component will only be displayed in the grid.

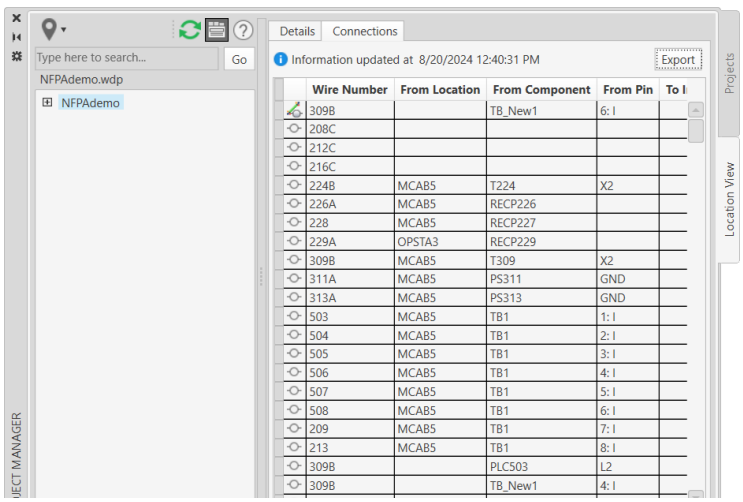


*Figure 2-56 The **Connections** pane with tabular information*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

If you right-click on the column name in the grid, a shortcut menu will be displayed, refer to Figure 2-57. The options in this shortcut menu are used to add or remove columns from the **Connections** pane and to restore default number of columns. Similarly, if you right-click on the cell(s) or row(s) of the grid, a shortcut menu will be displayed, refer to Figure 2-58. This shortcut menu consists of two options: **Copy** and **Surf**. The **Surf** option is already discussed in the Component Tree. The **Copy** option will be activated only when you select cell(s) or row(s) in the grid. Using this option, you can copy the content from the selected cell(s) or row(s).

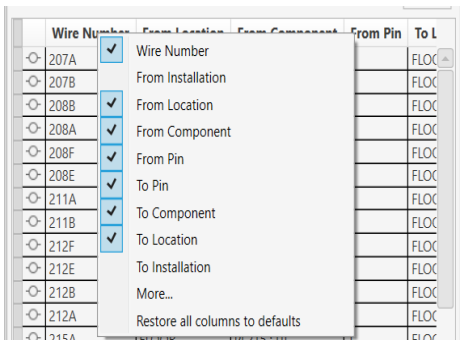


Figure 2-57 *The shortcut menu displayed on right-clicking on the column names*

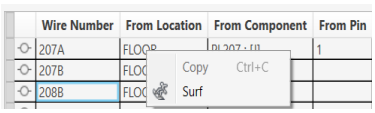


Figure 2-58 *The shortcut menu displayed on right-clicking on the cell(s) or row(s) of the grid*

The **Export** button located at the top right corner of the **Connections** pane is used to export the data in the grid to *.xls* or *.csv* file. Note that in the export process only the data that is currently displayed in the grid will be exported.



Note

1. If the project you are working is an electromechanical project and you are linking it with Autodesk Inventor, some additional buttons will be available in the ***PROJECT MANAGER*** *such as* ***Filter the view by link status***, ***Refresh the data from Inventor***, *and so on. Also, some additional shortcut menu options will be available for the grid.*

2. You can link AutoCAD Electrical component to the Inventor part or vice-versa. To do so, right-click on Inventor part or AutoCAD Electrical part and choose ***Assign to Existing in Component Tree*** *from the shortcut menu displayed. The selected components will be linked with some differences. You need to resolve these differences using the* ***Details*** *and* ***Connections*** *pane in the* ***Location View*** *tab.*

3. To insert connector from the ***Location View*** *tab in the Inventor assembly of the electromechanical project, right-click on a node or a connector and then choose* ***Insert Connector (From List)*** *from the shortcut menu displayed.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

TUTORIALS

The tutorials given next are available in video format.

Tutorial 1

In this tutorial, you will create a new project and add project description to it, refer to Figure 2-59. You will also create a new drawing in the project. **(Expected time: 10 min)**

The following steps are required to complete this tutorial:

a. Create a new project.
b. Open the CADCIM project.
c. Create a new drawing.

Creating a New Project

1. Start AutoCAD Electrical 2025 by double-clicking on the shortcut icon of AutoCAD Electrical 2025 on the desktop of your computer.

2. In the AutoCAD Electrical 2025 interface, choose the **New** button in the **Start** tab. The **PROJECT MANAGER** is displayed by default on the left of the screen. If it is not displayed, choose the **Manager** tool from the **Project Tools** panel of the **Project** tab. Alternatively, choose the **Project Manager** tool from the **ACE:Main Electrical 2** toolbar to display the **PROJECT MANAGER**.

3. In the **PROJECT MANAGER**, make sure the **Projects** tab is chosen. Next, choose the **New Project** button; the **Create New Project** dialog box is displayed, as shown in Figure 2-60.

4. Enter **CADCIM** in the **Name** edit box.

5. Choose the **Browse** button; the **Browse For Folder** dialog box is displayed. By default, the **Proj** folder is selected in this dialog box. Choose the **OK** button; the location of the project is automatically displayed in the **Location** edit box.

