Design Master Electrical
User Manual

Table of Contents

VII

Concepts and Procedures 593

Design Master Electrical

You are reading the user manual for Design Master Electrical 8.7.

You can **download a PDF version of the user manual** for printing and offline documentation purposes.

Technical Support

M - F, 9am - 5pm Eastern
support@designmaster.biz
1-866-516-9497 x2

Copyright 2012-2023, Design Master Software, Inc.

Overview

Design Master Electrical is a complete electrical drafting and calculation software program that works directly inside AutoCAD and BricsCAD.

One-line Riser Diagrams: Design and draft an intelligent one-line diagram. Automatically keep your one-line riser diagram up-to-date as your project changes.

Panel Schedules and Service Load Calculations: Keep track of all of the devices connected to your panels. Calculate service loads and size feeders.

Fault Current and Voltage Drop Calculation: Calculate available fault current at each panel. Calculate voltage drop on your feeders and branch circuits.

Lighting and Power Plans: Simplify the design and drafting of your lighting and power plans.

Equipment Connections and Motors: Create equipment connections and motors.

Photometrics: Calculate indoor and outdoor photometrics.

Command Reference

This section describes all of the commands available in Design Master Electrical. It is a direct listing of all of the commands and what they do. More complete discussions about how to apply the commands are found in the **Concepts and Procedures** section.

The commands in this section are in the same order as the Design Master Electrical pulldown menu:

```
🔍  Project Explorer
🔁  Update All Schedules and One-Line Diagram
    Distribution Equipment                    >
    Distribution Equipment Schedules          >
    One-Line Diagram                          >
    Circuiting and Homeruns                   >

    Modify Devices                            >
    Light Fixtures                            >
    Receptacles                               >
    Equipment                                 >
    Junction Boxes                            >
    Switches                                  >
    Low Voltage Devices                       >
    Conduits                                  >
    Cable Tray                                >
    3D-BIM                                    >
    Layer Management                          >

    Point-by-Point Photometrics               >
    Building Definition                       >
    Takeoffs                                  >
    Lighting Groups                           >

    Utilities                                 >
    Help                                      >
    Customization                             >

    Alignment Points                          >
    Start New Electrical Project
```

Project Explorer

The `Project Explorer` command lists all of the items included in the current project, organized by item type and by drawing.

To view a list of all items included in the current project, or to find a specific item, go to

Ribbon: `DM Electrical->Utilities->` 🔍 `Project Explorer`

Pulldown Menu: `DM Electrical->Project Explorer`

Project Explorer Dialog Box
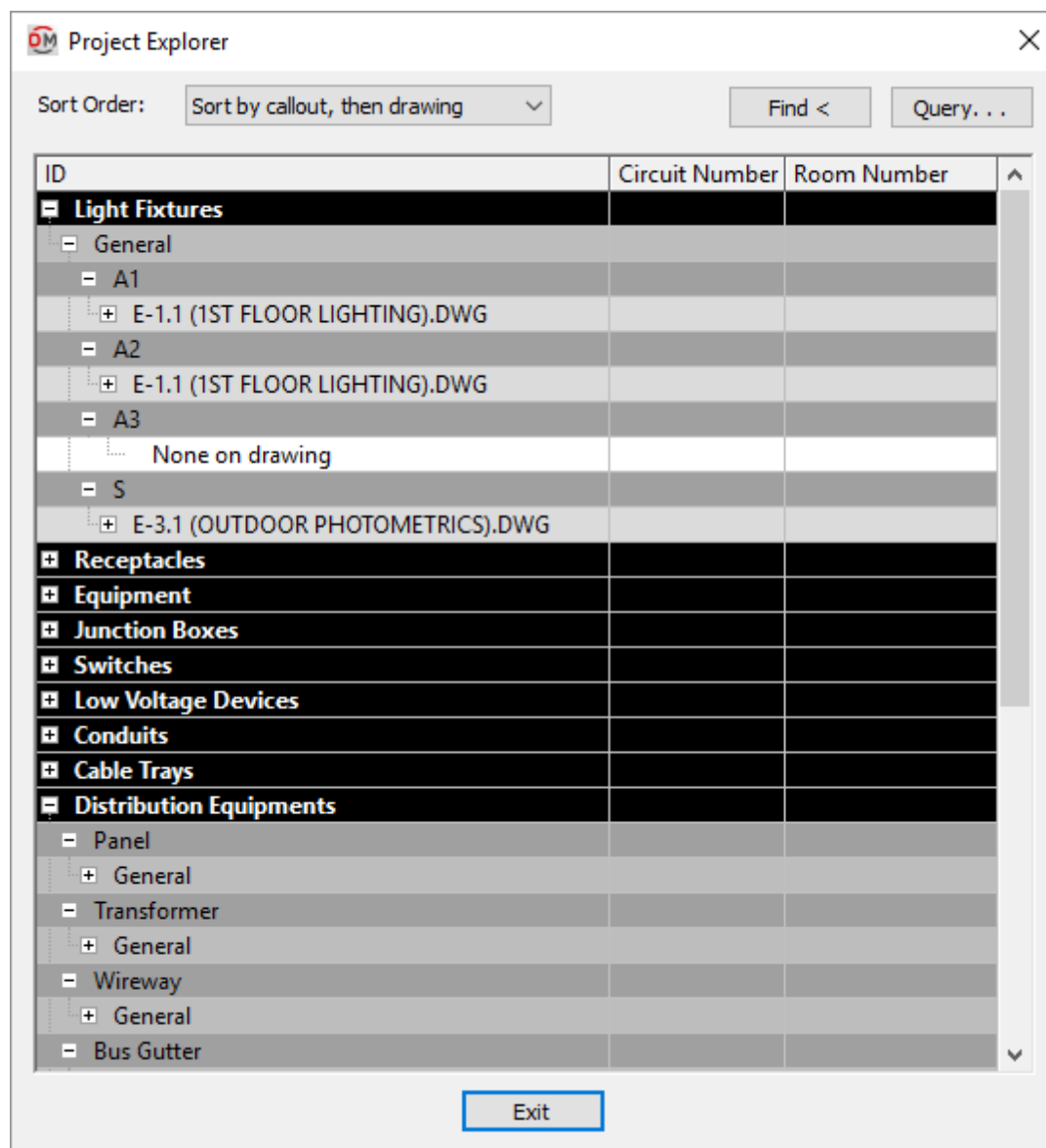
Sort Order: How the list is sorted.
- **Sort by callout, then drawing:** Under each device type, all of the callouts are listed. Under each callout, all of the drawings containing devices using that callout are listed. Callouts that do not have devices inserted on a drawing are listed, but no drawings are listed under them.
- **Sort by drawing, then callout:** Under each device type, all of the drawings containing devices of that type are listed. Under each drawing, all of the callouts on that drawing are listed. Callouts that do not have devices inserted on a drawing are not listed.

ID: This tree lists all of the items in the current project. The main item types are listed first, followed by each callout or drawing depending upon the specified *Sort Order*.

A number is displayed for each individual device. This number is the unique internal identifier used in the project database for the device. The specific value does not have any meaning other than the fact that it is different for each device. In general, devices inserted later in the project will have higher values than devices that are inserted earlier.

Circuit Number: If a device is connected to a circuit, the circuit number is listed here. This information is provided to help you differentiate between the devices in your project.

Room Number: If a device is inserted inside a room, the room number is listed here. This information is provided to help you differentiate between the devices in your project.

Finding an Item Using the Project Explorer Dialog Box

Find: Press this button to display the selected device on the drawing. If the device is located on another drawing, the other drawing will be opened and the device highlighted. A line will be displayed between the cursor and the device on the drawing to help you locate it.

You will be prompted at the command line regarding what to do next.

[Query device/Next device/Previous device/<Return to dialog>]:

> *Query device:* The device is queried as if the `Query Device` command had been used on it. See the **Query Device** section for more information.

> *Next device:* Locate the next item of the current type.

> *Previous device:* Locate the previous item of the current type.

> *Return to dialog:* Return to the **Project Explorer** dialog box.

Querying an Item Using the Project Explorer Dialog Box

Query: Press this button to query a callout or individual device.

Devices are queried as if the `Query Device` command had been used. See the **Query Device** section for more information.

Callouts are queried using the appropriate schedule command. The schedule dialog boxes are described in their respective sections.

Update All Schedules and One-Line Diagram

To update all schedules and one-line diagrams on the current drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` ⟳ `Update All Schedules and One-Line`

```
Diagram
```

Pulldown Menu: `DM Electrical->Update All Schedules and One-Line Diagram`

Design Master Electrical Dialog Box
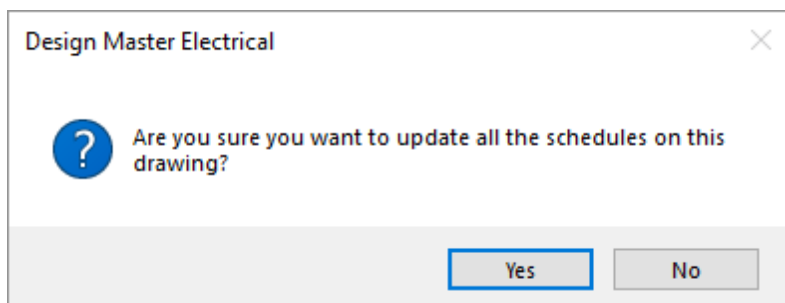
Press **Yes** to update all of the schedules.

Every schedule and the one-line diagram will be updated on the current drawing.

Addressing Distribution Equipment Calculation Errors

When updating the fault schedule and voltage drop schedule, it is possible that the calculation will not be correct if all distribution equipment is not properly defined. A dialog box will be displayed alerting you to these errors if they occur. For more information about fixing these calculation errors, see the **Addressing Distribution Equipment Calculation Errors** section.

Distribution Equipment

This section describes how to create and manage distribution equipment. "Distribution equipment" is a term we use to refer to all of the electrical distribution devices that may appear in your project except for devices on branch circuits.

Types of Distribution Equipment

There are thirteen different types of distribution equipment that can be created. If your project includes equipment that is not explicitly listed, use another type of equipment that is most similar.

- **Panels**
- **Switchboards**
- **Transformers**
- **Wireways**
- **Bus Gutters and Bus Ducts**
- **Enclosed Breakers and Disconnects**
- **Uninterrupted Power Supplies and Phase Inverters**
- **Motor Control Centers**
- **Meter Centers**
- **Generators**
- **Transfer Switches**
- **Low Voltage Panels**
- **Recurring Panel Templates**

Training Videos

- **Recurring Panels Overview**

Distribution Equipment Dialog Box Settings

Distribution equipment is created and modified using the distribution equipment dialog box. The dialog box will display different fields and buttons dependent upon the type of equipment, but typically contains the following settings:

- **General Settings**
- **Group Settings**
- **Fault Analysis Settings**
- **Feeder Specification**
- **Feeder and Service Calculation Settings**
- **Plan-View and One-Line Diagram Block Settings**
- **Schedule and Length Settings**
- **Notes Settings**
- **Arc-Flash Settings**

General Settings

This section describes the general settings in the distribution equipment dialog boxes. This area is used to specify values related to load, voltage, connection type and more. The information listed below is applicable to all distribution equipment unless otherwise indicated.

Connected Load: The connected load on the panel, in kVA and in amps.

Calculated Load: The calculated load on panel, including all of the diversity factors, in kVA and in amps.

Volts: The voltage of the equipment. Does not apply to low voltage panels, transformers and UPSs.

The *Volts* setting can be changed when there are no connections to or from the distribution equipment. If the equipment is connected to another equipment, another equipment is connected to it, or any branch circuit devices are connected, the *Volts* setting cannot be directly changed. Instead, use the **Change** button to change the *Volts* of the equipment. This button is required to properly handle anything that is already connected. See the **Changing the Voltage of Distribution Equipment** section below for more information.

Primary/Input Volts / Secondary/Output Volts: The primary or input voltage and secondary or output voltage of the equipment. Applies to transformers and UPSs. If the transformer or UPS is connected to another, different equipment, use the **Change** button to change this setting. See the **Changing the Voltage of Distribution Equipment** section for more information.

The *Primary Volts* and *Secondary Volts* settings for transformers must both be single-phase or three-phase. The *Input Volts* and *Output Volts* settings can be different phases for UPSs.

Bus Amps: The size of the bus in amps. Applies to panels, switchboards, bus gutters/bus ducts, motor control centers, and meter centers.
Motor control centers have two settings: *Horizontal Bus Amps* and *Vertical Bus Amps*.

Main Disconnect Type: The type of disconnect for the equipment. The graphics on the panel bus on the one-line

diagram are controlled by this field. Applies to panels, switchboards, motor control centers, and meter centers.

- **Main Lugs Only:** The bus bar on the one-line diagram does not have a disconnect in it.
- **Breaker:** The bus bar on the one-line diagram includes a breaker graphic.
- **Fused Switch:** The bus bar on the one-line diagram includes a fused switch graphic.

Trip: The trip size of the main disconnect breaker or fused switch, in amps. This value cannot be set if *Main Disconnect Type* is set to **Main Lugs Only**. Applies to panels, switchboards, motor control centers, and meter centers.

Frame: The frame size of the main disconnect breaker or fused switch, in amps. This value cannot be set if *Main Disconnect Type* is set to **Main Lugs Only**. Applies to panels, switchboards, motor control centers, and meter centers.

Set Curve: Press this button to configure the curve for the main disconnect breaker or fused switch. The **Select Curve** dialog box will appear. If *Main Disconnect Type* is set to **Main Lugs Only**, this button is disabled. Applies to panels, switchboards, motor control centers, and meter centers.

Mounting: How the equipment is mounted. The value of this setting can be displayed on the distribution equipment schedule. It is for informational purposes only and is not otherwise used. Does not apply to transformers, enclosed breakers/disconnects, or low voltage panels.

Neutral: The size of the distribution equipment neutral. The value of this setting can be displayed on the distribution equipment schedule. It is for informational purposes only and is not otherwise used. Applies to panels, switchboards, UPSs, transfer switches, and meter centers.

Number of <Connection Types>: How many connection locations are available on this equipment.
The label on this field depends upon the equipment type. The possible labels include *Number of Poles*, *Number of Positions*, *Number of Lugs*, *Number of Taps*, *Number of Starters*, and *Number of Breakers*.

Starting <Connection Type>: The number used for numbering the connection type listed.

The default is to start at the number 1 and go from there. However, in some cases, it is important to be able to start the numbering at a different point.

For example, on a two-section panel that is treated like an 84-circuit panel, the circuits are numbered from 1 to 42 and 43 to 84. On the second panel, you would set the *Starting Circuit* to **43**.

The label on this field depends upon the equipment type. The possible labels include *Starting Circuit*, *Starting Position*, *Starting Lug*, *Starting Tap*, *Starting Starter*, and *Starting Breaker*.

Lugs: The lugs available on the device. This setting controls the type of connection you can make using the **Connect Distribution Equipment** command and the graphics displayed on the bus bar on the one-line diagram. Applies to panels, switchboards, enclosed breakers/disconnects, transfer switches, and meter centers.
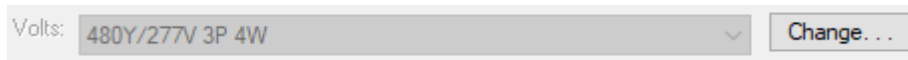
- **STANDARD:** One set of lugs at the top of the bus.
- **DOUBLE:** Double lugs located at the top of the bus. Enables the *Top Lugs* option in the **Connect Distribution Equipment** command.
- **FEEDTHRU:** Lugs located at the top and bottom of the bus. Enables the *Feedthrough* option in the **Connect Distribution Equipment** command.

IG Bus: Whether the equipment contains an isolated ground bus. The value of this setting can be displayed on the distribution equipment schedule. It is for informational purposes only and is not otherwise used. Applies to panels, switchboards, UPSs, transfer switches, and meter centers.

Layer System: See the **Layer System** section for more information.

Changing the Voltage of Distribution Equipment

When a distribution equipment is first created, it is possible to change the voltage using the *Volts* field. When the device is connected to another device, another device is connected to it, or any branch circuit devices are connected, the *Volts* setting cannot be directly changed. Instead, use the **Change** button to change the *Volts* of the device. This button is required to properly handle anything that is already connected.
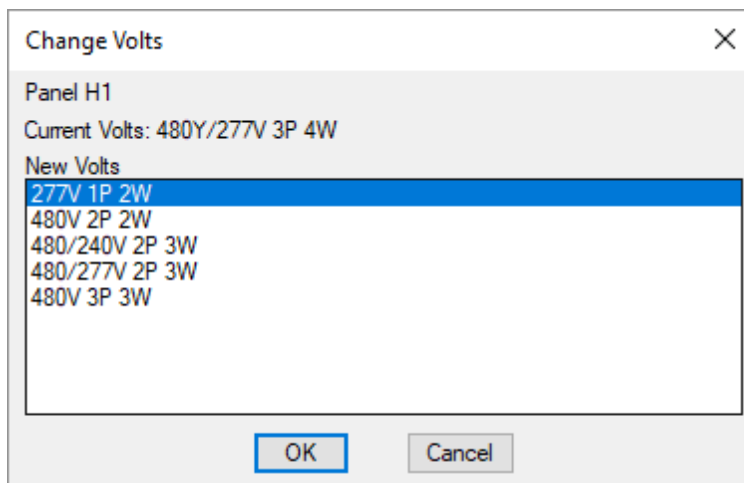
Press the **Change** button to display the **Change Volts** dialog box. The dialog box that is displayed depends upon whether you are changing the voltage of a transformer or another type of distribution equipment.

Change Volts Dialog Box (Non-transformers)

The dialog box shown below is used for distribution equipment other than transformers:

Current Volts: The current voltage supplied to the distribution equipment.

New Volts: A list of the current and possible voltages for the equipment. The listed voltages are the ones that share at least one circuit voltage in common.

Change Volts Dialog Box (Transformers and UPSs)

The dialog box shown below is used for transformers and UPSs:
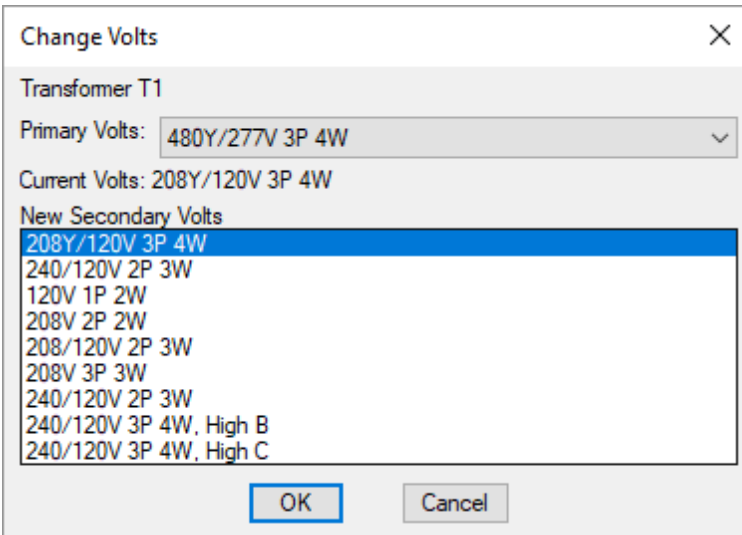
Primary Volts: The primary voltage supplied to the transformer.

Current Volts: The current voltage supplied to the transformer.

New Secondary Volts: A list of the current and possible voltages for the transformer. The listed voltages are the ones that share at least one circuit voltage in common.

The *Primary Volts* and *Secondary Volts* settings for transformers must both be single-phase or three-phase. The settings can be different phases for UPSs.