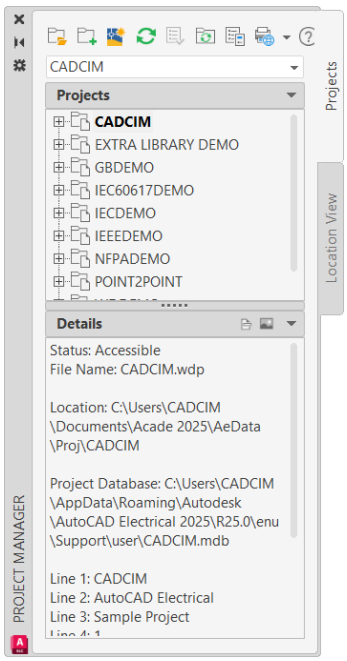


Figure 2-59 *The* ***PROJECT MANAGER*** *displaying the* ***CADCIM*** *project and its description*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

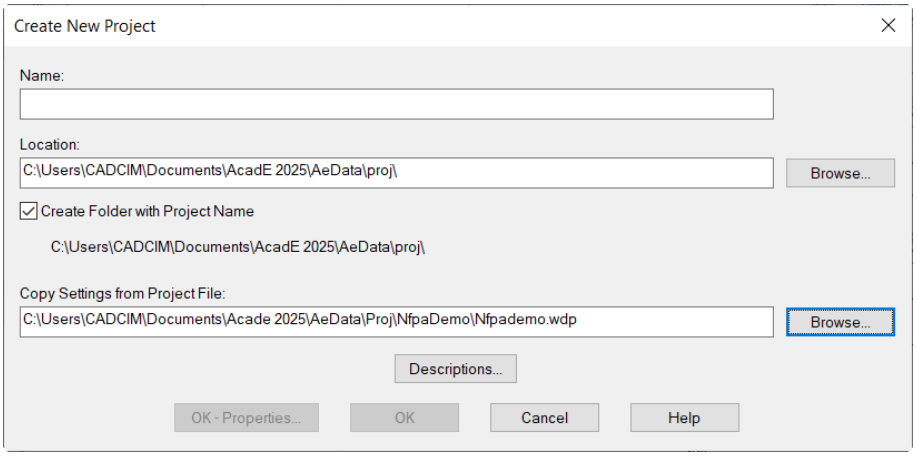


Figure 2-60 *The* ***Create New Project*** *dialog box*

6. Select the **Create Folder with Project Name** check box, if not selected.

7. By default, the name and path of the existing project is displayed in the **Copy Settings from Project File** edit box as *C:\Users\User Name\Documents\Acade 2025\AeData\proj\NFPAdemo\NFPAdemo.wdp*. If it is not displayed, choose the **Browse** button; the **Select Project File** dialog box is displayed. Select the ***NFPADemo*** folder and then select the **NFPAdemo.wdp** project file.

   Next, choose the **Open** button; the path and name of the existing project is displayed in the **Copy Settings from Project File** edit box.

8. Choose the **Descriptions** button from the **Create New Project** dialog box; the **Project Description** dialog box is displayed.

9. Enter the following information in the **Project Description** dialog box, as shown in Figure 2-61.

   LINE1= CADCIM
   LINE2= AutoCAD Electrical
   LINE3= Sample Project
   LINE4= 1
   LINE5= 25/06/2024
   LINE6= Sham
   LINE7= John
   LINE8= Crystal
   LINE9= 1.00

   Next, select the **in reports** check box corresponding to each of these lines to include this description in the reports, refer to Figure 2-61.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

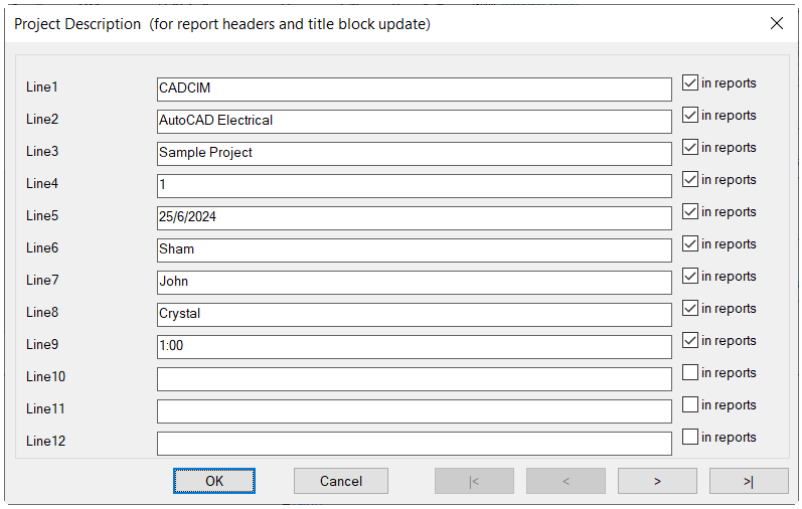


Figure 2-61 *The* ***Project Description*** *dialog box*

Note

The information that you enter in the description lines of the ***Project Description*** *dialog box will be displayed at a particular location in the title block of the drawing files of the* ***CADCIM*** *project, as shown in Figure 2-62. The title blocks will be discussed in detail in Chapter 12.*

10. Choose the **OK** button from the **Project Description** dialog box to save the changes made in this dialog box.

11. Choose the **OK** button from the **Create New Project** dialog box. You will notice that the **CADCIM** project appears in bold text at the top of the project list displayed in the **PROJECT MANAGER**, refer to Figure 2-63.

12. Next, select the **CADCIM** project from the **PROJECT MANAGER**.

13. Choose the **Details** button from the **Details/Preview** rollout. You will notice that the project description/information entered in the first nine lines of the **Project Description** dialog box is displayed in the **Details/Preview** area, as shown in Figure 2-63.

Note

1. It is recommended to save all the tutorials in the forthcoming chapters in the ***CADCIM*** *project.*

2. If the ***CADCIM*** *project is not displayed in the* ***Projects*** *rollout of the* ***PROJECT MANAGER****, select the* ***Open Project*** *option from the Project selection drop-down list; the* ***Select Project File*** *dialog box is displayed. Select the* ***Proj*** *folder from the* ***Look in*** *drop-down list and then double-click on the* ***CADCIM*** *folder name. Next, select the* ***CADCIM*** *file and choose the* ***Open*** *button; the* ***CADCIM*** *project is displayed in the* ***Projects*** *rollout and it becomes the active project.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

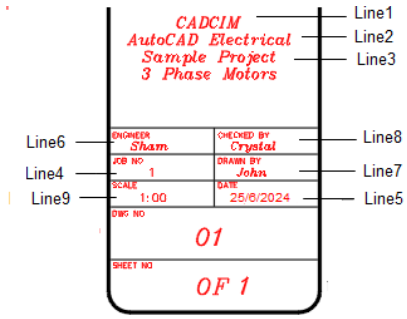


Figure 2-62 *The title block showing the description entered in the* ***Project Description*** *dialog box*

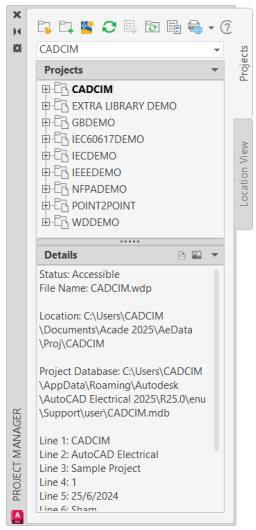


Figure 2-63 *The* ***PROJECT MANAGER*** *displaying the* ***CADCIM*** *project and its description*

Creating a New Drawing

1. In the **PROJECT MANAGER**, choose the **New Drawing** button; the **Create New Drawing** dialog box is displayed, as shown in Figure 2-64.

2. Enter **C02_tut01** in the **Name** edit box. Next, choose the **Browse** button; the **Select template** dialog box is displayed. In this dialog box, select **ACAD_ELECTRICAL.dwt** from the list displayed and then choose the **Open** button; the name and path of the template is displayed in the **Template** edit box.

3. Clear the **For Reference Only** check box, if it is selected. Next, choose the **Browse** button available on the right of the **Location** edit box; the **Browse For Folder** dialog box is displayed. By default, the **CADCIM** project is selected. Choose the **OK** button; the location of the drawing is automatically displayed in the **Location** edit box.

4. Enter **3 Phase Motors** in the **Description 1** edit box.

5. Enter **Motor Control Circuit** in the **Description 2** edit box and enter **01** in the **Drawing** edit box of the **Sheet Values** area.

6. Choose the **OK** button from the **Create New Drawing** dialog box; you will notice that the drawing that you created gets added to the active project. To view the drawing that you created, double-click on **CADCIM** in the **PROJECT MANAGER**; the drawing list is displayed and the drawing *C02_tut01.dwg* appears in bold text in the **PROJECT MANAGER**. In this case, the active project is **CADCIM** and therefore, the drawing gets added to this project. Click on the drawing; the details and the preview of the drawing are displayed in the **PROJECT MANAGER**, as shown in Figures 2-65 and 2-66.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