Specifying the New Voltage

Select the desired new voltage and press the **OK** button. The **Design Master Electrical** dialog box shown below will appear.
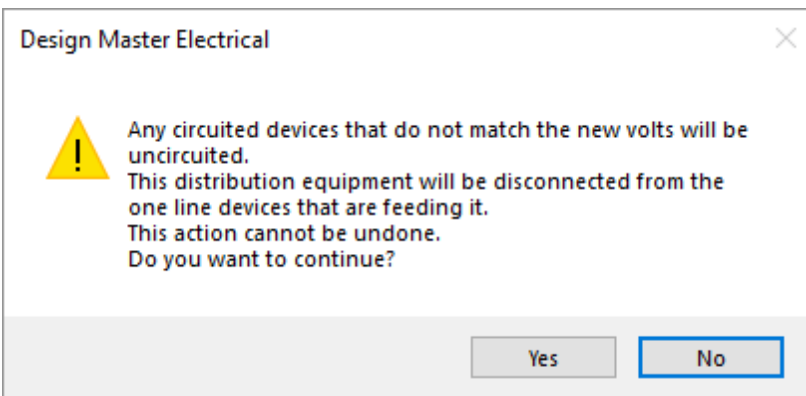
When the voltage on the distribution equipment is changed, the circuited devices are all checked for consistency. Any devices that cannot be connected to the newly selected voltage are removed and devices that can remain connected do so.

All distribution equipment connected to the selected equipment are disconnected. The distribution equipment is also disconnected from the device to which it was connected. These connections must be reestablished using

the **Connect Distribution Equipment** command.

Group Settings

This section describes the **Group** area in the distribution equipment dialog boxes. This area is used to assign the equipment to a schedule group and adjust group settings. The information listed below is applicable to all distribution equipment.

Select Group: Opens the **Schedule Groups** dialog box.

Move to Another Group: Opens a list of available groups the equipment can be moved to.

Copy to Another Group: Opens a list of available groups the equipment can be copied to. You can set a new callout for the copy.

Fault Analysis Settings

This section describes the **Fault Analysis** area in the distribution equipment dialog boxes. This area is used to specify values related to fault calculation analysis. The information listed below is applicable to all distribution equipment unless otherwise indicated.

To manually specify a value, check the box next to it unless otherwise indicated. Use this when you know the value of a specific item or want to override what is calculated for you.

Feeder Length: The length of the conductor feeding the equipment. The value is automatically calculated if both connected devices are inserted on the drawing.

The label on this field depends upon the device type. The possible labels include *Feeder Length*, *Primary Length*, and *Normal Feeder Length*.

Bypass Feeder Length / Emergency Feeder Length: The length of the second conductor feeding a UPS or a transfer switch. The *Normal Feeder Length* corresponds to the device connected to Source 1. This feeder length corresponds to the device connected to Source 2. The value is automatically calculated if both connected devices are inserted on the drawing.

Feeder Length Settings: Press this button to open the **Feeder Length Settings** dialog box. Use this dialog box to specify how the feeder length is calculated and set the feeder length and building angle. See the **Feeder Length Settings Dialog Box** section below for more information.

Transformer Impedance %: The impedance through a transformer, as a percentage. The fault current on the secondary of the transformer decreases when the impedance percentage increases. Applies only to transformers.

Manually enter this value if you know what it should be. It can typically be obtained from the transformer manufacturer.

If you do not know the value, a default based upon the kVA of the transformer is used:

Transformer kVA	Default Transformer Impedance %
0 - 100	1.75%
112.5 - 300	2%

Transformer kVA	Default Transformer Impedance %
500	2.5%
750+	5.75%

Transformer X/R Ratio: The X/R Ratio of the transformer. Applies only to transformers.

Manually enter this value if you know what it should be. It can typically be obtained from the transformer manufacturer.

A default value of 5 is used if you do not enter a value.

Set Fault @ <Device Type>: Press this button to set the fault at the device, in amps. This button is disabled if the box to the right is unchecked.

The label on this field depends upon the device type. The possible labels include *Set Fault @ Panel*, *Set Fault @ Primary*, *Set Fault @ Secondary*, and *Set Fault @ Tap*.

For transformers, the fault can be set at the primary and at the secondary.

Check the box next to this button to open the **Lock Fault** dialog box. Use this dialog box to specify the fault value at the equipment, based upon the information that you know. See the **Lock Fault Dialog Box** section below for more information.
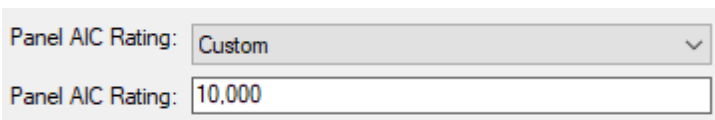
Total Fault: The most recently calculated fault at the device, taking into account the fault from the utility and any motors in the project.

Recalculate Fault: Press this button to recalculate the *Total Fault* value. Does not apply to UPSs and generators.

<Device Type> AIC Rating: The ampere interrupting capacity (AIC) rating for the device. This value is not automatically calculated. It is important to specifically choose this value for all of the equipment in the project. Does not apply to generators or low voltage panels.
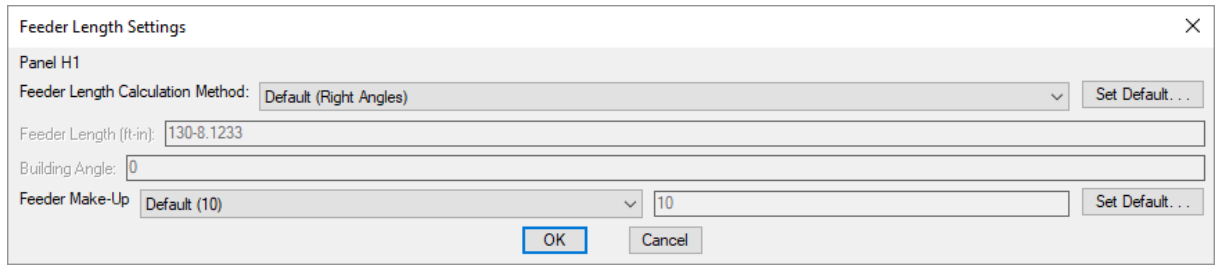
The label on this field depends upon the device type. The possible labels include *Panel AIC Rating*, *Secondary Breaker AIC Rating*, *Device Tap AIC Rating*, *Tap AIC Rating*, *Bypass Breaker AIC Rating*, *MCC AIC Rating*, *TS AIC Rating*, and *AIC Rating*.

When the **Custom** value is selected, a second *AIC Rating* field is available for manual input.

Feeder Length Settings Dialog Box

The **Feeder Length Settings** dialog box is displayed when you press the **Feeder Length Settings** button. It is used to set the feeder length calculation method or specify a feeder length manually.

Feeder Length Calculation Method: Sets how the feeder length is calculated. The values you can set vary depending upon the setting of this field.

- **Default:** The default calculation method for the project. The **Set Default** button allows you to change the *Feeder length calculation method* setting in the **Distribution Equipment** options section.
- **Right Angles:** Calculate the feeder length using line segments at right angles to one another. The direction of the segments is determined by the *Building Angle*.
- **Straight Line:** Calculate the feeder length using a straight line.
- **Custom:** Set the feeder length manually.

Feeder Length: The calculated length of the feeder in feet and inches. This value can be set manually when the calculation method is set to **Custom**.

Building Angle: Sets the building angle relative to the alignment point when *Feeder Length Calculation Method* is set to **Right Angles**. See the **How Building Angle Affects Calculations** article in the knowledge base for more information about this setting.

Feeder Make-Up: The extra length of wire included in the feeder length to account for the wire used for distribution equipment connections. It is used in the fault calculation, voltage drop calculation, and takeoffs. The **Set Default** button allows you to change the *Feeder make-up* setting in the **Distribution Equipment** options section.

Lock Fault Dialog Box

The **Lock Fault** dialog box is displayed when you check the box next to the **Set Fault** button or press the **Set Fault** button. It is used to specify the fault at the equipment.

Lock: The values you can set vary depending upon the setting of this field. The other values are calculated based upon the ones you have locked.

- **Fault:** Lock the *Fault* at the equipment. An *X/R Ratio* of 5 is used. Use this when you know the *Fault* but not the *X/R Ratio* of the equipment. This setting is the most common choice.
- **Fault and X/R Ratio:** Lock the *Fault* at the equipment and specify the *X/R Ratio*. Use this when you know the *Fault* and *X/R Ratio* of the equipment.
- **Z and X/R Ratio:** Specify the impedance (*Z*) and *X/R Ratio*.
- **X and R:** Specify the reactance (*X*) and resistance (*R*).

Line-to-Neutral Voltage: The line-to-neutral voltage of the equipment. This voltage is used in the fault calculations.

Fault: The fault at the equipment.

Z: The impedance of the equipment.

X/R Ratio: The X/R ratio of the equipment.

X: The reactance of the equipment.

R: The resistance of the equipment.

Ignore Impedance in Downstream Feeder: Whether the impedance in the downstream feeder is ignored. The fault at the downstream equipment will be the same as the fault at this equipment. This setting is required by plan reviewers in some jurisdictions.

Feeder Specification

This section describes the **Feeder Specification** area in the distribution equipment dialog boxes. This area is used to configure the upstream feeder to the equipment. The information listed below is applicable to all distribution equipment except generators and low voltage panels unless otherwise indicated.

UPSs and transfer switches have a **Normal Feeder Specification** area and a **Bypass Feeder Specification** area. These areas correspond to the two feeders to these types of equipment. They work the same as the **Feeder Specification** areas in the other dialog boxes.

Training Videos

- **Wire Sizing**

Fed From: The equipment that this equipment is connected to. It is the equipment upstream from this equipment that is closer to the utility.

Description in Upstream Device: The description used in the distribution equipment schedule where this equipment is connected. This description defaults to the equipment type and the equipment name. To override this value, check the box next to the *Description in Upstream Device* field and enter a new description.

Upstream OCP / Primary OCP: The trip size of the overcurrent protection (OCP) in the device upstream of the distribution equipment.
- **Same as Main Disconnect or Bus Amps:** The OCP is sized to match the *Main Disconnect Amps* setting. If the *Main Disconnect Type* is set to **Main Lugs Only**, the OCP is sized to match the *Bus Amps*.
  Size to Match kVA: The OCP is sized to match the *KVA* setting. Applies to transformers and UPSs.
  Same as TS Amps: The OCP is sized to match the transfer switch amps. Applies to transfer twitches.
- **Lugs or N/A:** There is no OCP for this device. No breaker size is listed in the upstream panel. The feeder is sized based upon the **Same as Main Disconnect or Bus Amps** setting.
- **Specific Breaker Size:** The OCP is set to the specific size chosen from the list.

Set Curve: Press this button to configure the curve for the upstream OCP. The **Select Curve** dialog box will appear. If *Upstream OCP* is set to **Size to Match kVA** or **Lugs or N/A**, this button is disabled.

Frame Amps: The frame size of the overcurrent protection (OCP) in the device upstream of the distribution equipment.
- **Same as Main Disconnect or Bus Amps:** The frame is sized to match the *Main Disconnect Amps* setting. If the *Main Disconnect Type* is set to **Main Lugs Only**, the frame is sized to match the *Bus Amps*.
  Size to Match kVA: The frame is sized to match the *KVA* setting. Applies to transformers and UPSs.

Same as TS Amps: The frame is sized to match the transfer switch amps. Applies to transfer switches.
- **Lugs or N/A:** There is no frame for this device. No breaker size is listed in the upstream panel. The feeder is sized based upon the **Same as Main Disconnect or Bus Amps** setting.
- **Specific Breaker Size:** The frame is set to the specific size chosen from the list.

Conductor Amps: Sets the size of the hot wires in the feeder.
- **Size Automatically:** The hot wires are sized based upon the *Upstream OCP* setting.
- **Specific Breaker Size:** The hot wires are sized to match the breaker size chosen from the list.

Neutral: Sets the size of the neutral wire in the feeder.
- **Same as Phase:** The neutral wire is the same size as the phase wires as specified in the *Conductor Amps* field.
- **Double Phase:** The neutral wire is twice the size of the phase wires as specified in the *Conductor Amps* field. This is accomplished by using two neutral wires.
- **None:** No neutral wire is included in the feeder.
- **Specific Breaker Size:** The neutral wire is sized to match the wire size chosen from the list.

Ground: Sets the size of the ground wire in the feeder.
- **NEC 250.122:** The ground wire is sized based upon the *Equipment Ground* setting for the ampacity specified in the *Upstream OCP* field.
- **NEC 250.102:** The ground wire is sized based upon the *Service Ground* setting for the ampacity specified in the *Conductor Amps* field.
- **None:** No ground wire is included in the feeder.
- **Specific Breaker Size:** The ground wire is sized to match the wire size chosen from the list.

IG Conductor: Whether the feeder includes an isolated ground.

Conduit Size: Sets the size of the conduit for the feeder. The conduit fill percentage is displayed next to the *Conduit Size* field. Changing the conduit size will update the fill percentage.
- **Size Automatically:** The conduit is sized automatically based upon the wires in the feeder. All conduits are sized using a 40% conduit fill (*NEC Table 1*).
- **None:** No conduit is included for the feeder.
- **Specific Breaker Size:** The conduit is sized to match the conduit size chosen from the list.

Size: The callout for the feeder listing all of the chosen wire sizes. This callout can be displayed on various schedules and on the one-line diagram.

Feeder and Service Calculation Settings

The **Feeder and Service Calculation Settings** dialog box is used to manage how loads are calculated for the selected distribution equipment. The chosen calculation method impacts how the various load types assigned to devices are used to determine the calculated load. See the **Feeder and Service Calculations** article in the knowledge base for more information about how to correctly set the load types, occupancy areas, and demand factors for each load calculation method.

Load Calculation Method: Determines how the service load is calculated, depending upon the type of structure. The **Set Default** button allows you to change the *Default load calculation method* setting in the **Distribution Equipment** options section.

- **General Commercial or Dwelling Unit, Part III:** Use *NEC 220 Part III*.
- **New Restaurant, All Electric:** Use the first column in *NEC 220.88*.
- **New Restaurant, Not All Electric:** Use the second column in *NEC 220.88*.
- **School:** Use *NEC 220.86*.
- **Dwelling Unit:** Use *NEC 220.82*.
- **Dwelling Unit, Existing:** Use *NEC 220.83*.
- **Multifamily Dwelling (Single units use Part III):** Use *NEC 220.84* and *NEC 220.85*. If the distribution equipment serves a single dwelling unit, the calculation uses *NEC 220 Part III*.
  The *Multifamily Dwelling Units* field sets the number of dwelling units used to calculate the diversity. The number of dwelling units on a panel will include the total number on downstream equipment.
- **Multifamily Dwelling (Single units use Part IV):** Use *NEC 220.84* and *NEC 220.85*. If the distribution equipment serves a single dwelling unit, the calculation uses *NEC 220.82*.
  The *Multifamily Dwelling Units* field sets the number of dwelling units used to calculate the diversity. The number of dwelling units on a panel will include the total number on downstream equipment.
- **Multifamily Dwelling (Single units use Existing):** Use *NEC 220.84* and *NEC 220.85*. If the distribution equipment serves a single dwelling unit, the calculation uses *NEC 220.83*.
  The *Multifamily Dwelling Units* field sets the number of dwelling units used to calculate the diversity. The number of dwelling units on a panel will include the total number on downstream equipment.
- **Marina:** Use *NEC 220 Part III*. Use *NEC 220.120* for the *Marina / Mobile Home / RV* load type.
  The *Shore Power Receptacles* field sets the number of shore power receptacles used to calculate the diversity.
- **Marina with Sub-Meters:** Use *NEC 220 Part III*. Use *NEC Table 220.120 Note 2* for the *Marina / Mobile Home / RV* load type.
  The *Shore Power Receptacles* field sets the number of shore power receptacles used to calculate the diversity.
- **Mobile Home Park:** Use *NEC 220 Part III*. Use *NEC 550.31* for the *Marina / Mobile Home / RV* load type.
  The *Mobile Homes* field sets the number of mobile homes used to calculate the diversity.
- **RV Park:** Use *NEC 220 Part III*. Use *NEC 551.73* for the *Marina / Mobile Home / RV* load type.
  The *Recreational Vehicle Sites* field sets the number of sites used to calculate the diversity.
- **None (Dynamic footer will be blank):** Use *NEC 220 Part III*. When the schedule is inserted on the

drawing, load calculations will not be displayed in the footer.

Default General Lighting Circuit Value: Sets whether circuits connected to the distribution equipment are automatically designated as General Lighting Circuits as defined by the NEC. Typically, this should be set to **No**. The **Set Default** button allows you to change the *Default general lighting circuit value* setting in the **Distribution Equipment** options section.

Schedule Block: Sets the schedule block for the distribution equipment. The **Set Default** button will open the **Set Default Schedule Block** dialog box. This dialog box allows you to set a default schedule block for the distribution equipment type. Select a schedule block from the available list and press the **OK** button.

Metered Demand: The current load for an existing building.

Future Demand: The anticipated load for future work. Can be entered in kVA or as a percentage of the calculated load.

Occupancy Areas

This section is used to calculate the demand for occupancy areas on the selected distribution equipment.

Type of Occupancy: The type of occupancy area. The available types are defined in *NEC Table 220.42(A)*.

The *Description* and *Unit Load* are based upon the selected type. Select **Custom** to enter these values manually.

Area: The size of the occupancy area, in square feet.

Description: The description of the occupancy area.

Unit Load: The unit load for the occupancy area. The unit load is based upon the type of occupancy as defined in *NEC Table 220.42(A)*.

Demand Factor: The demand factor for the occupancy area.
- **Other (Continuous):** The demand factor is based upon the *All others* row of *NEC Table 220.45*. The lighting will be calculated as a continuous load with a 125% multiplier applied.
- **Other (Noncontinuous):** The demand factor is based upon the *All others* row of *NEC Table 220.45*. The lighting will be calculated as a noncontinuous load with a 100% multiplier applied.
- **Dwelling Unit:** The demand factor is based upon the *Dwelling units* row of *NEC Table 220.45*.
- **Hospital:** The demand factor is based upon the *Hospitals* row of *NEC Table 220.42 (NEC 2017 and earlier)*. This demand factor should not be used for areas where the entire lighting is likely to be used at one time, as in operating rooms or dining rooms.
- **Hotel or Motel:** The demand factor is based upon the *Hotels and motels, including apartment houses without provision for cooking by tenants* row of *NEC Table 220.45*. This demand factor should not be used for areas where the entire lighting is likely to be used at one time, as in ballrooms or dining rooms.
- **Warehouse (Storage):** The demand factor is based upon the *Warehouses (storage)* row of *NEC Table 220.45*.

Related Options

Default load calculation method: Sets the default for the *Load Calculation Method* setting.

Plan View and One-Line Diagram Block Settings

The **Plan View and One-Line Diagram Block Settings** dialog box is used to manage the blocks that represent the distribution equipment on the plan and one-line diagram. The information listed below is applicable to all distribution equipment except generators and low voltage panels unless otherwise indicated.

Elevation: The elevation of the distribution equipment. See the **Elevation** section for more information.

Select Block Based upon kVA: Whether the *Plan View Block* is automatically set based upon the kVA setting. Applies to transformers only.

Plan View

Plan View Block: The block used for the distribution equipment on the drawing. This list is based upon the settings in the **Distribution Equipment Plan View Blocks** command.

If the *Plan View Block* is set to a block with *Custom Size* set to **Yes** in the **Distribution Equipment Plan View**

Blocks dialog box, the *Depth*, *Width*, and *Height* fields can be used to set the size of the equipment, in inches. These fields are disabled when a block with *Custom Size* set to **No** is selected.

Depth: The size of the equipment in the y direction, perpendicular to the rotation angle of the block.

Width: The size of the device in the x direction, parallel to the rotation angle of the block.

Height: The size of the device in the z direction.

One-Line Diagram

This section sets the blocks used for the distribution equipment and its connections on the one-line diagram. The **Set Default** button allows you to change the default for each block in the **One-Line Diagram** options section.

Insert Feeder to Distribution Equipment: Whether the upstream feeder for the distribution equipment is included on the one-line diagram.

Initial Block: The block used for the distribution equipment itself when it is generated. Changing this setting does not affect blocks already on the one-line diagram.

Initial Fed From: Whether the feeder is drawn to the top or bottom of the block when it is generated. Changing this setting does not affect blocks already on the one-line diagram.

Upstream OCP Block+: The block for the upstream OCP. **<Use upstream distribution equipment OCP block>** uses the same OCP block as the upstream distribution equipment. Changing this setting will affect blocks already on the one-line diagram.

Upstream OCP Block 2+: The block for the second upstream OCP. Applies to transfer switches and UPSs only.

Circuit OCP Block+: The block for the downstream OCP. Changing this setting will affect blocks already on the one-line diagram.

Main Disconnect Block: The block for the main disconnect. Only visible if *Initial Block* is set to **<Custom Panel Box with Bus>**. Not available if *Main Disconnect Type* is set to **Main Lugs Only**. Does not apply to transformers, transfer switches, or UPSs.

Related Options

Elevation input: Sets whether the elevation is measured from the alignment point or the ground by default.

Schedule and Length Settings

The **Schedule and Length Settings** dialog box is used to manage length calculation settings for the selected distribution equipment, as well as how or whether the distribution equipment is displayed in different schedules.

Schedule Block: The block used for the distribution equipment schedule. This list is based upon the settings in the **Distribution Equipment Schedule Blocks** command.

Length Calculation Method

Connected Feeder Length Calculation Method: Sets the feeder length calculation method when measuring from this distribution equipment to another distribution equipment. See the **Feeder Length Settings Dialog Box** section for more information.

Branch Circuit Length Calculation Method: Sets the branch circuit length calculation method when measuring from this distribution equipment to a light fixture, receptacle, or other device. The **Set Default** button allows you to change the *Circuit length calculation method* setting in the **Distribution Equipment** options section.

Feeder and Circuit Make-Up

Feeder Make-Up: The extra length of wire included in the feeder length to account for the wire used for distribution equipment connections. It is used in the fault calculation, voltage drop calculation, and takeoffs. The **Set Default** button allows you to change the *Feeder make-up* setting in the **Distribution Equipment** options section.

Branch Circuit Make-Up: The extra length of wire included in branch circuit lengths to account for branch circuit connections. It is used in the fault calculation, voltage drop calculation, and takeoffs. The **Set Default** button allows you to change the *Branch circuit make-up* setting in the **Circuits, Load Types, and Wire Sizes** options section.

Add Branch Circuit Make-Up For Each Device: Whether the length specified in *Branch Circuit Make-Up* is added for each device on the circuit. The **Set Default** button allows you to change the *Add branch circuit make-up for each device* setting in the **Circuits, Load Types, and Wire Sizes** options section.

- **Yes:** The make-up added to the branch circuit length is multiplied for each device on the circuit.
- **No:** The make-up is only added to the branch circuit length once, regardless of the number of devices on the circuit.

Show in Feeder Schedule: Whether the upstream feeder for the distribution equipment is displayed in the feeder schedule.

Show in Fault Schedule: Whether the distribution equipment is displayed in the fault schedule.

Show in Voltage Drop Schedule: Whether the distribution equipment is displayed in the voltage drop schedule.

Show in One-Line Diagram: Whether the distribution equipment is displayed on the one-line diagram.

Show in List: Whether the distribution equipment is displayed in the list of distribution equipment of this type.

Related Options

Feeder length calculation method: Sets the default for the *Feeder Length Calculation Method* setting.

Circuit length calculation method: Sets the default for the *Branch Circuit Length Calculation Method* setting.

Building angle: Sets the default value for the *Building Angle* field.

Feeder make-up: Sets additional length to be added to feeders for connections.
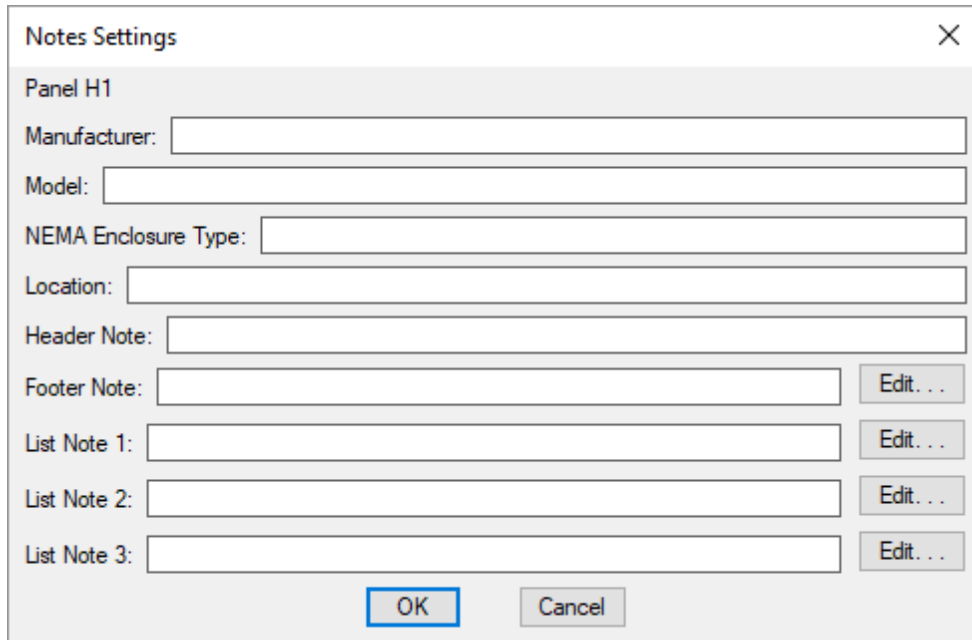
This option can be further customized with the **Ignore make-up on feeders shorter than this** option.

Branch circuit make-up: Sets additional length to be added to branch circuits for connections.

This option can be further customized with the **Add branch circuit make-up for each device** option.

Notes Settings

The **Notes Settings** dialog box is used to manage notes for the selected distribution equipment. They are for informational purposes only and are not otherwise used.

Manufacturer: The manufacturer of the distribution equipment. This note can be displayed on the header of distribution equipment schedules, distribution equipment lists, and the one-line diagram.

Model: The model of the distribution equipment. This note can be displayed on the header of distribution equipment schedules, distribution equipment lists, and the one-line diagram.

NEMA Enclosure Type: The NEMA rating for the distribution equipment. This note can be displayed on the header of distribution equipment schedules, distribution equipment lists, and the one-line diagram.

Location: The location of the distribution equipment. In the default distribution equipment schedule blocks, this note is displayed in the header.

Header Note: A generic note field. In the default distribution equipment schedule blocks, this note is displayed in the header.

Footer Note: A generic note field. In the default distribution equipment schedule blocks, this note is displayed in the footer below the schedule.

List Note 1: A generic note field that is displayed on distribution equipment lists only. It cannot be displayed on the distribution equipment schedule or one-line diagram.

List Note 2: A generic note field that is displayed on distribution equipment lists only. It cannot be displayed on the distribution equipment schedule or one-line diagram.

List Note 3: A generic note field that is displayed on distribution equipment lists only. It cannot be displayed on the distribution equipment schedule or one-line diagram.

Arc-Flash Settings

The **Arc-Flash Settings** dialog box is used to manage the variables that affect arc-flash calculations for the selected distribution equipment.

Voltage: The voltage of the distribution equipment. The voltage determines the arc-flash calculation method that is used.

Calculate Arc-Flash: Whether arc-flash is calculated for the distribution equipment. Equipment that is not calculated will not be displayed in the arc-flash schedule on the drawing and will not have stickers created. Because it is difficult to sustain an arc-flash below 208V, equipment with voltages below 208V do not need arc-flash calculations performed and will have this option set to **No** by default.

Electrode Configuration: The electrode configuration as described in *IEEE Std 1584-2018 Table 9*. Horizontal configurations generally have higher incident energy than vertical configurations. Enclosed configurations generally have higher incident energy than open configurations. Most distribution equipment inside buildings should be set to an enclosed configuration. See the **Electrode Configurations for Arc-Flash Calculations** article in the knowledge base for more information about these configurations.
- **Unknown:** This setting should be used if the electrode configuration is unknown. The calculations provide conservative results.

- **Vertical conductors or electrodes inside a metal box or enclosure:** This setting has the lowest incident energy among enclosed configurations.
- **Vertical conductors or electrodes terminated in an insulating barrier inside a metal box or enclosure:** This setting has a lower incident energy than **Horizontal conductors or electrodes inside a metal box or enclosure**, but higher than **Vertical conductors or electrodes inside a metal box or enclosure**.
- **Horizontal conductors or electrodes inside a metal box or enclosure:** This setting has the highest incident energy among all configurations.
- **Vertical conductors or electrodes in open air:** This setting has the lowest incident energy among all configurations.
- **Horizontal conductors or electrodes in open air:** This setting has the highest incident energy among open configurations.

Enclosure Size: The dimensions of the enclosure. The smaller the enclosure, the higher the incident energy. The size of the enclosure does not affect arcing current and arcing times.
- **Unknown:** This setting should be used if the dimensions are unknown. The calculations will be based upon a 20" x 20" x 9" enclosure.
- **Known:** Enter values for the *Enclosure Width* and *Enclosure Height* fields and set **Enclosure Depth** manually. If possible, set the dimensions based upon the manufacturer's specifications. Typical enclosure dimensions are listed in *IEEE Std 1584-2018 Table 8*. Dimensions for equipment between 208V and 600V are listed in the table below.

Equipment Type	Enclosure Size (H x W x D)
Switchgear	20" x 20" x 20"
MCC	14" x 12" x 8"
Panel	14" x 12" x 8"
Cable Junction Box	14" x 12" x 8"

Enclosure Height: The height of the enclosure.

Enclosure Width: The width of the enclosure.

Enclosure Depth: The depth of the enclosure.
- **Typical:** The calculations will be based upon a depth greater than 8".
- **Shallow:** The calculations will be based upon a depth of 8" or less.
- **Specific:** Enter a value for the *Enclosure Depth* field manually.

Gap Between Conductors: The gap between the conductors. For devices between 208V and 600V, the most accurate calculation can be used when the gap between conductors is between 0.25" and 3". Within that range, the larger the gap, the higher the incident energy. For gaps outside this range, the calculations provide conservative results.
- **Unknown:** This setting should be used if the gap is unknown. For devices between 208V and 600V, the calculations will be based upon a gap of 3". For devices above 600V, the calculations will be based upon a gap of 10".
- **Known:** Enter a value for the *Gap Between Conductors* manually. If possible, set the gap based upon the manufacturer's specifications. Typical gaps between conductors are listed in *IEEE Std 1584-2018 Table 8*. Gaps for devices between 208V and 600V are listed in the table below.

Equipment Type	Gap Between Conductors
Switchgear	1.25"
MCC	1"
Panel	1"
Cable Junction Box	0.5"

Working Distance: The distance from the possible arc point to the torso of the person working on the distribution equipment. The incident energy will be calculated at this distance from the equipment. The farther away from the equipment, the lower the incident energy. Any part of the body closer to the equipment than this distance will be exposed to higher incident energy than is calculated.
- **Unknown:** This setting should be used if the working distance is unknown. The calculations will be based upon a working distance of 12".
- **Known:** Enter a value for the *Working Distance* manually. If possible, set the working distance based upon the actual dimensions of the equipment. Typical working distances are listed in *IEEE Std 1584-2018 Table 10*. Working distances for equipment between 208V and 600V are listed in the table below.

Equipment Type	Working Distance
Switchgear	24"
MCC	18"
Panel	18"
Cable Junction Box	18"

Bolted Fault Current: The bolted fault current is the fault at the equipment. This value is calculated based upon the other settings in your project. All of the arc-flash calculations assume the equipment and the fault are three-phase. For single-phase equipment, the calculations provide conservative results. If the fault is less than 500 A or greater than 106,000 A, the calculations provide conservative results.

Arcing Current, Maximum and Reduced: The predicted three-phase arcing current used to determine the operating time for the protective devices. The maximum arcing current is the upper limit for the equipment. The reduced arcing current is the lower limit for the equipment.

These values are calculated based upon ranges specified in *IEEE Std 1584-2018*. If any of the following values for the equipment are outside the listed range, the calculation will be performed using the Lee Method:
- **Voltage:** 208V to 15,000V
- **Bolted Fault Current:** 500A to 106,000A at 208V to 600V; 200A to 65,000A above 600V
- **Gap Between Conductors:** 0.25" to 3" at 208V to 600V; 0.75" to 10" above 600V

If the Lee Method is used, only the *Arcing Current, Maximum* will be calculated.

Manually Set Arcing Time: Whether the arcing times are entered manually or set automatically based upon time-current curves set in the **Select Curve** dialog box.
- **Yes:** Enter values for the arcing times and *Arcing Time Basis* field manually.
- **No:** The arcing times and *Arcing Time Basis* field will be set automatically based upon the next protective device that is upstream of the equipment. If an upstream protective device has not been set, the arcing times will be set to 2 seconds and the *Arcing Time Basis* will display "UNKNOWN BREAKER" for the protective device.

Duration at Maximum Arcing Current and Reduced Arcing Current: The duration of the arc-flash at the *Maximum Arcing Current* and *Reduced Arcing Current*. The duration has a significant impact on the incident energy in seconds. The longer you are exposed to the flash, the more intense the burn. The arcing time is based upon the time-current curve for the specific breaker you are using. These values are provided by the manufacturer and can typically be found on their website.

The incident energy is a function of the arcing current and the duration. Lower arcing currents that take longer to close the breaker can result in higher incident energy values than high arcing currents. To account for this, incident energy is calculated using both the *Maximum Arcing Current* and *Reduced Arcing Current*.

Fuses must be handled differently from breakers. The time-current curves for fuses may include both melting and clearing times. Use the clearing time if it is provided. If only the melting time is provided, you can approximate the clearing time by adding 10% to the melting time if it is greater than 0.3 seconds, or 15% if it is less than 0.3 seconds.

If the arcing current is above the total clearing time at the bottom of the curve (0.01 seconds), use 0.01 seconds.

Arcing Time Basis: The time-current curve the arcing times are based upon. The value of this setting can be displayed on the arc-flash schedule and one-line diagram. It is for informational purposes only and is not otherwise used.

Incident Energy, Maximum Arcing Current and Reduced Arcing Current: The total incident energy based upon the arcing current, arcing times, and working distance.

Arc-Flash Boundary Distance: The limited approach boundary for unprotected workers. At this distance, the harm inflicted upon an unprotected worker during an arc-flash incident will be limited to second-degree burns.

Insert on One-Line Diagram

Press the **Insert on One-Line Diagram** button to insert the active distribution equipment on the one-line diagram. The distribution equipment dialog box will close and the **Insert Distribution Equipment** command will run with the active distribution equipment selected.

If the selected distribution equipment is not connected to another device, after inserting it on the one-line diagram, you will be prompted to connect to an upstream distribution equipment that is closer to the utility.

Select upstream distribution equipment to connect to:

Select another device on the one-line diagram to proceed the same way as running the **Connect Distribution Equipment on One-Line Diagram** command, or press **ENTER** to finish the command without making a connection.

Insert on Plan View

Press the **Insert on Plan View** button to insert the active distribution equipment on the drawing. The distribution equipment dialog box will close and the **Insert Plan View Block** command will run with the active distribution equipment selected.

Querying Distribution Equipment

When the `Query Device` command is used on distribution equipment, it displays and can modify information related to the distribution equipment on the drawing. It does not display all of the information about the equipment.
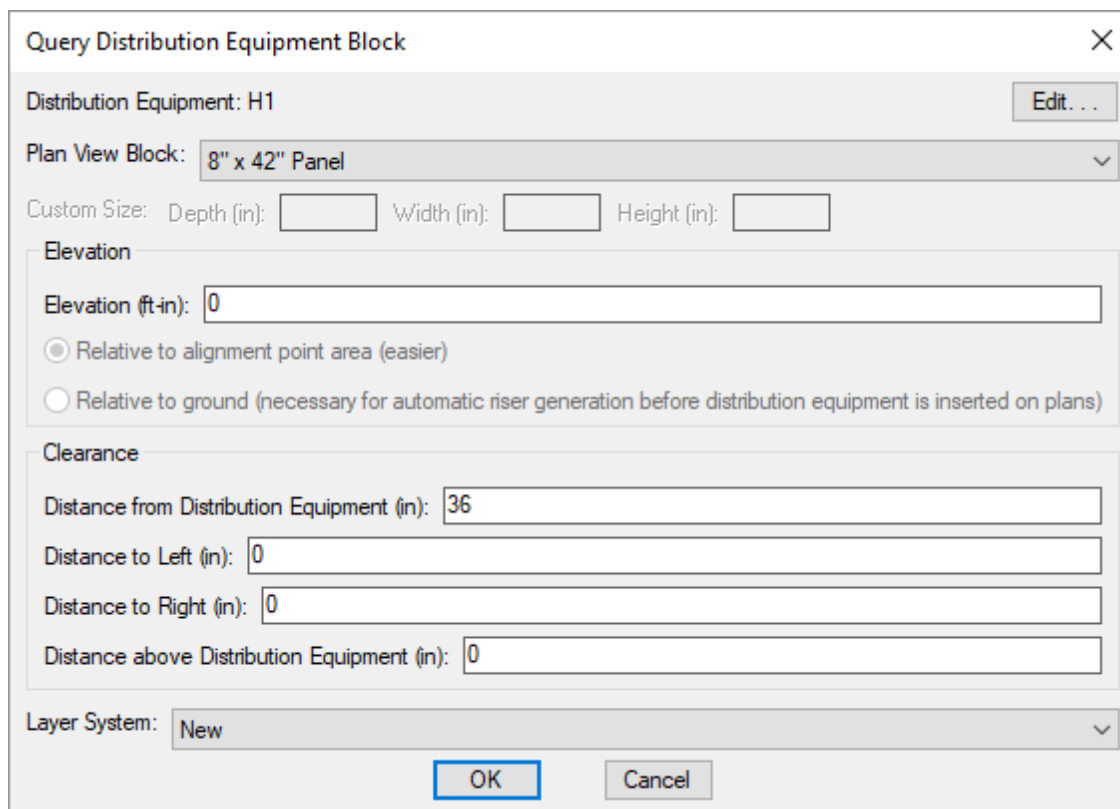
To modify a distribution equipment that is inserted on the drawing, go to

Ribbon: `DM Electrical->Devices->` `Query Device`

Pulldown Menu: `DM Electrical->Modify Devices->Query Device`

Select the distribution equipment device on the drawing you want to query.

Query Distribution Equipment Block Dialog Box

Changes can be made to the distribution equipment block settings when querying the device in the same way as when it was inserted. See the **Insert Plan View Block** command for more information.

Edit Distribution Equipment

To edit distribution equipment that has already been created, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Edit Distribution Equipment`

Pulldown Menu: `DM Electrical->Distribution Equipment->Edit Distribution Equipment`

Edit Distribution Equipment Dialog Box
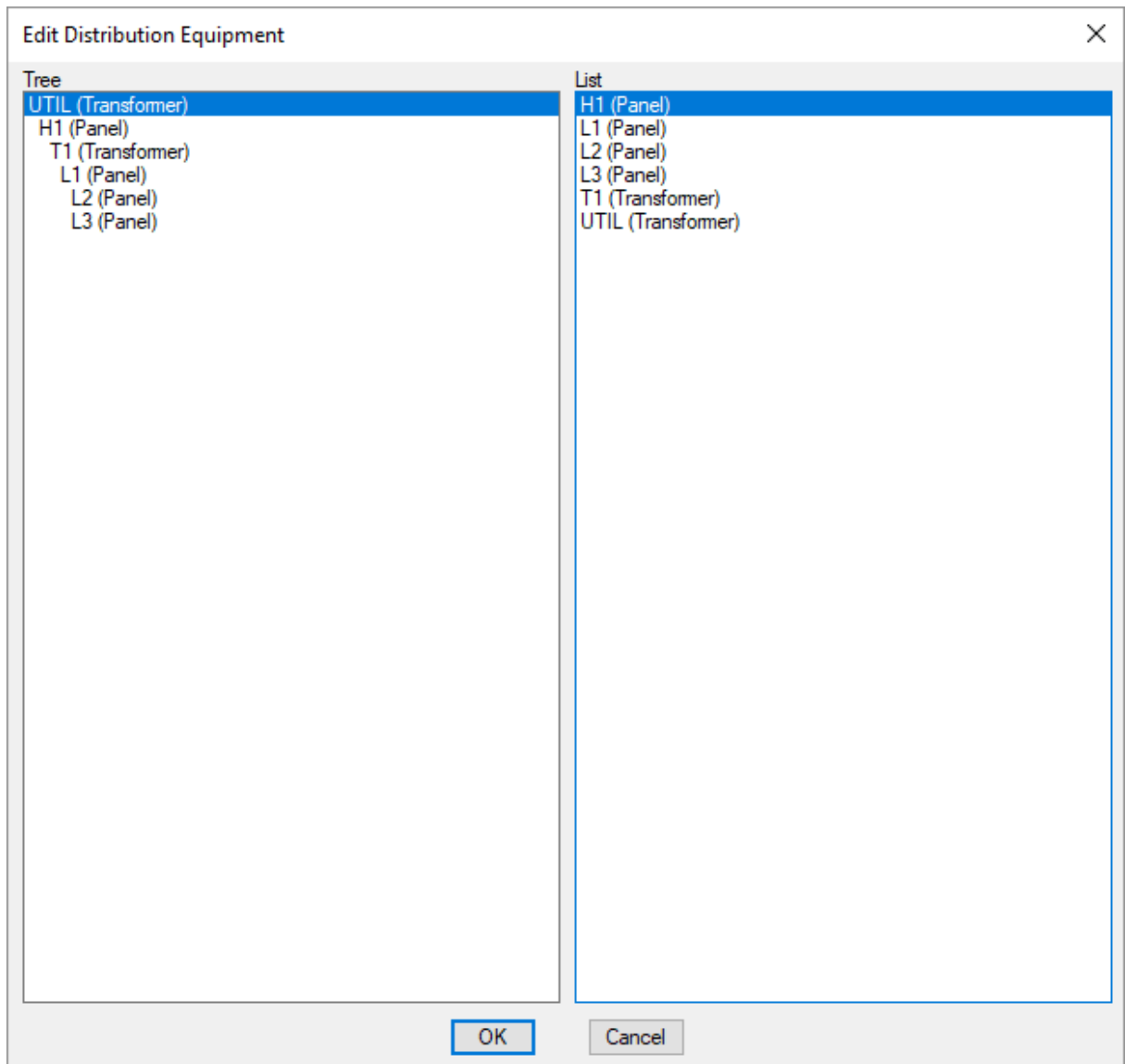
Select a distribution equipment and press the **OK** button. The corresponding distribution equipment dialog box will be opened with that equipment selected. You can then view the equipment and make changes as necessary.