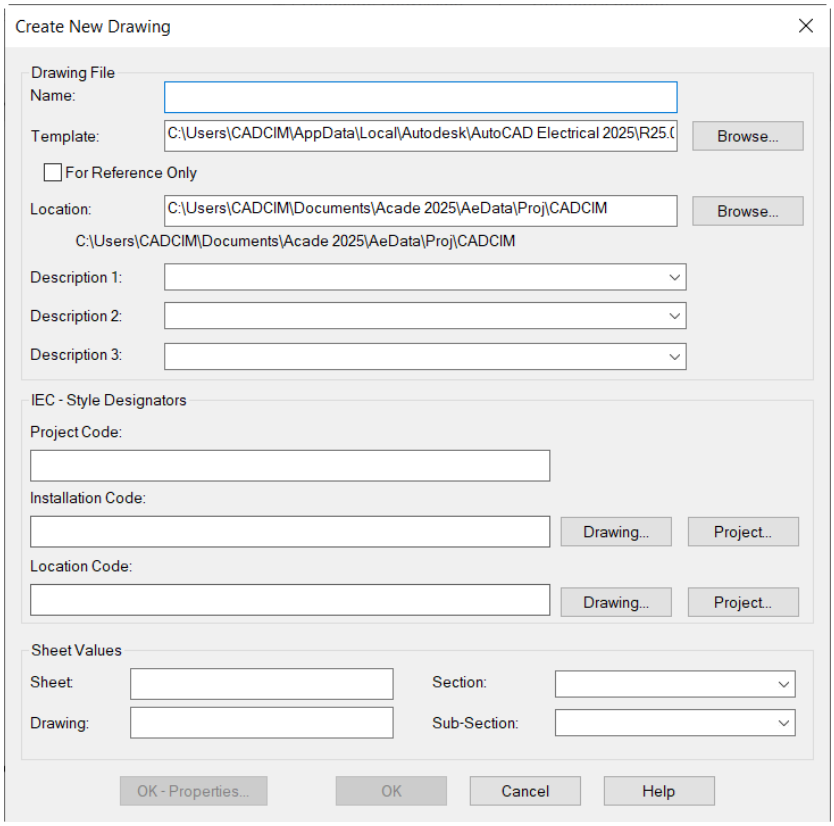


Figure 2-64 *The* ***Create New Drawing*** *dialog box*

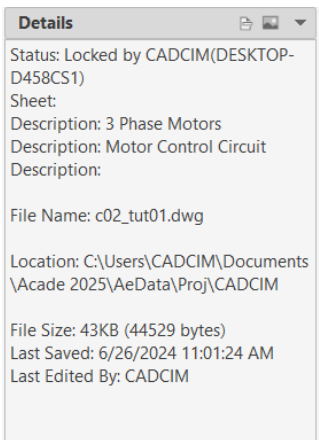


Figure 2-65 *The* ***Details*** *rollout displaying the description of C02_tut01.dwg*

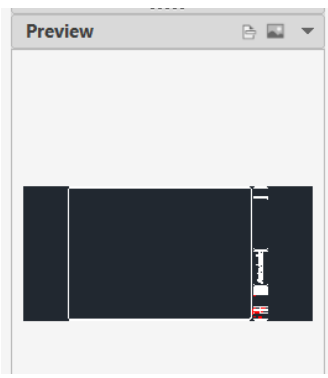


Figure 2-66 *The* ***Preview*** *area displaying the preview of C02_tut01.dwg*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Tutorial 2

In this tutorial, you will add and remove drawings from the **CADCIM** project and replace the drawings of the **POINT2POINT** project with the **CADCIM** project that you created in Tutorial 1 of this chapter. **(Expected time: 15 min)**

The following steps are required to complete this tutorial:

a. Add drawings from the existing project.
b. Remove the drawing.
c. Replace the drawing.

Adding Drawings from the Existing Project

1. Right-click on the **CADCIM** project; a shortcut menu is displayed. Choose the **Add Drawings** option from the shortcut menu; the **Select Files to Add** dialog box is displayed.

2. In this dialog box, select the *c:\Users\User Name\Documents\Acade 2025\AeData\proj* folder from the **Look in** drop-down list; a list of all the projects saved in this folder is displayed. Next, double-click on the **NfpaDemo** folder from the **Select Files to Add** dialog box; the drawings present in this folder are displayed.

3. Press SHIFT/CTRL and select the *DEMO01*, *DEMO02*, *DEMO03*, *DEMO04*, and *DEMO05* drawings. Next, choose the **Add** button; the **Apply Project Defaults to Drawing Settings** message box is displayed.

4. Choose the **Yes** button from this message box; the selected drawings are added to the **CADCIM** project, as shown in Figure 2-67.

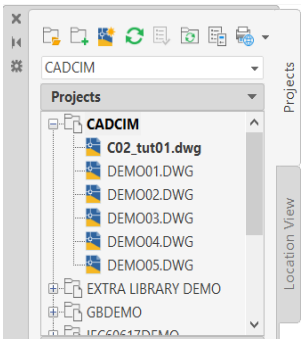


Figure 2-67 *The* ***PROJECT MANAGER*** *displaying the* ***CADCIM*** *project with the drawings added to it*

Removing the Drawing

1. Right-click on the *DEMO01.dwg* drawing and then choose the **Remove** option from the shortcut menu displayed; the **PROJECT - Manager Remove Files** message box is displayed. Choose the **Yes** button in this message box; the *DEMO01.dwg* drawing is removed from the drawing list, see Figure 2-68.

Replacing the Drawing

1. To replace the *DEMO02.*dwg drawing of the **CADCIM** project with the *Connector.dwg* of the **POINT2POINT** project, right-click on the *DEMO02.dwg* drawing; a shortcut menu is displayed. Choose the **Replace** option from the shortcut menu; the **Select Replacement Drawing** dialog box is displayed.

2. Select the **Proj** folder from the **Look in** drop-down list. Next, double-click on the **Point2Point** folder; the *Connector.dwg* drawing is displayed in the **Select Replacement Drawing** dialog box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

3. Select the *Connector.dwg* drawing from this dialog box and choose the **Select** button; the **Apply Project Defaults to Drawing Settings** message box is displayed. Next, choose the **Yes** button; the *DEMO02.dwg* is replaced with the *Connector.dwg* drawing in the **CADCIM** project, as shown in Figure 2-69.