Edit Multiple Distribution Equipment

For each distribution equipment type, there is an `Edit Multiple` command that is used to query or edit multiple distribution equipment of that type.

For instance, to edit or query multiple panels, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Edit Multiple Panels`

Pulldown Menu: `DM Electrical->Distribution Equipment->Edit Multiple->Panels`

You can select multiple distribution equipment in the list using the `SHIFT` or `CTRL` keys.

Select one or more distribution equipment and press the **OK** button.

Select from Drawing: Press this button to select one or more distribution equipment on the drawing. The dialog box will close and you will be prompted to select equipment.

Select objects:

The corresponding distribution equipment dialog box will be opened for the selected equipment. You can then view the equipment and make changes as necessary.

Connect Distribution Equipment in Database

The `Connect Distribution Equipment in Database` command is used to connect distribution equipment to each other without needing to insert them on the drawing.

To connect distribution equipment in the database, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Connect Distribution Equipment in Database`

Pulldown Menu: `DM Electrical->Distribution Equipment->Connect Distribution Equipment in Database`

Distribution Equipment Connections Dialog Box

Step 1: Select the Device to be Connected (Downstream Device)

Begin by selecting the device that is going to be connected in this section of the dialog box. This device is the downstream distribution equipment that is further from the utility. This device is also commonly called a sub-feed panel or a sub-panel.

Generators are not listed in this step. They cannot be connected to other equipment.

Press the **Edit** button to open the **Distribution Equipment Dialog Box** for the selected distribution equipment.

Step 2: Select the Source that Feeds the Device to be Connected (Upstream Device)

Select the device that the first device will be connected to from this section of the dialog box under *Source*. This device is the upstream distribution equipment that is closer to the utility.

Source 1: Make the connection for the first source of the downstream distribution equipment. Available for all distribution equipment types.

Source 2: Make the connection for the second source of the downstream distribution equipment. Available for UPSs and transfer switches.

Source: The upstream distribution equipment that the downstream distribution equipment is connected to.

The **Utility** selection at the top of the *Source* list is a special connection option. It is not a distribution equipment that is created by the user. It represents the utility connection that is beyond the definition of the project. Select this option for the utility transformer or any other device that is connected to the utility.

Edit: Press this button to open the **Distribution Equipment Dialog Box** for the selected distribution equipment.

View Source Device: Press this button to open the **Panel View** dialog box for the currently selected *Source* panel. All of the breakers or taps for the panel and their descriptions will be displayed.

Breaker / Tap: All devices will allow you to select **Breaker / Tap** as the connection type. Select the breaker or tap to connect to using the *Odd Breakers / Taps* or *Even Breakers* lists.

Top Lugs: This selection is available for panels, switchboards, breakers, disconnects, and transfer switches that have their *Lugs* set to **DOUBLE**.

Feedthrough: This selection is available for panels, switchboards, breakers, disconnects, and transfer switches that have their *Lugs* set to **FEEDTHRU**.

Odd Breakers / Taps: For panels, a list of the odd breakers. For all other distribution equipment, a list of all of the taps. The breaker or tap selected in this list will be used for the connection when the **Step 3** button is pressed. The top blank entry will be selected for panels when an even breaker is used for the connection.

Even Breakers: For panels, a list of the even breakers. Not used for all other distribution equipment. The breaker selected in this list will be used for the connection when the **Step 3** button is pressed. The top blank entry will be selected for panels when an odd breaker is used for the connection.

Change Number of Poles: Press this button to open the **Change Number of Poles** dialog box and change the number of poles of the currently selected circuit. This dialog box can be used to convert two-pole and three-pole circuits to multiple single-pole circuits. See the **Change Number of Poles** section for more information.

Set Connection Layer System: Press this button to change the layer system of the selected breaker or tap. See the **Layer System** section for more information.

Step 3: Connect the Device

After you have selected the two devices to connect and configured the source device connection, press the **Step 3** button to make the connection.

Panels

To create a panel, go to

Ribbon: `DM Electrical->Distribution Equipment->` PAN `Panels`

Pulldown Menu: `DM Electrical->Distribution Equipment->Panels`

Panel Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Switchboards

To create a switchboard, go to

Ribbon: `DM Electrical->Distribution Equipment->` SWBD `Switchboards`

Pulldown Menu: `DM Electrical->Distribution Equipment->Switchboards`

Switchboard Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Transformers

To create a transformer, go to

Ribbon: `DM Electrical->Distribution Equipment->` XFMR `Transformers`

Pulldown Menu: `DM Electrical->Distribution Equipment->Transformers`

Transformer Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

KVA: The kVA rating of the transformer.

K-Rated: Whether the transformer is K-rated. The value of this setting can be displayed on the distribution equipment schedule. It is for informational purposes only and is not otherwise used.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Wireways

To create a wireway, go to

Ribbon: DM Electrical->Distribution Equipment-> WIRE Wireways

Pulldown Menu: DM Electrical->Distribution Equipment->Wireways

Wireway Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Bus Gutters / Bus Ducts

To create a bus gutter or a bus duct, go to

Ribbon: DM Electrical->Distribution Equipment-> ᴮᵁˢ Bus Gutter / Bus Duct

Pulldown Menu: DM Electrical->Distribution Equipment->Bus Gutter / Bus Duct

Bus Gutter / Bus Duct Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Enclosed Breakers / Disconnects

To create an enclosed breaker or a disconnect, go to

Ribbon: `DM Electrical->Distribution Equipment->` BKR `Enclosed Breakers / Disconnects`

Pulldown Menu: `DM Electrical->Distribution Equipment->Enclosed Breakers / Disconnects`

Enclosed Breaker / Disconnect Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

Type: The type of breaker or disconnect.
- **Enclosed Breaker**
- **Disconnect**
- **Fused Disconnect**

Frame Size: The physical size of the breaker or disconnect.

Breaker Trip/Fuse Rating: The overcurrent protection size of the breaker or disconnect.

Set Curve: Press this button to configure the curve for the disconnect. The **Select Curve** dialog box will appear.

Enclosure: The NEMA rating for the breaker or disconnect enclosure.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Uninterruptible Power Supplies (UPS) / Phase Inverters

To create an uninterruptible power supply (UPS) or a phase inverter, go to

Ribbon: `DM Electrical->Distribution Equipment->` UPS `UPSs / Phase Inverters`

Pulldown Menu: `DM Electrical->Distribution Equipment->UPSs / Phase Inverters`

UPS / Phase Inverter Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next**, **New**, **Copy**, **Save**, and **Delete** buttons.

The **Input KVA**, **Efficiency**, **Output KVA**, **Output KW**, and **Power Factor** values are all related to each other. Use the radio buttons to choose one value from each column to calculate. The other values must be entered manually.

Input KVA: The input kVA of the UPS. If this is calculated, it is equal to **Output KVA / Efficiency**.

Efficiency: The efficiency of the UPS. If this is calculated, it is equal to **Input KVA / Output KVA**.

Output KVA: The output kVA of the UPS. If this is calculated, it is equal to **Input KVA * Efficiency**.

Output KW: The output of the UPS in kW. If this is calculated, it is equal to **Output KVA * Power Factor**.

Power Factor: The power factor of the output of the UPS. If this is calculated, it is equal to **Output KVA / Output KW**.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Motor Control Centers

To create a motor control center, go to

Ribbon: `DM Electrical->Distribution Equipment->` MCC `Motor Control Centers`

Pulldown Menu: `DM Electrical->Distribution Equipment->Motor Control Centers`

Motor Control Center Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next**, **New**, **Copy**, **Save**, and **Delete** buttons.

Horizontal Bus Amps: The size of the horizontal bus, in amps, feeding the vertical buses.

Number of Sections: The number of sections in the motor control center.

Section Height: The height of each motor control center section, in inches.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Meter Centers

To create a meter center, go to

Ribbon: DM Electrical->Distribution Equipment-> MC Meter Centers

Pulldown Menu: `DM Electrical->Distribution Equipment->Meter Centers`

Meter Center Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Generators

To create a generator, go to

Ribbon: `DM Electrical->Distribution Equipment->` GEN `Generators`

Pulldown Menu: `DM Electrical->Distribution Equipment->Generators`

Generator Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

The **KVA**, **KW**, and **Power Factor** values are all related to each other. Use the radio button to choose which value you want calculated. You must enter the other values manually.

KVA: The output of the generator, in kVA. If this is calculated, it is equal to **KW / Power Factor**.

KW: The output of the generator, in kW. If this is calculated, it is equal to **KVA * Power Factor**.

Power Factor: The power factor of the generator. If this is calculated, it is equal to **KVA / KW**.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.
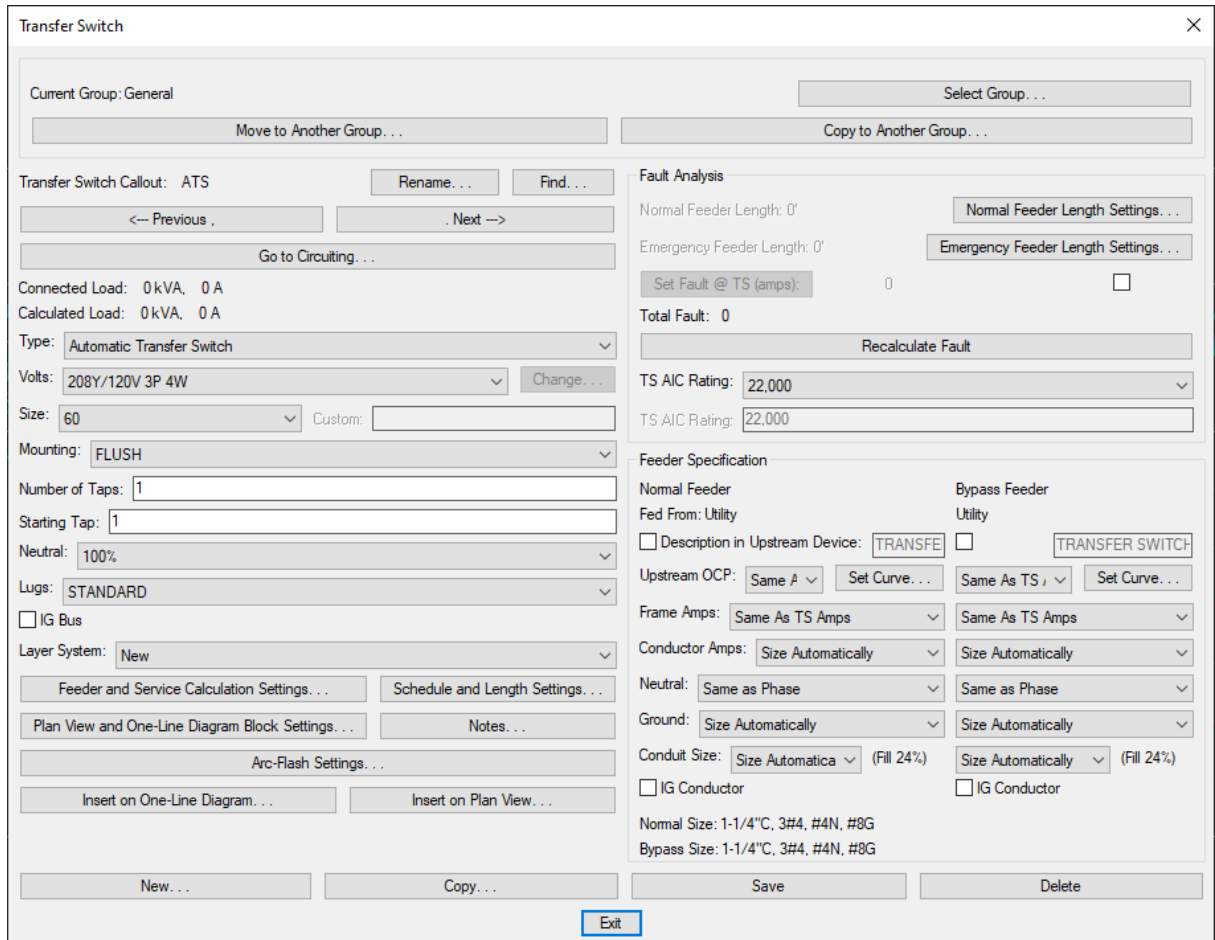
Transfer Switches

To create a transfer switch, go to

Ribbon: `DM Electrical->Distribution Equipment->` TS `Transfer Switches`

Pulldown Menu: `DM Electrical->Distribution Equipment->Transfer Switches`

Transfer Switch Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New, Copy, Save**, and **Delete** buttons.

Type: The type of transfer switch.
- **Automatic Transfer Switch**
- **Manual Transfer Switch**

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Low Voltage Panels

To create a low voltage panel, go to

Ribbon: `DM Electrical->Distribution Equipment->` LVD `Low Voltage Panels`

Low Voltage Panel Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next**, **New**, **Copy**, **Save**, and **Delete** buttons.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Recurring Panel Templates

Recurring panel templates allow you to create multiple panels with the same settings. This can be useful when planning hotels, apartments, and other buildings in which multiple panels with the same specifications and connections will be used.

If the settings are changed for a recurring panel template, all recurring panel instances will change.

To create a recurring panel template, go to

Ribbon: `DM Electrical->Distribution Equipment->` RPAN `Recurring Panel Templates`

Pulldown Menu: `DM Electrical->Distribution Equipment->Recurring Panel Templates`

Recurring Panel Template Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next**, **New**, **Copy**, **Save**, and **Delete** buttons.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Create Recurring Panel Instance

To create a recurring panel instance, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Create Recurring Panel Instance`

Pulldown Menu: `DM Electrical->Distribution Equipment->Recurring Panel Instances->Create`

Create Recurring Panel Instance Dialog Box

Callout: Name of the recurring panel instance. The callout is used to label the instance on the drawing and in the recurring panel instance schedule.

Recurring Panel Template: The template the recurring panel instance will follow.

Some values in this dialog box, such as *Feeder Length Calculation Method* and the **Feeder Specification** section, are dependent upon the recurring panel template selected. To change these values, check the corresponding box. Checked values will not change if you change the corresponding template.

Elevation: The elevation of the recurring panel instance relative to the alignment point when the instance is inserted on the drawing. See the **Elevation** section for more information.

Distance from Distribution Equipment: The distance the clearance extends in front of the distribution equipment, in inches.

Distance to Left: The distance the clearance extends to the left of the distribution equipment, in inches. Use this field if the clearance is required to be wider than the equipment.

Distance to Right: The distance the clearance extends to the right of the distribution equipment, in inches. Use this field if the clearance is required to be wider than the equipment.

Distance above Distribution Equipment: The distance the clearance extends above the distribution equipment, in inches. This distance is not displayed on the drawing. It is used when the equipment is exported to an IFC file.

See the **Distribution Equipment Dialog Box Settings** section for more information about the other values and buttons in this dialog box.

Create Recurring Panel Instance on Drawing

To add the recurring panel instance to the drawing, press the **Create on Drawing** button.

You will be prompted to insert the instance on the drawing.

Specify insertion point:

After you specify the location for the instance, you will be prompted for the rotation angle.

Rotation angle <0>:

After you specify the rotation angle, you will be prompted to insert the callout. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the callout.

Create Recurring Panel Instance in Database

To add the recurring panel instance in the database, press the **Create in Database** button.

The equipment will be added to the database and the following dialog box will appear:
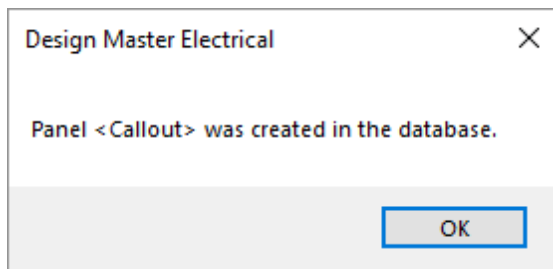
Query or Edit Multiple Recurring Panel Instances

To query or edit multiple recurring panel instances, go to

Ribbon: `DM Electrical->Distribution Equipment->` [RPAN EM] `Query or Edit Multiple Recurring Panel Instances`

Pulldown Menu: `DM Electrical->Distribution Equipment->Recurring Panel Instances->Query or Edit Multiple`

You can select multiple recurring panel instances in the list using the `SHIFT` or `CTRL` keys.

Select one or more recurring panel instances and press the **OK** button.

Select from Drawing: Press this button to select one or more recurring panel instances on the drawing. The dialog box will close and you will be prompted to select instances.

Select recurring panel instances to edit:

Select objects:

The **Edit Multiple Recurring Panel Instances** dialog box will be opened with those instances selected. You can then view the instances and make changes as necessary.

Insert Recurring Panel Instance on this Drawing

To insert a recurring panel instance on a drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` ⬛ `Insert Recurring Panel Instance on this Drawing`

Pulldown Menu: `DM Electrical->Distribution Equipment->Recurring Panel Instances->Insert on this Drawing`

Inserting a Recurring Panel Instance

Select the recurring panel instance to be inserted and press the **OK** button.

You will be prompted to specify the location on the drawing where the instance will be inserted.

Specify insertion point:

After you specify the location for the instance, you will be prompted for the rotation angle.

Specify rotation angle:

After you specify the rotation angle, you will be prompted to insert the callout. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the callout.

Copying and Erasing Recurring Panel Instance Plan View Blocks

A recurring panel instance shown on the drawing can be copied using the standard CAD `COPY` command.

Using the standard CAD `ERASE` command removes the block and deletes the recurring panel instance from the database. To remove an instance from the drawing without deleting it from the database, use the **Remove Recurring Panel Instance from this Drawing** command.

Remove Recurring Panel Instance from this Drawing

To remove a recurring panel instance from a drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` 🔄 `Remove Recurring Panel Instance from this Drawing`

Pulldown Menu: `DM Electrical->Distribution Equipment->Recurring Panel Instances->Remove from this Drawing`

You will be prompted to select the recurring panel instance to remove from the drawing.

Select recurring panel instance to remove from drawing:

After you select the instance, it will be removed from the drawing but remain in the database.

Delete Recurring Panel Instance from Database

To delete a recurring panel instance from the project database, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Delete Recurring Panel Instance from Database`

Pulldown Menu: `DM Electrical->Distribution Equipment->Recurring Panel Instances->Delete from Database`

Select a recurring panel instance from the list. You can select multiple items in the list using the `SHIFT` or `CTRL` keys. Press the **Delete** button to delete the selected instances from the database.