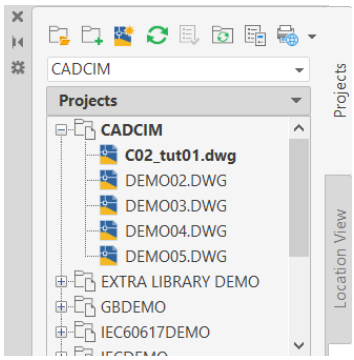


Figure 2-68 *The* ***PROJECT MANAGER*** *displaying the* ***CADCIM*** *project after removing the DEMO01.dwg drawing*

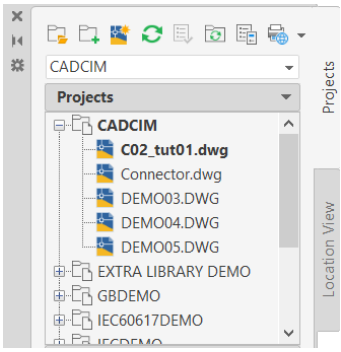


Figure 2-69 *The* ***PROJECT MANAGER*** *displaying the* ***CADCIM*** *project with the DEMO02.dwg drawing replaced by the Connector.dwg drawing*

Tutorial 3

In this tutorial, you will create a subfolder within a **CADCIM** project and configure the drawing list display for this project. **(Expected time: 15 min)**

The following steps are required to complete this tutorial:

a. Create a subfolder.
b. Configure the drawing list display.

Creating a Subfolder

In this section, you will create a subfolder in the **CADCIM** project that is created in Tutorial 1.

1. Make sure that the **CADCIM** project is activated in the **PROJECT MANAGER**. Next, right-click on it; a shortcut menu is displayed.

2. Choose the **Add Subfolder** option from the shortcut menu; a subfolder with the name **NEW FOLDER** is created within the **CADCIM** project. Rename it as *TUTORIALS*.

3. Drag and drop the *C02_tut01.dwg* drawing created in Tutorial 1 on the *TUTORIALS* subfolder; the *C02_tut01.dwg* drawing is moved to the *TUTORIALS* subfolder, refer to Figure 2-70.

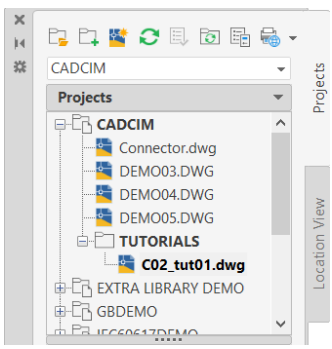


Figure 2-70 *The C02_tut01.dwg moved to the TUTORIALS subfolder*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Note

1. In AutoCAD Electrical, all projects and its subfolders names are displayed in uppercase.
*2. You can collapse the **CADCIM** project by right-clicking on its name and choosing the **Collapse All** option from the shortcut menu displayed.*

Configuring the Drawing List Display

1. Choose the **Drawing List Display Configuration** button from the **PROJECT MANAGER**; the **Drawing List Display Configuration** dialog box is displayed. You will notice that only the **File Name** is displayed in the **Current Display Order** area.

2. Select **Drawing Number(%D)** from the **Display Options** area and then choose the **>>** button; **Drawing Number(%D)** is moved to the **Current Display Order** area. Similarly, select **Drawing Description 1** from the **Display Options** area and choose the **>>** button; **Drawing Description 1** is moved to the **Current Display Order** area. Next, choose the **OK** button to close the **Drawing List Display Configuration** dialog box.

   You will notice that drawing numbers are added to the first four drawings in the **CADCIM** project and description is added to the *C02_tut01.dwg*.

Tutorial 4

In this tutorial, you will copy an existing project with the name **Copy_Point2Point** and add three drawings into it. Next, you will find the list of drawings that have different settings. You will also compare and match the settings of one of the added drawings with the copied project.

(Expected time: 15 min)

The following steps are required to complete this tutorial:

a. Copy the **Point2Point** project.
b. Add the drawings.
c. Find the Exception list and compare the settings.

Copying the Point2Point Project

1. Choose the **Copy** tool from the **Project Tools** panel of the **Project** tab; the **Copy Project: Select Existing Project to Copy** dialog box is displayed, as shown in Figure 2-71.

2. In this dialog box, choose the Browse button located next to the **Enter existing project path** edit box; the **Open** dialog box is displayed. In this dialog box, select the *c:\Users\User Name\Documents\Acade 2025\AeData\proj* folder and then double-click on the **Point2Point** folder. Next, select the **Point2Point.wdp** file, refer to Figure 2-72. Now, choose the **Open** button; path of the *Point2Point.wdp* file is displayed in the **Enter existing project path** edit box.