Recurring panel instances that have been placed on a drawing will not appear in the dialog box. Removing an instance from a drawing using the standard CAD `ERASE` command will also remove it from the database.

Convert Panel to Recurring Panel Template

The `Convert Panel to Recurring Panel Template` command is used to create a recurring panel template from an existing panel. This can be useful when you need multiple panels with the same specifications and connections as a panel you have already created.

To convert a panel to a recurring panel template, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Convert Panel to Recurring Panel Template`

Pulldown Menu: `DM Electrical->Distribution Equipment->Recurring Panel Instances->Convert Panel to Recurring Panel Template`

Select a panel from the list. Press the **OK** button to convert the panel to a recurring panel template.

Insert Panel List

A list of panels in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` PAN `Insert Panel List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Panel`

Insert Panel List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Panel List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Insert Switchboard List

A list of switchboards in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` 🔲 `Insert Switchboard List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Switchboard`

Insert Switchboard List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Switchboard List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Insert Transformer List

A list of transformers in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` ⊞ `Insert Transformer List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Transformer`

Insert Transformer List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Transformer List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Insert Wireway List

A list of wireways in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` WIRE `Insert Wireway List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Wireway`

Insert Wireway List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been

inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Wireway List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

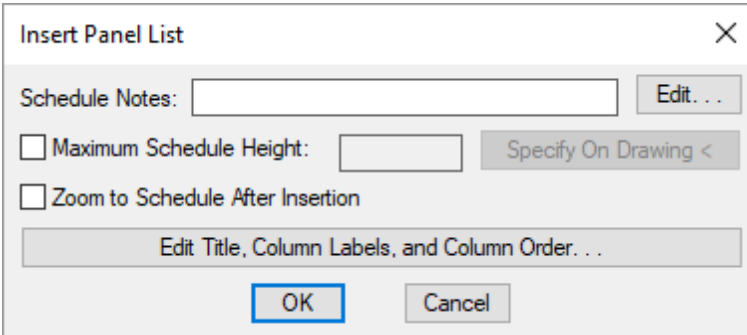# Insert Bus Gutter / Bus Duct List

A list of bus gutters and bus ducts in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Insert Bus Gutter / Bus Duct List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Bus Gutter / Bus Duct`

Insert Bus Gutter / Bus Duct List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Bus Gutter / Bus Duct List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

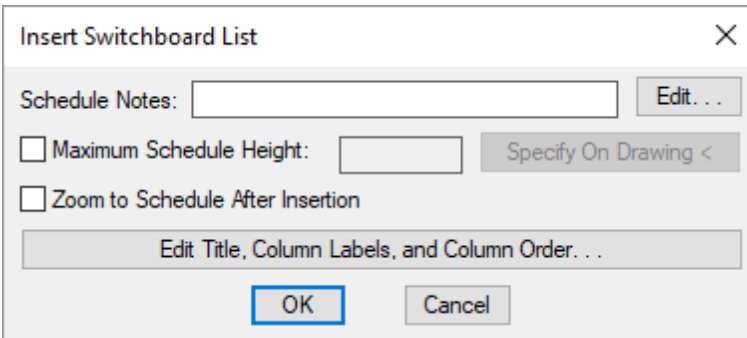# Insert Enclosed Breaker / Disconnect List

A list of enclosed breakers and disconnects in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` BKR `Insert Enclosed Breaker / Disconnect List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Enclosed Breaker / Disconnect`

Insert Enclosed Breaker / Disconnect List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Enclosed Breaker / Disconnect List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Insert UPS / Phase Inverter List

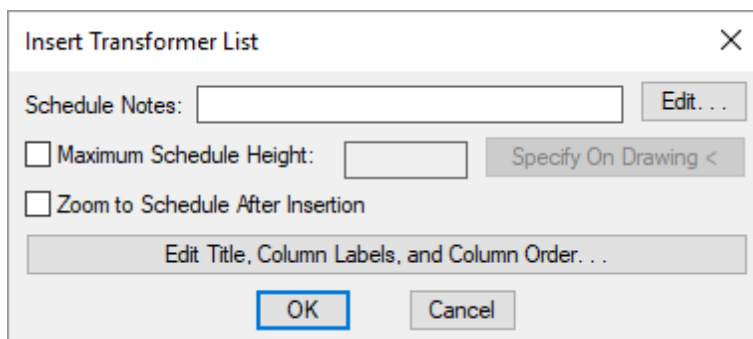A list of UPSs and phase inverters in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` 🔲 `Insert UPS / Phase Inverter List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->UPS / Phase Inverter`

Insert UPS / Phase Inverter List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line

straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit UPS / Phase Inverter List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Insert Motor Control Center List

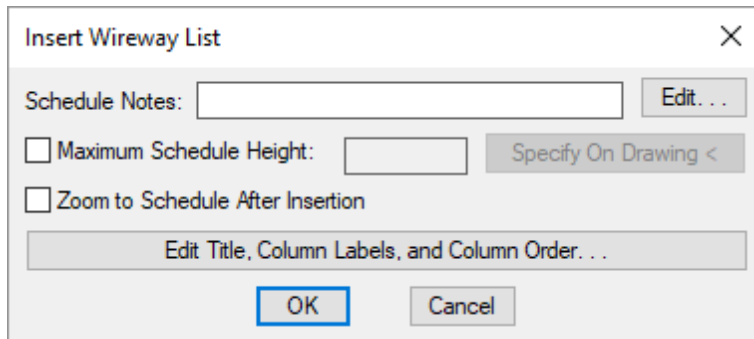A list of motor control centers in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` MCC `Insert Motor Control Center List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Motor Control Center`

Insert Motor Control Center List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Motor Control Center List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.
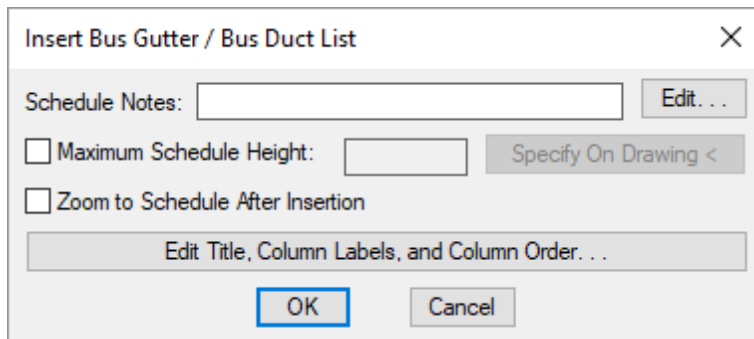
Insert Meter Center List

A list of meter centers in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` MC `Insert Meter Center List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Meter Center`

Insert Meter Center List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Meter Center List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Insert Generator List
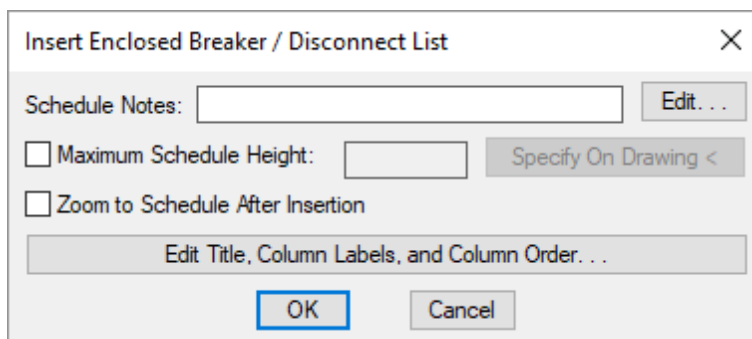
A list of generators in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` GEN `Insert Generator List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Generator`

Insert Generator List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line

straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Generator List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.
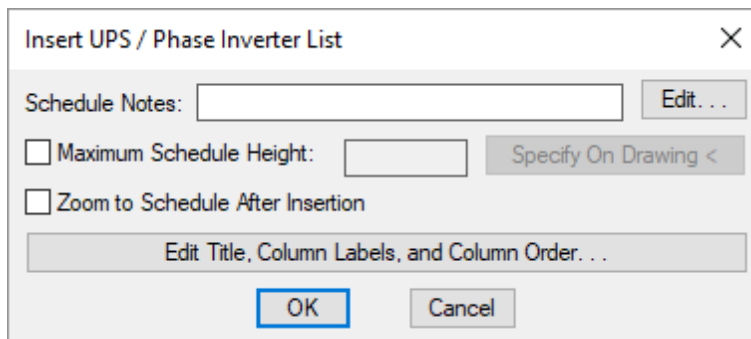
Insert Transfer Switch List

A list of transfer switches in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` T$ `Insert Transfer Switch List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Transfer Switch`

Insert Transfer Switch List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Transfer Switch List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.
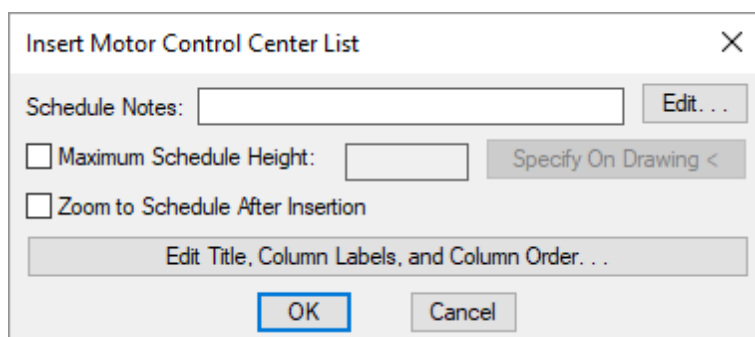
Insert Low Voltage Panel List
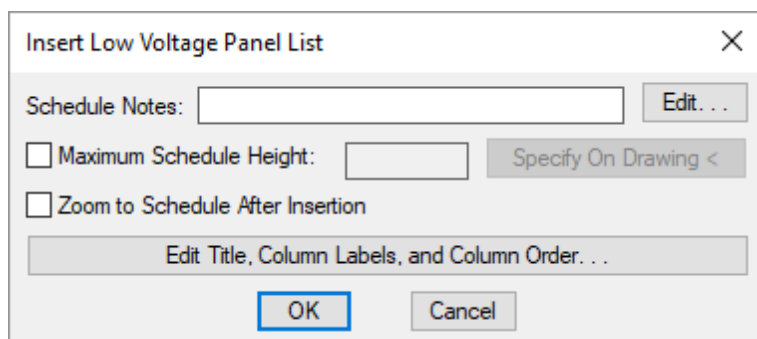
A list of low voltage panels in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Insert Low Voltage Panel List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Low Voltage Panel`

Insert Low Voltage Panel List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Low Voltage Panel List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.
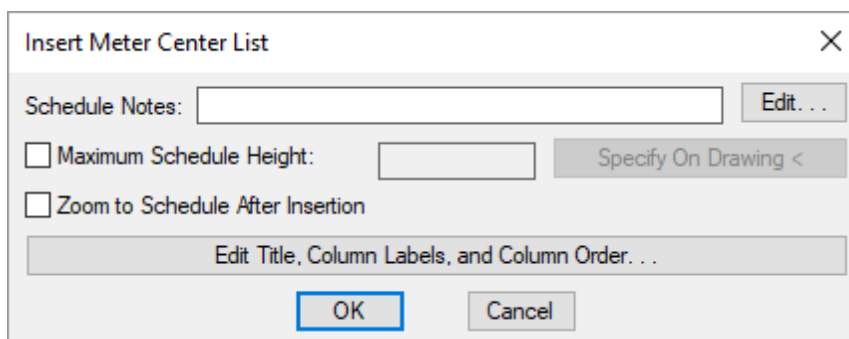
Insert Recurring Panel Template List

A list of recurring panel templates in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: DM Electrical->Distribution Equipment-> ⊞ Insert Recurring Panel Template List

Pulldown Menu: DM Electrical->Distribution Equipment->Insert List->Recurring Panel Template

Insert Recurring Panel Template List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Recurring Panel Template List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

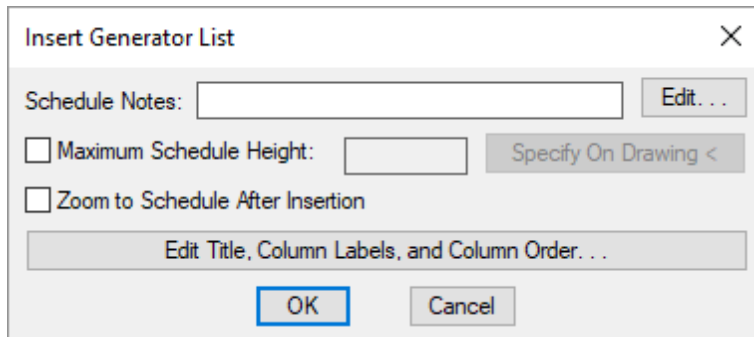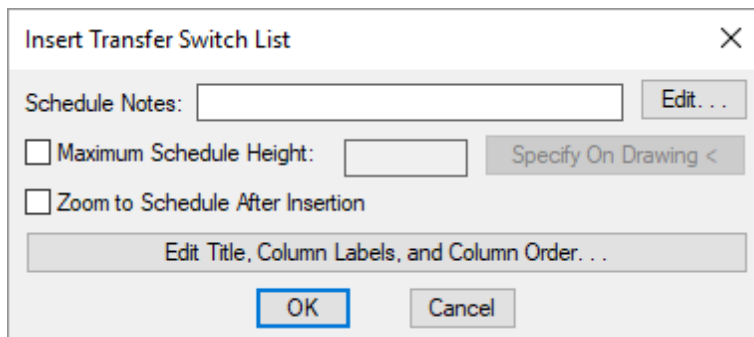# Insert Recurring Panel Instance List

A list of recurring panel instances in the current project can be inserted on the drawing.

To insert this list onto the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Insert Recurring Panel Instance List`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert List->Recurring Panel Instance`

Insert Recurring Panel Instance List Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Recurring Panel Instance List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Insert Plan View Block

The `Insert Plan View Block` command is used to locate distribution equipment on the drawing. The location is used to calculate the length of the feeders between distribution equipment. The feeder length is used in the fault current calculation, voltage drop calculation, and takeoffs.

To insert distribution equipment on a drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` Insert Plan View Block

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert Plan View Block`

Insert Distribution Equipment Block Dialog Box

Insert Distribution Equipment Block

H1   (Panel)
L1   (Panel)
L2   (Panel)
L3   (Panel)
T1   (Transformer)
UTIL   (Transformer)

☐ Select Block based upon KVA

Plan View Block: 8" x 42" Panel

Custom Size: Depth (in): [   ]   Width (in): [   ]   Height (in): [   ]

Elevation

Elevation (ft-in): [0]

⦿ Relative to alignment point area (easier)

◯ Relative to ground (necessary for automatic riser generation before distribution equipment is inserted on plans)

Clearance

Distance from Distribution Equipment (in): [36]

Distance to Left (in): [0]

Distance to Right (in): [0]

Distance above Distribution Equipment (in): [0]

Layer System: New

[OK]   [Cancel]

See the **Plan View and One-Line Diagram Block Settings** section for more information about the settings in this dialog box.

Distance from Distribution Equipment: The distance the clearance extends in front of the distribution equipment, in inches.

Distance to Left: The distance the clearance extends to the left of the distribution equipment, in inches. Use this field if the clearance is required to be wider than the equipment.

Distance to Right: The distance the clearance extends to the right of the distribution equipment, in inches. Use this field if the clearance is required to be wider than the equipment.

Distance above Distribution Equipment: The distance the clearance extends above the distribution equipment, in inches. This distance is not displayed on the drawing. It is used when the equipment is exported to an IFC file.

Inserting a Distribution Equipment Plan View Block

Select the distribution equipment block to be inserted and press the **OK** button.

You will be prompted to specify the location on the drawing where the block will be inserted.

Specify insertion point:

After you specify the location for the distribution equipment, you will be prompted for the rotation angle.

Specify rotation angle:

After you specify the rotation angle, you will be prompted to insert the callout. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the callout.

Copying and Erasing Distribution Equipment Plan View Blocks

A distribution equipment shown on the drawing cannot be copied using standard CAD commands. If you attempt to do so, the copied device will automatically be erased. This will occur when using both the standard CAD `COPY` command and the `MIRROR` command. If you want to copy a distribution equipment, use the **Copy** button on the appropriate distribution equipment dialog box.

Similarly, erasing a distribution equipment from the drawing only removes the block. It does not delete the entire device from the database. To erase a distribution equipment, use the **Delete** button on the appropriate distribution equipment dialog box.

Related Options

Insert 3D blocks on drawing: Sets whether a 3D block is inserted on the drawing along with the 2D block.

Insert Plan View Blocks in Another Area or Drawing

The `Insert Plan View Blocks in Another Area or Drawing` command is used to place distribution equipment that is already inserted in an alignment point area into another alignment point area. The second alignment point area can be on the same or a different drawing. When the distribution equipment is modified in one area, it is automatically modified in the other area as well.

This feature is commonly used when placing distribution equipment on lighting and power plans.

To insert the distribution equipment plan view blocks in another area or drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` ⟳ `Insert Plan View Blocks in Another Area or Drawing`

Pulldown Menu: `DM Electrical->Distribution Equipment->Insert Plan View Blocks in Another Area or Drawing`

Insert Distribution Equipment Blocks in Another Area or Drawing Dialog Box
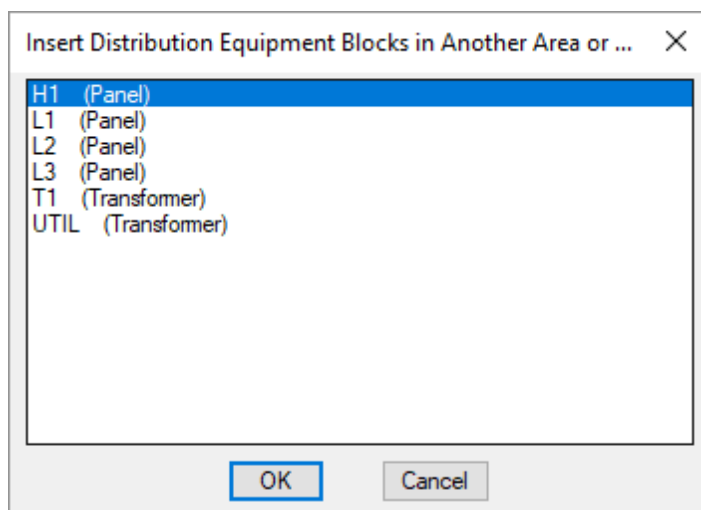
Select the devices to place in a new alignment point area and press the **OK** button. You can select multiple items in the list using the `SHIFT` or `CTRL` keys.

If there is more than one alignment point area in your drawing, you will be prompted to select which area to use.

Specify point in alignment point area to use:

The distribution equipment will be inserted in the area you select. If there is only one alignment point area in your drawing, the distribution equipment will be inserted in that area with no further input from you.

The distribution equipment are located on the drawing relative to the alignment point. If they do not appear to be in the right location, first check that your alignment points are all in the same location on all your drawings.

Distribution Equipment Schedules

A number of schedules can be inserted on the drawing to document your design.

Schedules are not updated automatically. When you update your design, you need to manually update your schedules. Use the **Update All Schedules and One-Line Diagram** command to update all of the schedules on the current drawing.

Training Videos

- **[Breaker Curves](#)**

Insert Distribution Equipment Schedule

A distribution equipment schedule lists the description, load total summary, and individual circuits and taps for a specific distribution equipment.