3. Make sure the **Use Selected project as template project** check box is selected. Now, enter **Copy_Point2Point** in the **New project name** edit box. You will notice that the location of the project to be copied is updated in the **New project location** edit box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

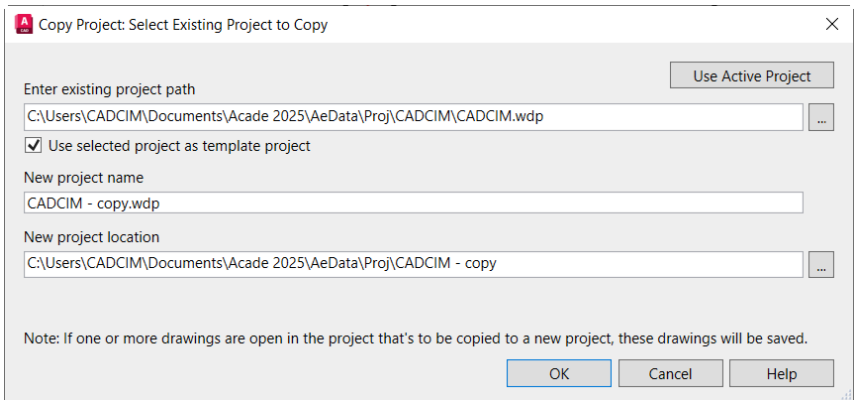


Figure 2-71 *The* ***Copy Project Select Existing Project to Copy*** *dialog box*

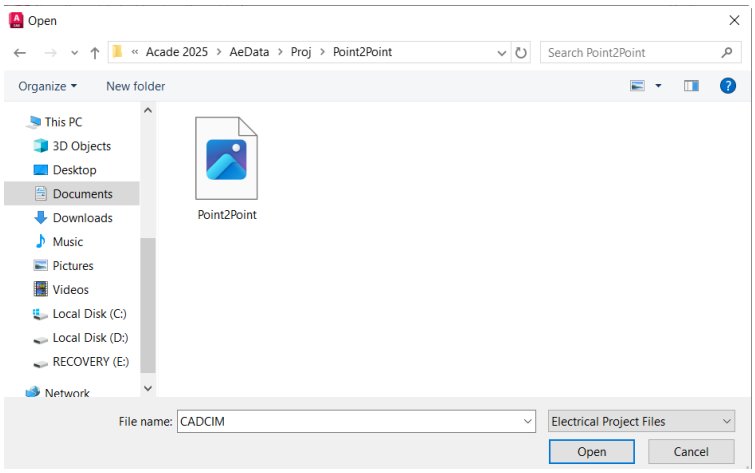


Figure 2-72 *The* ***Open*** *dialog box*

4. Choose the **OK** button in the **Copy Project: Select Existing Project to Copy** dialog box; the **Copy_Point2Point** project is added to the **Projects** rollout of the **PROJECT MANAGER**. Also, the drawings in the **Copy_Point2Point** project get saved in its project folder and this project becomes an active project.

Adding the Drawings

1. Right-click on the **Copy_Point2Point** project in the **PROJECT MANAGER**; a shortcut menu is displayed. Choose the **Add Drawings** option from the shortcut menu; the **Select Files to Add** dialog box is displayed.

2. In this dialog box, choose the **Up one level** button; the **Proj** folder is displayed in the **Look in** drop-down list. Next, double-click on the **NfpaDemo** folder; the drawings present in this folder are displayed. Press SHIFT/CTRL and select the *DEMO07, DEMO08,* and *DEMO09* drawings.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

3. Choose the **Add** button; the **Apply Project Defaults to Drawing Settings** message box is displayed.

4. Choose the **No** button from this message box; the selected drawings are added to the **Copy_Point2Point** project, as shown in Figure 2-73. If the drawings are not displayed in the **Copy_Point2Point** project, choose **Refresh** from the **PROJECT MANAGER**.

Finding the Exception List and Comparing the Settings

1. Right-click on the **Copy_Point2Point** project and choose **Exception List** from the shortcut menu displayed; the **Properties Exception List: COPY_POINT2POINT** dialog box is displayed, as shown in Figure 2-74. This dialog box lists the drawings which have different properties than the project default properties. Next, choose the **OK** button in this dialog box.

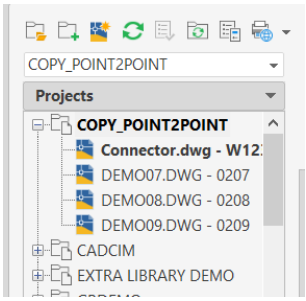


Figure 2-73 *The* ***Copy_Point2Point*** *project with the drawings added to it*

2. Select the **Connector.dwg** from **Copy_Point2Point** in the **Projects** rollout and right-click; a shortcut menu is displayed. Choose **Properties** from the shortcut menu; a cascading menu is displayed. Choose **Settings Compare** from the cascading menu; the **Compare Drawing and Project Settings: Connector.dwg Versus COPY_POINT2POINT** dialog box is displayed, as shown in Figure 2-75.

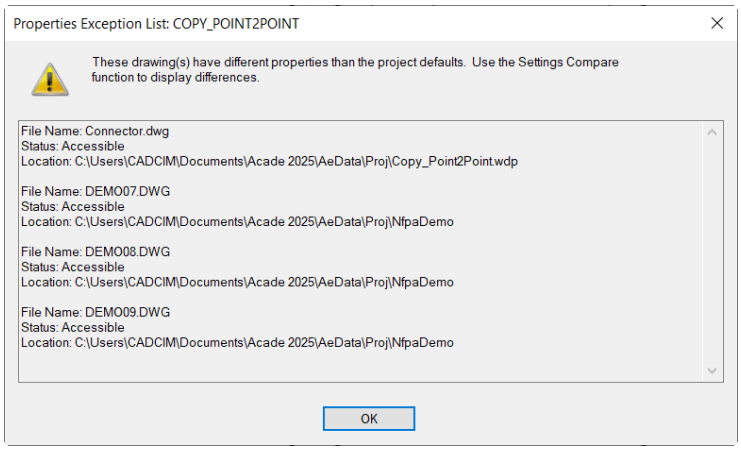


Figure 2-74 *The* ***Properties Exception List: COPY_POINT2POINT*** *dialog box*

You will notice that this dialog box displays the number of drawing settings that are different from its project. First entry in this dialog box indicates that component tags of the *Connector.dwg* are reference based and component tags of the **Copy_Point2Point** project are sequential in order.

3. Select the first entry and choose the **Match Drawing** button from the **Compare Drawing and Project Settings: Connector Versus COPY_POINT2POINT** dialog box. Next, choose **OK**; the component tag settings of the project are matched with the *Connector.dwg* file. To confirm this, repeat step 2. You will notice that component tag settings are not displayed any more in the list.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

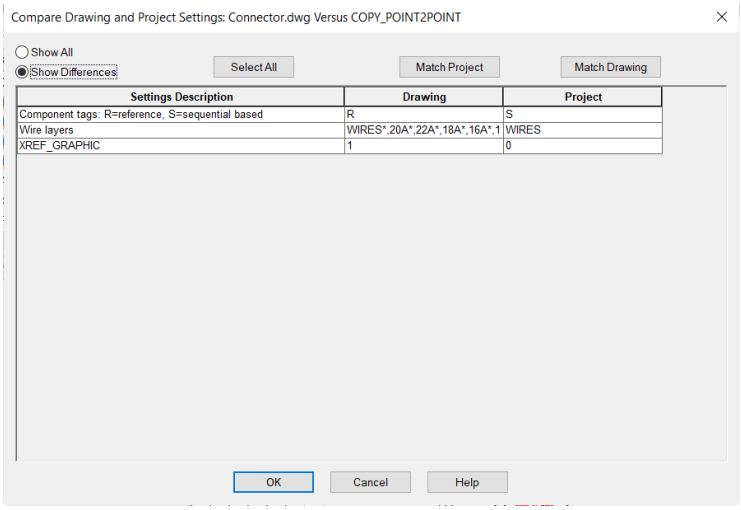


Figure 2-75 *The* ***Compare Drawing and Project Settings: DEMO007.DWG Versus COPY_POINT2POINT*** *dialog box*

Tutorial 5

In this tutorial, you will surf a component and extract component information and connection details of the **CADCIM** project to an external file using the **Location View** tab in the **PROJECT MANAGER**. **(Expected time: 15 min)**

The following steps are required to complete this tutorial:

a. Surf the component.
b. Extract component information.
c. Extract connection details.

Surfing the Component

1. Make sure the **CADCIM** project is activated. Choose the **Location View** tab from the **PROJECT MANAGER**. Next, choose the **CADCIM** node from the **PROJECT MANAGER**. Next, choose the **Display Details and Connections** button; the **PROJECT MANAGER** is expanded with the **Details** tab activated, as shown in Figure 2-76.

   You will notice that the component information is displayed in the **Details** pane in the grid.

2. Scroll down in the **PROJECT MANAGER** and select **LS406** from the **Tag** column. Next, right-click on it; a shortcut menu is displayed, as shown in Figure 2-77.

3. Choose **Surf** from the shortcut menu; the **Surf** dialog box is displayed, as shown in Figure 2-78. Choose **Go To** from the **Surf** dialog box; the *demo004.dwg* file opens with **LS406** zoomed in.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Note

*The options in the **Surf** dialog box are discussed in detail in Chapter 6.*

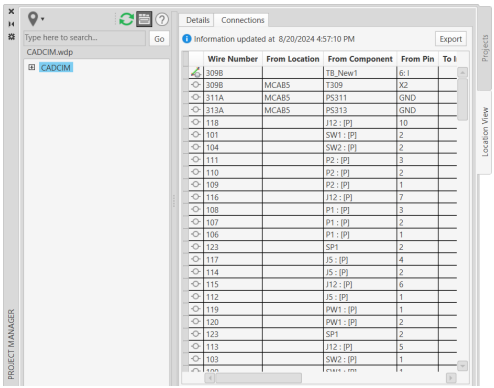


Figure 2-76** Expanded **PROJECT MANAGER

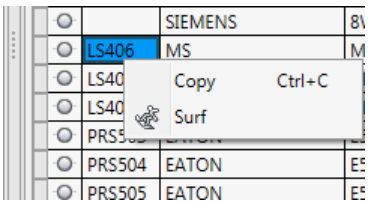


***Figure 2-77** Shortcut menu displayed*

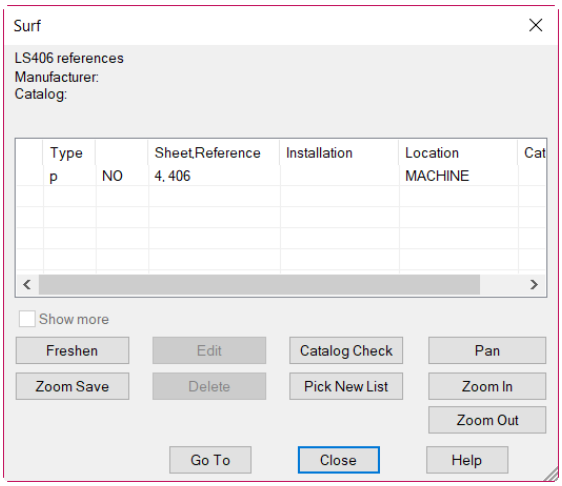


***Figure 2-78** The **Surf** dialog box*

Extracting the Component Information

1. Right-click on the **Tag** column in the grid; a shortcut menu is displayed. Choose **More** from this shortcut menu; the **Columns to display** dialog box is displayed.