To place a distribution equipment schedule on a drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` 📊 `Insert Distribution Equipment Schedule`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Insert Distribution Equipment Schedule`

Insert Distribution Equipment Schedule Dialog Box
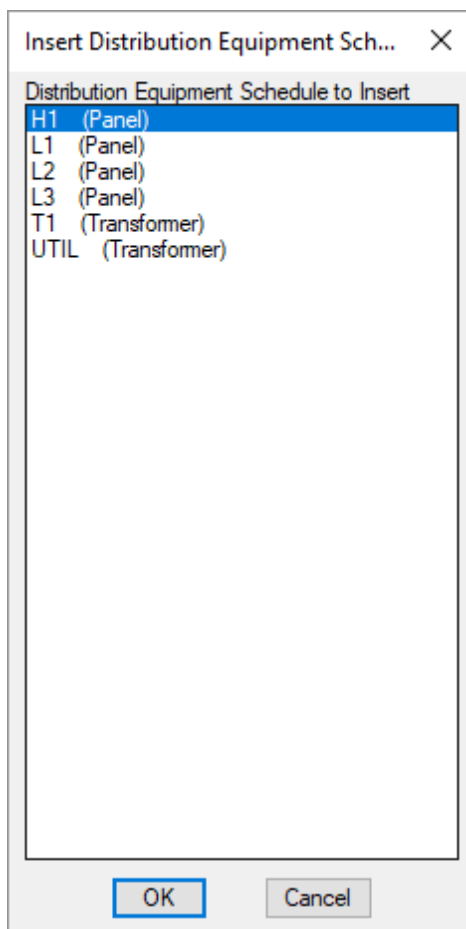
Distribution Equipment Schedule to Insert: A list of all of the distribution equipment in the database.

Select the schedule to insert and press the **OK** button.

Inserting Distribution Equipment Schedules on the Drawing

You will be prompted to select the insertion point for the distribution equipment schedule.

Insertion point for <distribution equipment name> / <Cancel>:

After you specify the location for the schedule, you will be prompted to insert the schedule for the next distribution equipment in the list.

Insertion point for <distribution equipment name> / <End>:

Continue inserting distribution equipment on the drawing, or press **ENTER** to finish the command.

The load totals for the distribution equipment are not updated when the schedules are inserted. To calculate these values and update the schedules, use the **Update Distribution Equipment Schedule** command.

Related Options

Panel schedule load display units: Sets the units used to display loads on panel schedules.

Show fixed number of decimals in panel schedules: Whether all values for loads are displayed with a set number of decimal places.

This option can be further customized with the **Number of decimals to display in panel schedule** option.

Warn if distribution equipment is overloaded: Sets whether a warning dialog box appears while inserting schedules if you have distribution equipment that is overloaded.

This option can be further customized with the **Distribution equipment overloaded threshold (% of panel capacity)** option.

Unused circuit description: Sets how unused circuits are labeled on the schedule.

Include room names in circuit descriptions: Sets whether room names are included in circuit descriptions on the schedule.

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Update Distribution Equipment Schedule
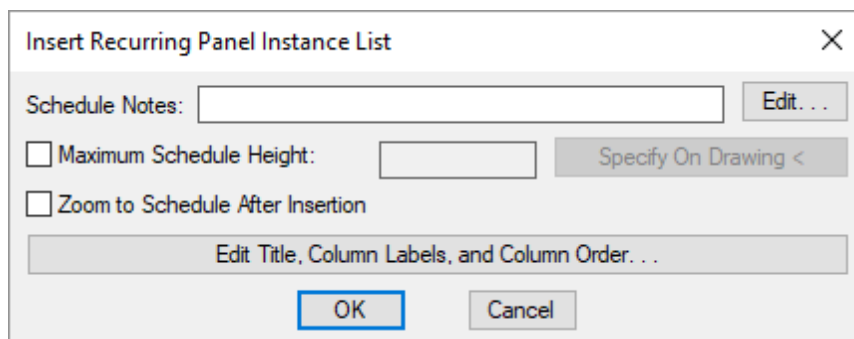
Design Master Electrical does not automatically update distribution equipment schedules when changes are made to the drawing. You must manually update the schedules before plotting to ensure the schedules reflect the most recent changes.

To update a distribution equipment schedule, go to

Ribbon: `DM Electrical->Distribution Equipment->` 🔃 `Update Distribution Equipment Schedule`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Update Distribution Equipment Schedule`

You will be prompted to identify which schedules to update.

Select distribution equipment schedules to update:

Select all of the schedules that need to be updated and press **ENTER**.

The dialog box below will appear, asking "Recalculate loads between distribution equipment?"
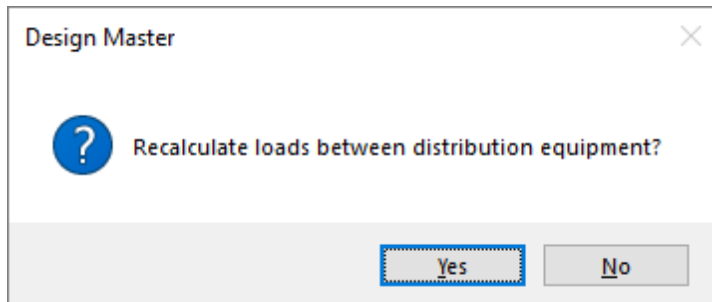
Yes: All of the values on the distribution equipment schedules will be recalculated to reflect the latest information. All of the load totals for the equipment, including loads transferred from distribution equipment connected to it, are recalculated.

No: The values from the most recent calculation will be used. This selection allows you to save time if you know that you recently ran these calculations and have not made any changes that may affect them.

After you answer the prompt on the dialog box, all of the distribution equipment schedules you selected will be updated.

Insert Feeder Schedule

To insert or update a feeder schedule, go to

Ribbon: `DM Electrical->Distribution Equipment->` 📋 `Insert Feeder Schedule`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Insert Feeder Schedule`

Edit Feeder ID Group Settings Dialog Box

The **Edit Feeder ID Group Settings** dialog box is used to set options for the feeder schedule. It is also used to set options for feeder ID blocks when they appear on the one-line diagram.

Include "Sizing Option" in schedule note: Inserts a note displaying the sizing method for the feeders that will appear below the schedule when inserted on the drawing. See the **Wire Sizing** section for more information.

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Block Can Rotate: Whether feeder ID blocks rotate when placed on feeders, depending upon the insertion point along the feeder.

Feeder ID Block: Sets the block that is displayed for feeder IDs. Press the **Browse** button to select a new block.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the

drawing. See the **Edit Feeder Schedule List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Include equipment feeders in feeder schedule: Sets whether feeders to equipment connections are automatically included on the feeder schedule.

Display neutral wire count separately from phase wire count: Sets how the neutral wire is displayed in wire callouts.

Conduit location: Sets how the conduit location is displayed in wire callouts.

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Insert Fault Schedule
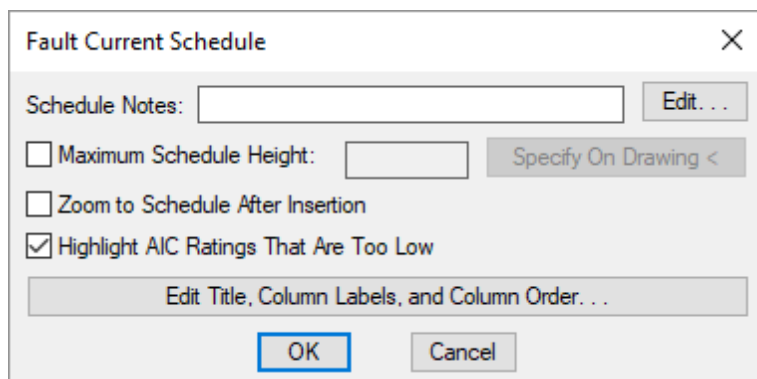
The `Insert Fault Schedule` command is used to calculate and display the fault current at all distribution equipment. See the **Fault Calculation Methods** section for information about how fault is calculated.

To insert or update the fault current schedule, go to

Ribbon: `DM Electrical->Distribution Equipment->` 🔲 `Insert Fault Schedule`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Insert Fault Schedule`

Fault Current Schedule Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Fault Schedule List** section for more information.

Highlight AIC Ratings That Are Too Low: Whether AIC ratings in the schedules that are less than the calculated fault for a device will be highlighted.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Errors

If there are errors in the distribution equipment definitions, an alert dialog box will appear and the errors will be shown at the command line. For more information about fixing these errors, see the **Addressing Distribution Equipment Errors** section.

Related Options

Fault Schedule: Highlight AIC ratings that are too low: Sets whether the *Highlight AIC Ratings That Are Too Low* box is checked by default.

Extra distance added to feeder length: Sets additional length to be added to feeders for connections.

This option can be further customized with the **Ignore extra distance on feeders shorter than this** option.

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Insert Voltage Drop Schedule

The `Voltage Drop Schedule` command is used to calculate and display the voltage drop on the feeders and branch circuits in the project. See the **Voltage Drop Calculation Method** section for information about how voltage drop is calculated.

To insert the voltage drop schedule, go to

Ribbon: `DM Electrical->Distribution Equipment->` ▦ `Insert Voltage Drop Schedule`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Insert Voltage Drop Schedule`

Voltage Drop Schedule Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Highlight Voltage Drops That Are Too High: Whether voltage drops that are too high are highlighted, and the threshold to be considered too high.
- **Yes, Feeder or Branch Circuit > 3%, Total > 5%:** Feeder voltage drops that exceed 3%, branch circuit voltage drops that exceed 3%, and total voltage drops that exceed 5% will be highlighted. These values are based upon *NEC 210.19 Informational Note* and *NEC 215.2(A)(2) Informational Note No. 2*.
- **Yes, Feeder > 2%, Branch Circuit > 3%:** Feeder voltage drops that exceed 2% and branch circuit voltage drops that exceed 3% will be highlighted.
- **No:** Voltage drops that are too high will not be highlighted.

Circuits to Display: The branch circuit with the highest voltage drop will be shown for every distribution equipment. You also can choose to display additional branch circuit voltage drops.
- **None:** No additional branch circuit voltage drops will be displayed.
- **All:** All branch circuit voltage drops will be displayed.
- **All with load:** All voltage drops for branch circuits with loads will be displayed.
- **All with voltage drop > 0:** All branch circuit voltage drops that exceed 0 will be displayed.
- **All with homerun drawn:** All voltage drops for branch circuits with homeruns drawn will be displayed.
- **Too high:** Displays branch circuits where the branch circuit voltage drop exceeds 3% or the total voltage drop exceeds 5%.

Feeder Voltage Drop: Whether voltage drop on feeders will be calculated based upon the actual load.
- **Based upon actual load:** Voltage drop will be calculated based upon the loads specified for each piece of distribution equipment.
- **Based upon 80% of panel capacity:** Voltage drop will be calculated based upon 80% of the capacity for each piece of distribution equipment.

Warn if Circuit Voltage Drop Cannot be Calculated: Whether you receive a warning if the branch circuit voltage drop cannot be calculated correctly due to circuiting errors.

If this is checked and any errors are found, an alert dialog box will appear and the errors will be shown at the command line. The dialog box will also ask if you want to continue receiving this warning. Press the **Yes** button to keep the setting checked the next time the **Voltage Drop Schedule** dialog box is opened. Press the **No** button to uncheck the setting.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Voltage Drop Schedule List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Errors

If there are errors in the distribution equipment definitions, an alert dialog box will appear and the errors will be shown at the command line. For more information about fixing these errors, see the **Addressing Distribution Equipment Errors** section.

Related Options

Voltage Drop Schedule: Highlight voltage drops: Sets the default for the *Highlight Voltage Drops That Are Too High* setting.

Voltage Drop Schedule: Circuits to display: Sets the default for the *Circuits to Display* setting.

Voltage Drop Schedule: Warn if branch circuit voltage drop cannot be calculated: Sets whether the *Warn if Circuit Voltage Drop Cannot be Calculated* box is checked by default.

Voltage drop and transformers: Sets how voltage drop is calculated through transformers.

Calculate voltage drop based upon 80% of panel capacity: Sets the default for the *Feeder Voltage Drop* setting.

Feeder make-up: Sets additional length to be added to feeders for connections.

This option can be further customized with the **Ignore make-up on feeders shorter than this** option.

Branch circuit make-up: Sets additional length to be added to branch circuits for connections.

This option can be further customized with the **Add branch circuit make-up for each device** option.

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.
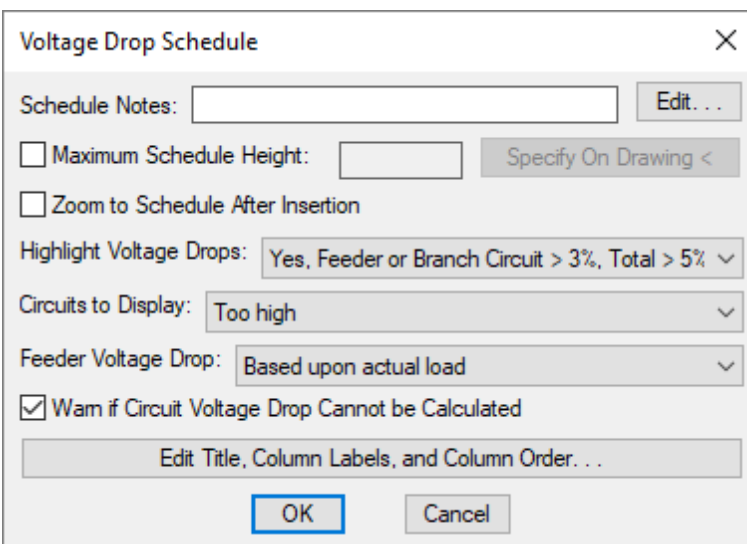
Insert Arc-Flash Schedule

The `Insert Arc-Flash Schedule` command is used to insert a schedule on the drawing that displays the
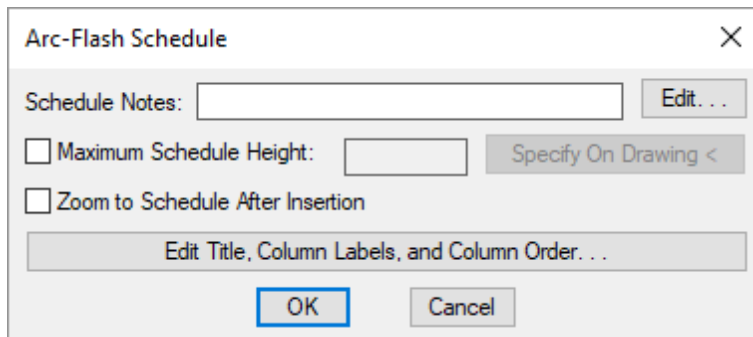
results of the arc-flash calculations. You can set the schedule to include all of the information in the **Arc-Flash Settings** dialog box. The default schedule displays only the most relevant fields.

To insert or update the arc-flash schedule, go to

Ribbon: `DM Electrical->Distribution Equipment->` ⬛ `Insert Arc-Flash Schedule`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Insert Arc-Flash Schedule`

Arc-Flash Schedule Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Arc-Flash Schedule List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Errors

If there are errors in the distribution equipment definitions, an alert dialog box will appear and the errors will be shown at the command line. For more information about fixing these errors, see the **Addressing Distribution Equipment Errors** section.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Insert Selective Coordination Graph

The `Insert Selective Coordination Graph` command is used to create coordination graphs for breakers and fuses on the distribution equipment and equipment connections in your project. Distribution equipment curves are defined in the **General Settings** and **Feeder Specification** settings of the distribution equipment dialog box. Equipment connection curves are defined in the **Create Equipment** dialog box.

To insert a selective coordination graph, go to

Ribbon: `DM Electrical->Distribution Equipment->` ⌐ `Insert Selective Coordination Graph`

Insert Selective Coordination Graph Dialog Box

Title: The title to be used for the graph.

Subtitle: A subtitle to be added below the title for the graph.

Note: A note that will appear below the graph when it is inserted on the drawing.

Curves: A list of the curves to be displayed in the graph.

Edit Curve Settings: Press this button to edit the selected curve.

For distribution equipment and equipment connection curves, the **Select Curve** dialog box will appear.

For custom curves, the **Add Custom Curve** dialog box will appear.

Edit Equipment Settings: Press this button to open the **Distribution Equipment Dialog Box** for the selected

distribution equipment, or the **Edit Equipment** dialog box for the selected equipment connection.

Move Up: Press this button to move the selected row up in the list of curves.

Move Down: Press this button to move the selected row down in the list of curves.

Remove Curve: Press this button to remove the selected row from the list of curves.

Add Distribution Equipment Curve: Press this button to add a distribution equipment curve to the list of curves.

If the selected distribution equipment has curves defined for the feeder and main disconnect, you will be asked to specify which curve to add.
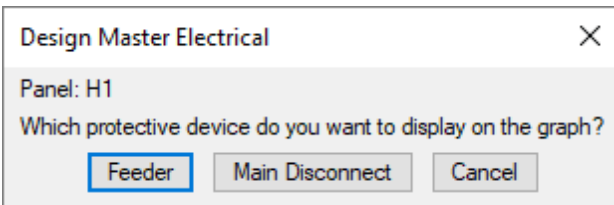
Add Equipment Connection Curve: Press this button to add an equipment connection curve to the list of curves.

If the selected equipment connection has curves defined for the circuit and equipment disconnect, you will be asked to specify which curve to add.
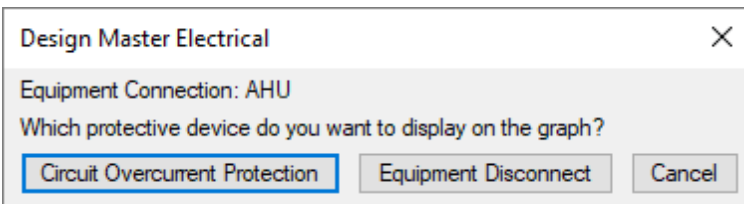
Add Custom Curve: Press this button to add a custom curve to the list of curves. The **Add Custom Curve** dialog box will appear.
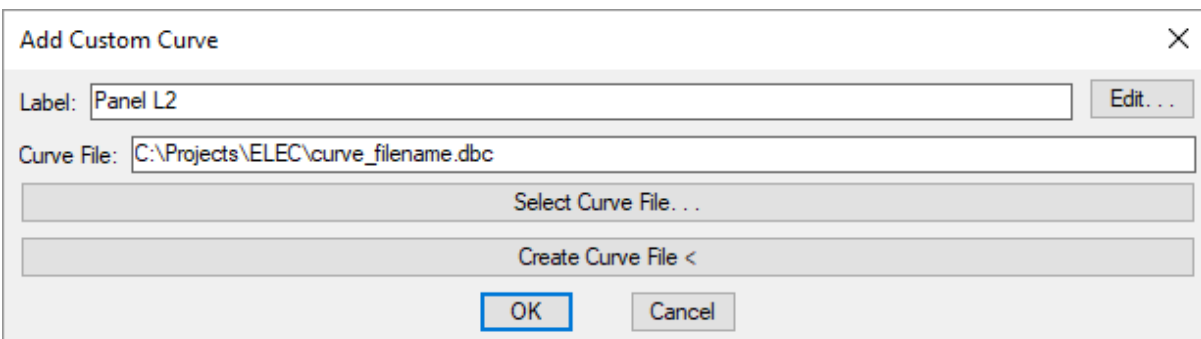
Label: The label that will correspond to the curve when the graph is inserted on the drawing.

Select Curve File: Press this button to select a custom curve file. The **Select Curve File** dialog box will appear.
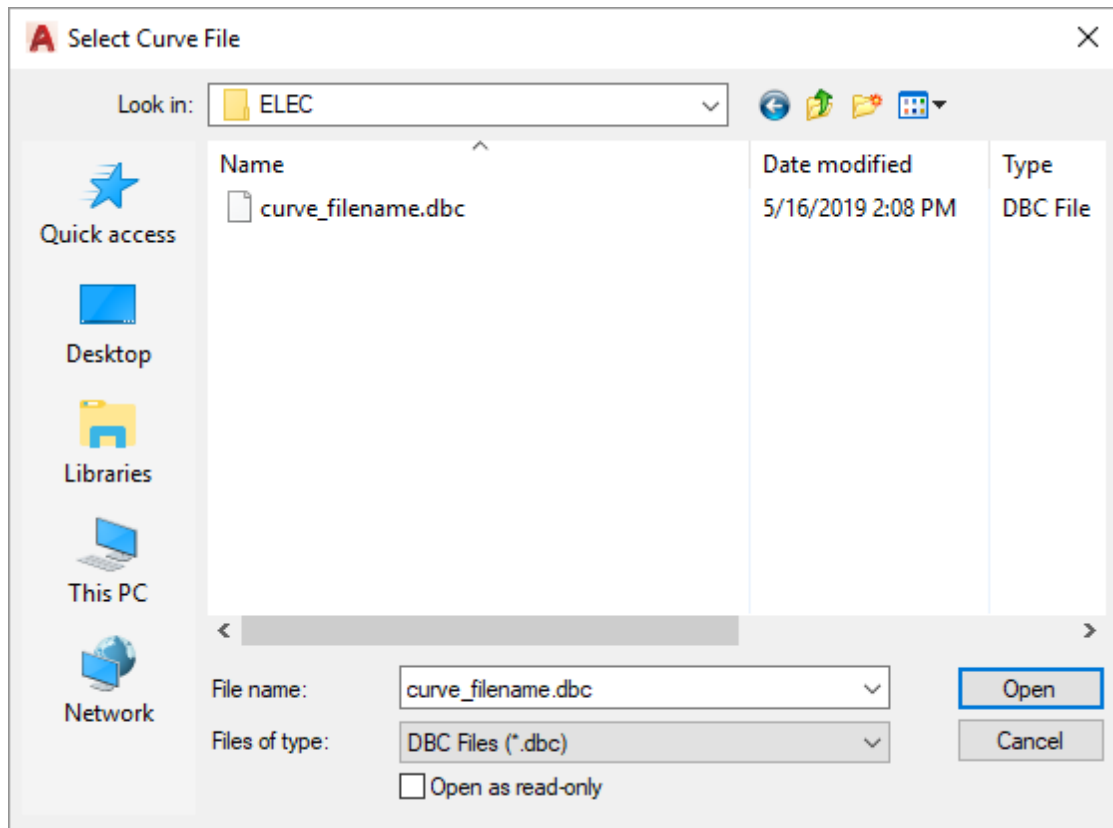
Select the curve file to be used and press the **Open** button. The file name and path will be displayed in the *Curve File* field.