2. Scroll down in the **Columns to display** dialog box and select the **Category**, **Rating1**, **Location** and **Terminal Number** check boxes, as shown in Figure 2-79. Next, choose **OK**; the **Category**, **Rating1**, **Location** and **Terminal Number** columns are added to the grid in the **Details** tab.

3. Choose **Export** from the **PROJECT MANAGER**; the **Save As** dialog box is displayed. Browse to c:*Users**User Name**Documents**Acade 2025**AeData**proj**CADCIM* and choose **Save**; the *ComponentDetails.csv* file is saved at the specified location.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Extracting the Connection Details

1. Choose the **Connections** tab in the **PROJECT MANAGER**; the **PROJECT MANAGER** is modified with connection details of the **CADCIM** project displayed in the grid.

2. Right-click on the **Wire Number** column in the grid; a shortcut menu is displayed. Choose **More** from this shortcut menu; the **Columns to display** dialog box is displayed.

3. Scroll down in the **Columns to display** dialog box and select the **Wire Type**, **Wire Color**, and **Wire Gauge** check boxes. Next, choose **OK**; the **Wire Type**, **Wire Color**, and **Wire Gauge** columns are added to the grid in the **Connections** tab.

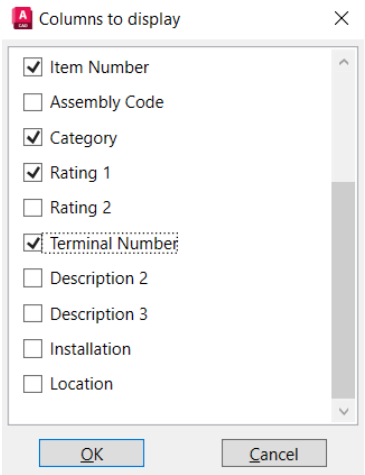


Figure 2-79 *The* ***Columns to display*** *dialog box*

4. Choose **Export** from the **PROJECT MANAGER**; the **Save As** dialog box is displayed. Navigate to c:*Users**User Name**Documents**Acade 2025**AeData**proj**CADCIM* and choose **Save**; the *ConnectionDetails.csv* file is saved at the specified location.

Self-Evaluation Test

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

1. Which of the following buttons in the **PROJECT MANAGER** is used to execute the pending updates?

   (a) **New Project** (b) **Refresh**
   (c) **Project Task List** (d) **Surfer**

2. The _____ option is used to plot all drawings of a project or a specified group of drawings.

3. The _____ option is used to create copies of an existing project with all the drawings and related files.

4. The _____ dialog box is used to edit the properties of a drawing.

5. Choose the _____ button to preview a selected drawing.

6. You can add up to _____ lines of description per page for each drawing in a project.

7. A project file is an ASCII text file. (T/F)

8. The **PROJECT MANAGER** is used to manage the project files only. (T/F)

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

9. A project file always consists of drawings placed in a single directory. (T/F)

10. You can switch between sequential drawings in an active project. (T/F)

Review Questions

Answer the following questions:

1. Which of the following buttons is used to change the appearance of a drawing list displayed in the **PROJECT MANAGER**?

   (a) **Plot Project** (b) **Project Task List**
   (c) **Move Up** (d) **Drawing List Display Configuration**

2. Which of the following options is used to add existing drawing(s) to a project?

   (a) **Descriptions** (b) **Add Active Drawing**
   (c) **Add Drawings** (d) **Title Block Update**

3. The **Remove Drawings** option is used both for removing and deleting a drawing file. (T/F)

4. You can close only the non-active projects. (T/F)

5. Projects can be closed by using the Project selection drop-down list. (T/F)

EXERCISES

Exercise 1

Create a new project with the project name **NEW_PROJECT** and add appropriate description to it. **(Expected time: 10 min)**



Note

*It is recommended to save all exercises of the forthcoming chapters in the **NEW_PROJECT** project.*

Exercise 2

Create a new drawing with the name *C02_exer02.dwg* in the project created in Exercise 1 and add appropriate description to it. **(Expected time: 10 min)**

Exercise 3

Using the **Copy** button, create a new project with the name **COPY_PROJECT** containing the drawings and settings of any existing project. **(Expected time: 10 min)**

Answers to Self-Evaluation Test

1. C, **2. Plot Project**, **3. Copy Project**, **4. Drawing Properties**, **5. Preview**, **6.** three, **7.** T, **8.** F, **9.** F, **10.** T

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com