Create Curve File: Press this button to create a custom curve file by tracing a time-current curve from another source, such as a PDF. The **Save Curve File As** dialog box will appear.
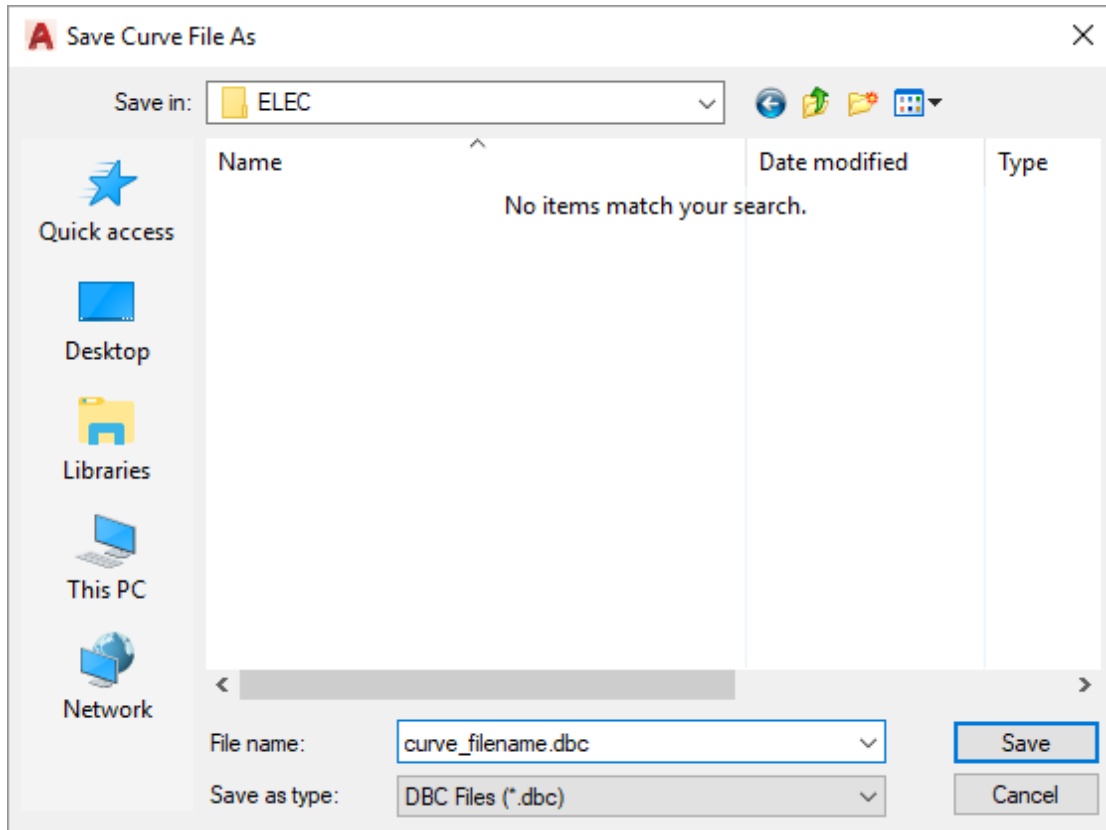
Enter a name and location for the DBC file, then press the **Save** button. You will then be prompted to enter a label for the curve.

After you have entered a label for the curve, you will be prompted to enter an amp value that appears on the graph you want to trace, then specify a point on the graph at that amp value along the x-axis.

Enter first current value in amps:

Specify location of <value> amps:

After you have specified the point, you will then be prompted to enter a second amp value and specify a point on the graph.

Enter second current value in amps:

Specify location of <value> amps:

After you have specified the point, you will then be prompted to specify points on the graph at the 1-second value and 10-second value on the y-axis.

Specify location of 1 second:

Specify location of 10 seconds:

After you have specified the points, you will then be prompted to specify the first point on the

curve.

Specify first point on curve:

Select a point at one end of the curve you want to trace. You will then be prompted to select additional points on the curve.

Specify next point on curve:

Continue tracing the curve, then press ENTER when you are finished. You will then be prompted to trace a second curve.

Specify first point on second curve / <Enter for no second curve>:

Specify first point: Select a point at one end of the second curve. You will be prompted to select additional points on the curve.

Specify next point on second curve:

Continue tracing the second curve. Press ENTER to finish the command and return to the **Add Custom Curve** dialog box. The curve will be displayed as a range with hatching between the two curves you traced.

No second curve: Press ENTER to finish the command and return to the **Add Custom Curve** dialog box. The curve will be displayed as a single line.

Inserting the Selective Coordination Graph on the Drawing

To insert the selective coordination graph on the drawing, press the **OK** button. The settings you specified in the dialog box will be used for the graph you insert.

You will be prompted for the insertion location of the graph.

Specify insertion point of selective coordination graph:

The location you specify will be used as the top-left corner for the graph.

Related Options

Selective coordination graph title: Sets the default text for the *Title* field.

Update Selective Coordination Graph

The `Update Selective Coordination Graph` command is used to update a selective coordination graph to reflect any changes to your design. Graphs are not automatically updated when changes are made. You must manually update them.

To update a selective coordination graph, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Update Selective Coordination Graph`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Update Selective Coordination Graph`

You will be prompted to specify a graph to update.

Select graph to update:

The graph will be updated to reflect the latest information.

Edit Selective Coordination Graph

To modify a selective coordination graph that is inserted on the drawing, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Edit Selective Coordination Graph`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Edit Selective Coordination Graph`

You will be prompted to select the graph to be edited.

Select graph to edit:

Edit Selective Coordination Graph Dialog Box

Changes can be made to the graph in the same way as when it was inserted. See the **Insert Selective Coordination Graph** section for more information.

Edit Curve

The `Edit Curve` command is used to set or modify the curve for a distribution equipment or equipment connection in a selective coordination graph. See the **Reading and Configuring Time-Current Curves** article in the knowledge base for more information about how the settings in this dialog box affect the curve.

To set or modify a curve, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Edit Curve`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Edit Curve`

You will be prompted to specify a curve on a selective coordination graph to edit.

Select curve to edit:

Select Curve Dialog Box

The curves displayed in this dialog box change depending upon the trip size of the breaker. If the breaker you want to use is not available, **contact us** so we can add it to the database.

Equipment: The name of the distribution equipment or equipment connection and the curve being modified.

Trip Size: The *Trip* or *Upstream OCP* value set in the **Distribution Equipment Dialog Box**, or the *Trip* or *Breaker Size* value set in the **Create Equipment** dialog box.

Manufacturer: The list of manufacturers for the specified *Trip Size*.

Group: The list of breaker types for the specified *Manufacturer* and *Trip Size*.

Breaker/Fuse: The list of breaker/fuse models for the specified *Manufacturer*, *Trip Size*, and *Group*.

Long-time Delay: This value determines how long the breaker will allow lower current increases before tripping. Applies to electronic trip breakers.

If the selected breaker allows the installer to input values digitally, the range and increments that can be input for this value will be displayed. Enter the value in the field below this list.

Short-time Pickup: This value determines the current at which the trip switches from the *Long-time Delay* to the *Short-time Delay*. Applies to electronic trip breakers.

If the selected breaker allows the installer to input values digitally, the range and increments that can be input for this value will be displayed. Enter the value in the field below this list.

I^2 T: This value determines whether the *Short-time Delay* value stays constant or varies depending upon the current. Applies to electronic trip breakers.

Short-time Delay: This value determines how long the breaker will allow higher current increases before tripping. Applies to electronic trip breakers.

If the selected breaker allows the installer to input values digitally, the range and increments that can be input for this value will be displayed. Enter the value in the field below this list.

Instantaneous Pickup: This value determines the current at which the breaker trips immediately.

If the selected breaker allows the installer to input values digitally, the range and increments that can be input for this value will be displayed. Enter the value in the field below this list.

Print Distribution Equipment Schedule

The `Print Distribution Equipment Schedule` command is used to create a distribution equipment schedule as an HTML file that can be printed.

To print a distribution equipment schedule, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Print Distribution Equipment Schedule`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Print Distribution Equipment Schedule`

Print Panel Schedule to HTML Dialog Box

Distribution Equipment to Print: A list of all of the distribution equipment in the database. You can select multiple items in the list using the `SHIFT` or `CTRL` keys.

Max Width: Restrict the schedule to a specified width in inches.

Max Height: Restrict the schedule to a specified height in inches.

Header Information: Choose what information is displayed in the schedule header by checking the corresponding box.

Circuit Information

Choose what circuit information is displayed in the schedule by checking the corresponding box.

Circuit Number: The number of the circuit.

Breaker Size: The size of the breaker.

Load: The circuit load in kVA, VA, and amps.

Description: The description of the circuit.

Remove breaker size from unused breakers: Whether breaker size is included for breakers not in use.

Remove circuit load and description from unused breakers: Whether circuit load and descriptions are included for breakers not in use.

Print Arc-Flash Stickers

The `Print Arc-Flash Stickers` command is used to create a sticker for each distribution equipment that has *Calculate Arc-Flash* set to **Yes** in the **Arc-Flash Settings** dialog box.

The sticker will also display the shock protection information for the device. Shock protection is a function of the voltage of the distribution equipment. Shock protection values are based upon *NFPA 70E-2018, Table 130.4(D)(a)*.

To create arc-flash stickers, go to

Ribbon: `DM Electrical->Distribution Equipment->` `Print Arc-Flash Stickers`

Pulldown Menu: `DM Electrical->Distribution Equipment Schedules->Print Arc-Flash Stickers`

Create Arc-Flash Stickers Dialog Box
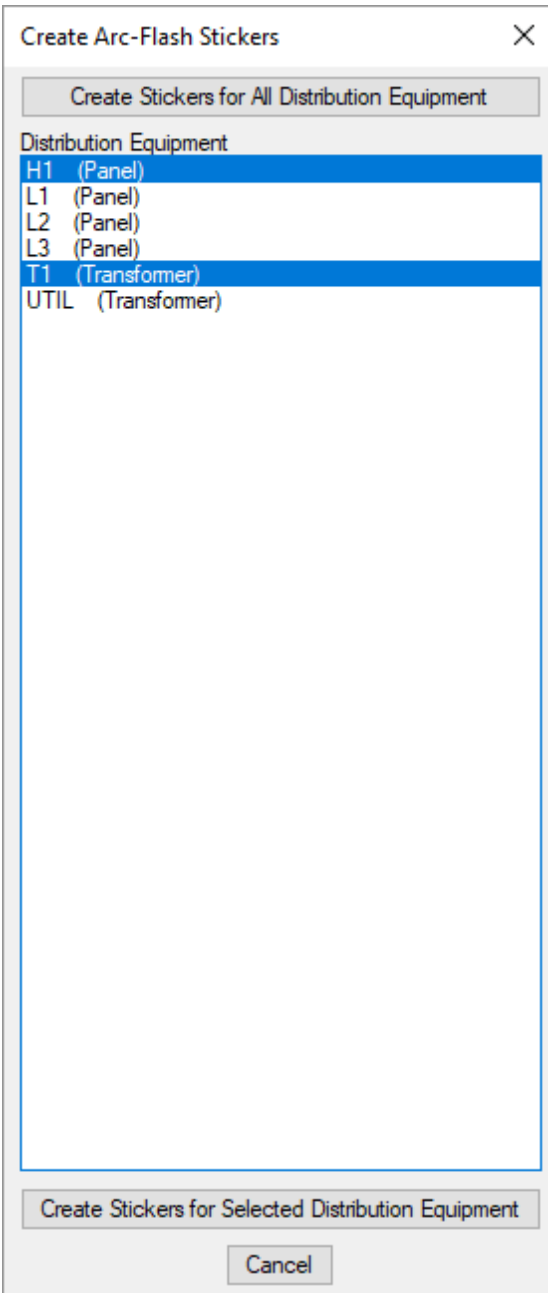
Create Stickers for All Distribution Equipment: Press this button to create stickers for all of the distribution equipment in the *Distribution Equipment* list.

Distribution Equipment: A list of all distribution equipment for which arc-flash has been calculated. You can select multiple items in the list using the SHIFT or CTRL keys.

Create Stickers for Selected Distribution Equipment: Press this button to create stickers for the selected distribution equipment.

Printing Arc-Flash Stickers

An example arc-flash sticker is shown below.

One-Line Diagram

This section describes the one-line diagram commands.

Training Videos

- **One-Line Diagram Overview**
- **The Generate One-Line and Generate Riser Commands**
- **Feeder Drafting Techniques**
- **One-Line Diagram Meters**
- **One-Line Diagram Labels**

Update One-Line Diagram

Use the `Update One-Line Diagram` command to update your one-line diagram to reflect any changes to your design.

Generally, the one-line diagram is updated automatically as you modify your design. Use this command if changes were made on a different drawing in the project, or when you need to be sure the one-line diagram reflects the latest changes.

To update the one-line diagram, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⟳ `Update One-Line Diagram`

Pulldown Menu: `DM Electrical->One-Line Diagram->Update`

This command will update:
- all of the labels on the one-line diagram
- any connections between devices, removing feeders that no longer exist and adding ones that are missing
- all of the layers, colors, and text styles to match your current customization settings

Insert Distribution Equipment

The `Insert Distribution Equipment` command is used to insert distribution equipment and equipment connections on the drawing for the one-line diagram.

To insert distribution equipment or an equipment connection, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⓘ `Insert Distribution Equipment`

Pulldown Menu: `DM Electrical->One-Line Diagram->Insert Distribution Equipment`

Insert Distribution Equipment Dialog Box
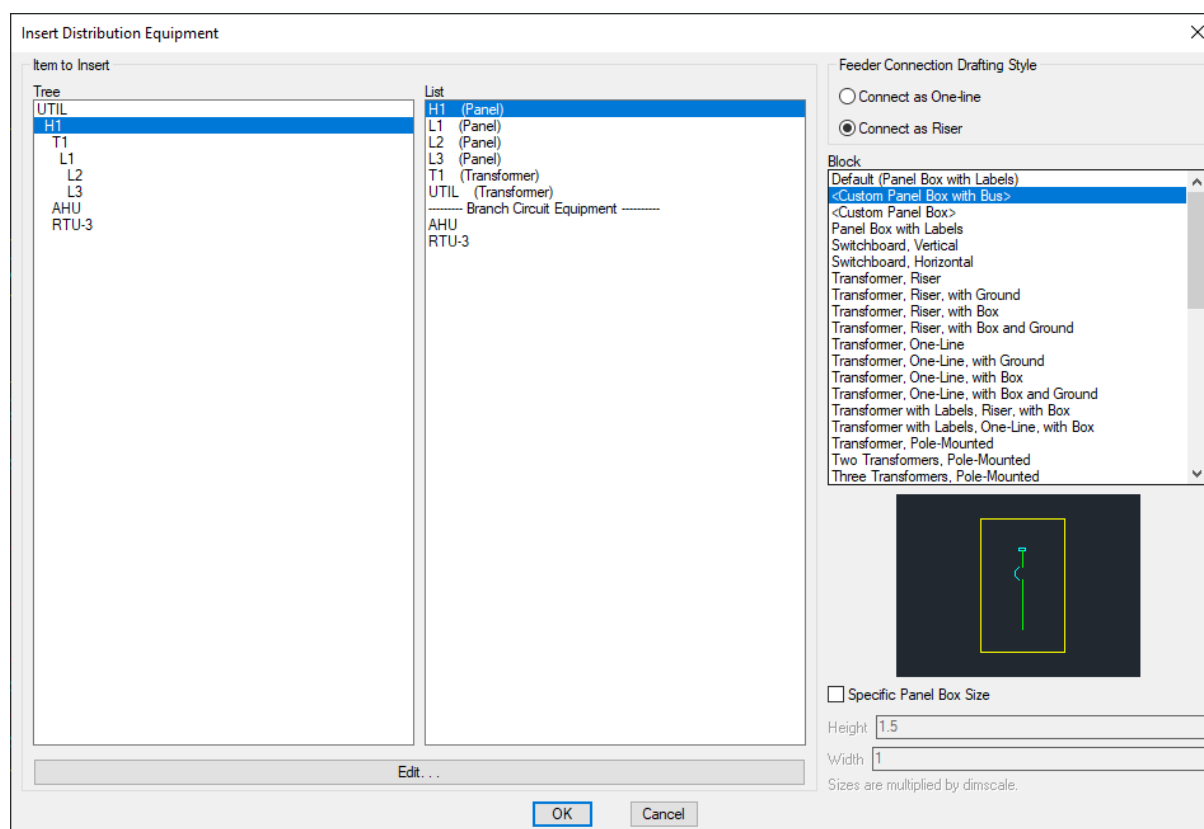
Tree / List: All distribution equipment and equipment connections in the current project. They are listed in a tree corresponding to how the equipment are connected to each other, and alphabetically by callout.

Feeder Connection Drafting Style: Whether feeders are connected to the selected block in a one-line configuration or a riser configuration.

Block: The block that will be used for the distribution equipment or equipment connection. You can add more blocks to this list using the **One-Line Diagram Distribution Equipment Blocks->Edit Project List** command. There are two special entries in the list:
- **<Custom Panel Box with Bus>:** A box representing the panel with a bus bar inserted in the middle of it.
- **<Custom Panel Box>:** A box representing the panel. No bus bar is inserted.

Specific Panel Box Size: If you selected either **Custom Panel Box and Bus** or **Custom Panel Box** from the *Block* list, you can check this box to set the dimensions for the block.

Press the **Edit** button to open the corresponding dialog box for the selected distribution equipment.

Press the **OK** button to insert the distribution equipment or equipment connection on the drawing.

Inserting a Panel Box

If you selected either **Custom Panel Box and Bus** or **Custom Panel Box** from the *Block* list and did not check **Specific Panel Box Size**, you will be prompted for the first corner of the panel box.

First corner of panel:

Select the location of the panel corner. You will then be prompted for the second corner of the panel box.

Second corner:

If you selected **Custom Panel Box and Bus**, you will be prompted to specify the starting point of the bus bar or press **ENTER** to insert it in the default location. The size of the panel block will affect the direction in which the bus will be inserted. The bus will be inserted in a direction matching the longest side of the box. A short, wide box will result in the bus being inserted horizontally. A tall, thin box will result in a vertical bus.

Feeders will be inserted between the distribution equipment or equipment connection and anything it is connected to that is already on the drawing.

Inserting a Distribution Equipment Diagram Block

If you selected a different block, or checked *Specific Panel Box Size*, you will be prompted for the insertion point.

Select insertion point:

If you selected **Custom Panel Box and Bus**, you will be prompted to specify the starting point of the bus bar or press **ENTER** to insert it in the default location. The size of the panel block will affect the direction in which the bus will be inserted. The bus will be inserted in a direction matching the longest side of the box. A short, wide box will result in the bus being inserted horizontally. A tall, thin box will result in a vertical bus.

Feeders will be inserted between the distribution equipment or equipment connection and anything it is connected to that is already on the drawing.

Related Options

Connect Distribution Equipment on One-Line Diagram

The `Connect Distribution Equipment on One-Line Diagram` command is used to connect distribution equipment that has been inserted on the one-line diagram.

To connect distribution equipment on the one-line diagram, go to

Ribbon: `DM Electrical->One-Line Diagram->` Connect Distribution Equipment on One-Line Diagram

Pulldown Menu: `DM Electrical->One-Line Diagram->Connect Distribution Equipment on One-Line Diagram`

You will be prompted to select the device to be connected. This device is the downstream distribution equipment that is further from the utility. This device is also commonly called a sub-feed panel or a sub-panel.

Select downstream equipment to connect:

You will then be prompted to select the device to which the first device will be connected. This device is the upstream distribution equipment that is closer to the utility.

Select upstream distribution equipment to connect to:

After you make your selections, the **Distribution Equipment Connection** dialog box will appear.

Distribution Equipment Connection Dialog Box

Top Lugs: This selection is available for panels, switchboards, breakers, disconnects, and transfer switches that have their *Lugs* set to **DOUBLE**.

Feedthrough: This selection is available for panels, switchboards, breakers, disconnects, and transfer switches that have their *Lugs* set to **FEEDTHRU**.

Breaker / Tap: All devices will allow you to select **Breaker / Tap** as the connection type. Select the breaker or tap to connect to using the *Odd Breakers / Taps* or *Even Breakers* lists.

Odd Breakers / Taps: For panels, a list of the odd breakers. For all other distribution equipment, a list of all of the taps. The breaker or tap selected in this list will be used for the connection. If an *Even Breaker* is selected, the top blank entry will be selected in this list.

Even Breakers: For panels, a list of the even breakers. Not used for all other distribution equipment. The breaker selected in this list will be used for the connection. If an *Odd Breaker* is selected, the top blank entry will be

106

selected in this list.

Change Number of Poles: Press this button to open the **Change Number of Poles** dialog box and change the number of poles of the currently selected circuit. This dialog box can be used to convert two-pole and three-pole circuits to multiple single-pole circuits. See the **Change Number of Poles** section for more information.

Feeder Connection Drafting Style: Whether the feeder connecting the selected devices is drawn in a one-line configuration or a riser configuration.

Press the **Connect** button to make the connection.

Generate One-Line

The `Generate One-Line` command creates a one-line diagram for distribution equipment and any downstream distribution equipment and equipment connections in a vertical, one-line configuration.

To generate a one-line diagram, go to

Ribbon: `DM Electrical->One-Line Diagram->` 🏛 `Generate One-Line`

Pulldown Menu: `DM Electrical->One-Line Diagram->Generate One-Line`

Generate One-Line Dialog Box
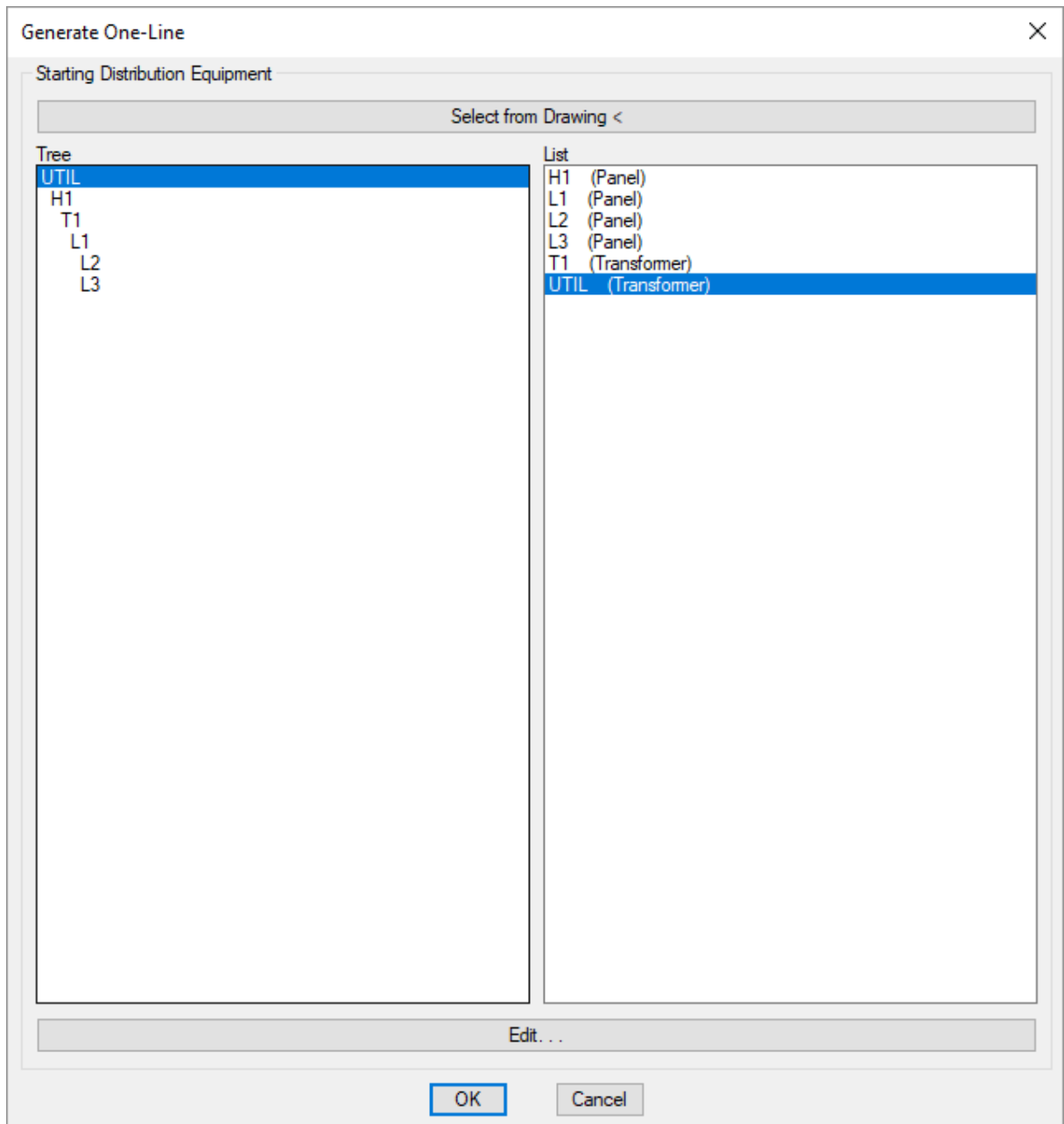
Select from Drawing

Press the **Select from Drawing** button to generate a one-line from an existing distribution equipment on the drawing.

You will be prompted to select a distribution equipment on the drawing.

Select distribution equipment to start at:

Any downstream distribution equipment and devices connected to the equipment will be generated. Feeders will be inserted between anything that is connected.

Select from Tree / List

All distribution equipment in the current project is listed in a tree corresponding to how the equipment are connected to each other, and alphabetically by callout.

Press the **Edit** button to open the corresponding dialog box for the selected equipment.

Press the **OK** button to insert the selected equipment on the drawing.

You will be prompted for the insertion point.

Select insertion point:

The selected distribution equipment and any downstream equipment and devices will be generated. Feeders will be inserted between anything that is connected.

Related Options

Distribution equipment sort order: Sets the order in which downstream equipment connections are generated on the diagram.

Feeder offset multiplier: Sets the offset distance between feeders generated on the diagram.

Default one-line blocks: Sets the default blocks used for each distribution equipment type when the diagram is generated.

Automatically create feeder IDs: Sets whether feeder IDs are created automatically when the diagram is generated.

There are several other options that affect the overall appearance of the one-line diagram. See the **One-Line Diagram Options** section for more information.

Generate Riser

The `Generate Riser` command creates a one-line diagram for distribution equipment and any downstream distribution equipment and equipment connections in a horizontal, riser configuration. Devices with a specified elevation are stacked vertically according to elevation.

To generate a riser diagram, go to

Ribbon: `DM Electrical->One-Line Diagram->` 📊 `Generate Riser`

Pulldown Menu: `DM Electrical->One-Line Diagram->Generate Riser`

Generate Riser Dialog Box
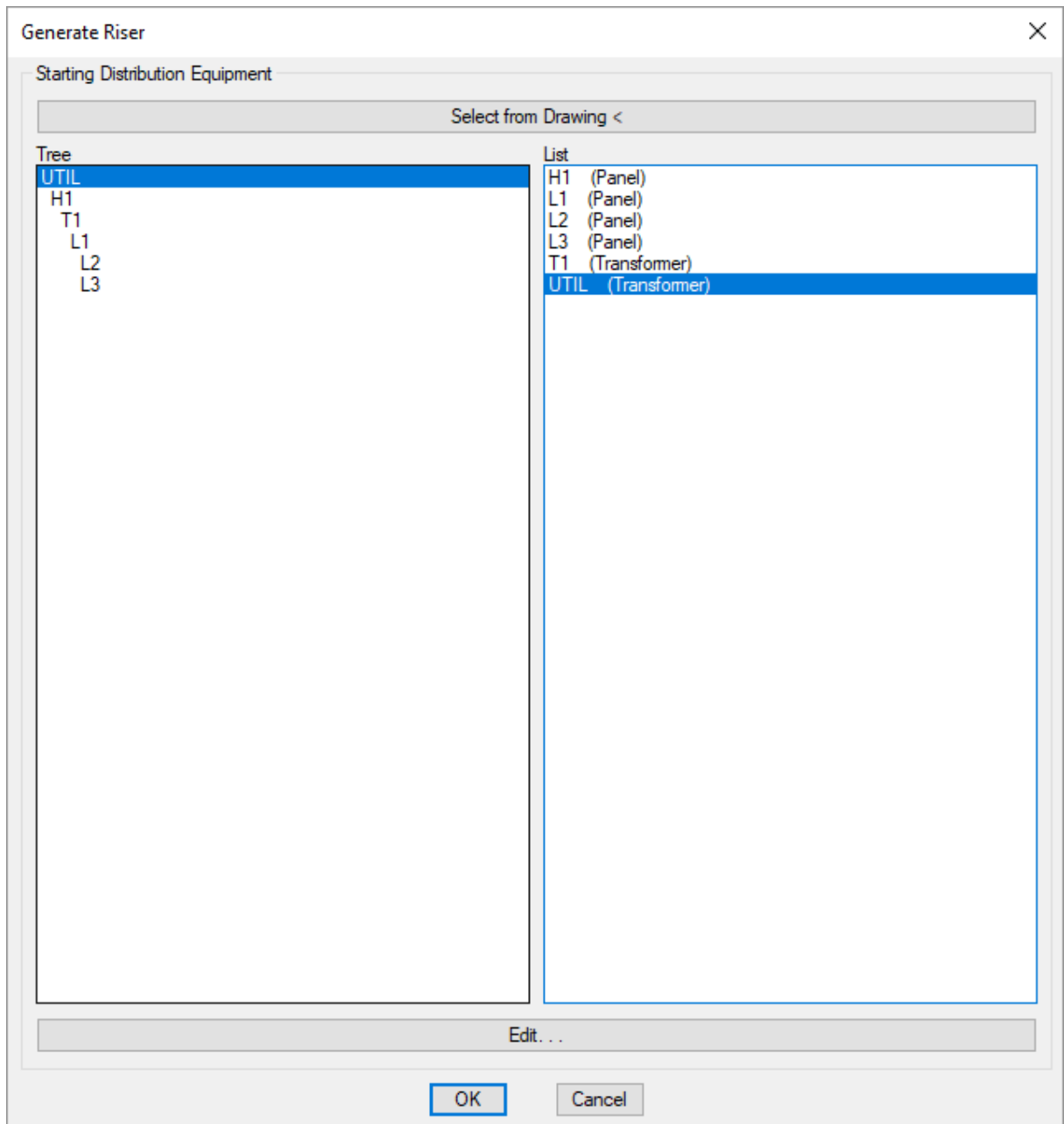
Select from Drawing

Press the **Select from Drawing** button to generate a riser from an existing distribution equipment on the drawing.

You will be prompted to select a distribution equipment on the drawing.

Select distribution equipment to start at:

Any downstream distribution equipment and devices connected to the equipment will be generated. Feeders will be inserted between anything that is connected.

Select from Tree / List

All distribution equipment in the current project is listed in a tree corresponding to how the equipment are connected to each other, and alphabetically by callout.

Press the **Edit** button to open the corresponding dialog box for the selected equipment.

Press the **OK** button to insert the selected equipment on the drawing.

You will be prompted for the insertion point.

Select insertion point:

The selected distribution equipment and any downstream equipment and devices will be generated. Feeders will be inserted between anything that is connected.

Related Options

Distribution equipment sort order: Sets the order in which downstream equipment connections are generated on the diagram.

Feeder offset multiplier: Sets the offset distance between feeders generated on the diagram.

Riser height multiplier: Sets the distance between elevations when the diagram is generated.

Default fed from: Sets whether feeders connect to the top or bottom of devices when the diagram is generated.

Feeder connection location in riser to panels with no bus: Sets where feeders connect to pieces of distribution equipment with no bus bar.

Default riser blocks: Sets the default blocks used for each distribution equipment type when the diagram is generated.

Automatically create feeder IDs: Sets whether feeder IDs are created automatically when the diagram is generated.

There are several other options that affect the overall appearance of the one-line diagram. See the **One-Line Diagram Options** section for more information.

Edit Distribution Equipment

The `Edit Distribution Equipment` command is used to change the settings of distribution equipment on the drawing.

If your one-line diagram is based upon a Revit model imported using Design Master Electrical RT, do not use this command. Use the **Edit** command for Design Master Electrical RT.

To edit distribution equipment, go to

Ribbon: `DM Electrical->One-Line Diagram->` ✏ `Edit Distribution Equipment`

Pulldown Menu: `DM Electrical->One-Line Diagram->Edit Distribution Equipment`

You will be prompted to select the distribution equipment to be changed.

Select device to query:

The corresponding **distribution equipment** dialog box will be opened with that equipment selected. You can then view the equipment and make changes as necessary.

Move Distribution Equipment

The `Move Distribution Equipment` command is used to move distribution equipment on the drawing. Using this command will also move any feeders and downstream connections attached to the equipment.

To move distribution equipment, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⬛➡ `Move Distribution Equipment`

Pulldown Menu: `DM Electrical->One-Line Diagram->Move Distribution Equipment`

You will be prompted to select the distribution equipment to move.

Select distribution equipment to move:

You can then move the selected distribution equipment the same way you would using the standard CAD `MOVE` command.

Change Graphics

The `Change Graphics` command is used to change the graphics displayed for distribution equipment, equipment connections, and feeders on the drawing. Blocks can be added to the start and the end of feeders. The block used for the OCP in the feeder can also be changed.

To insert or change the graphics on distribution equipment, equipment connections, or feeders, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⬛ `Change Graphics`

Pulldown Menu: `DM Electrical->One-Line Diagram->Change Graphics`

You will be prompted to select a distribution equipment or one or more feeders to edit.

Select distribution equipment or feeders to change graphics of:

After you make your selection(s), the **Change Distribution Equipment Block** or **Change Feeder Blocks** dialog box will appear.

If you select both distribution equipment and feeders, an alert box will prompt you to choose which type you want to change.

Change Distribution Equipment Block Dialog Box
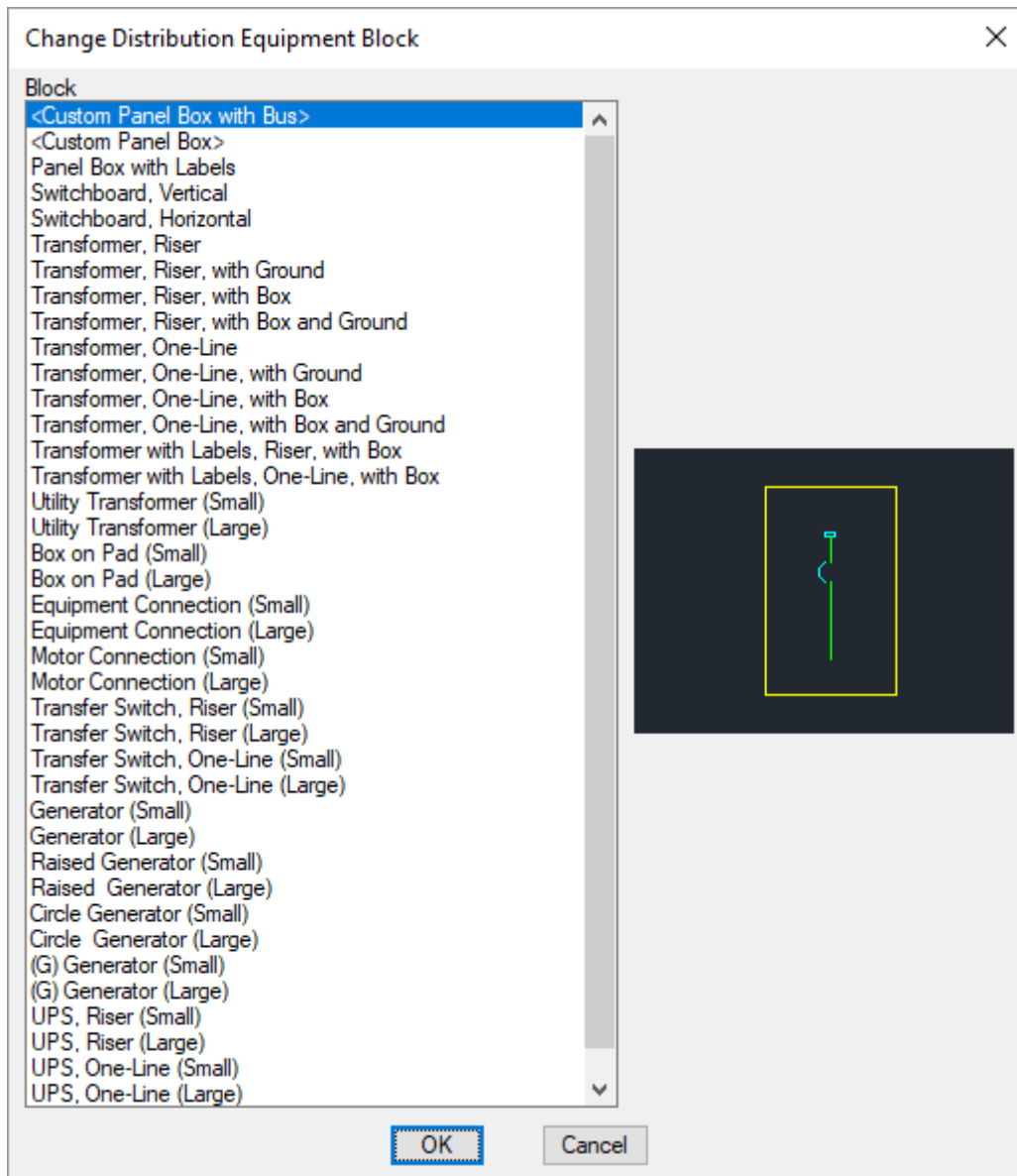
Block: The block or graphic used on the drawing for the distribution equipment or equipment connection. Select a new block to change the block used on the drawing.

Press the **OK** button to save the changes made to the distribution equipment or equipment connection.

Change Feeder Blocks Dialog Box

Feeder ID: Whether the feeder ID is displayed on the feeder.

OCP Block: The block used for the overcurrent protection on the feeder. The OCP block can be moved along the feeder wire.
- **<Use upstream distribution equipment OCP block>:** Select this entry to use the same OCP block as the upstream distribution equipment.
- **<None>:** Select this entry to remove the OCP block.

Disconnect Block: Applies only to feeders connected to an equipment connection. The block used for the disconnect on the feeder. The disconnect block can be moved along the feeder wire.
- **<None>:** Select this entry to remove the disconnect block.

Block at Start of Feeder: The block inserted at the start of the feeder. The location of this block cannot be moved.
- **<None>:** Select this entry to remove the block at the start of the feeder.

Block at End of Feeder: The block inserted at the end of the feeder. The location of this block cannot be moved.
- **<None>:** Select this entry to remove the block at the end of the feeder.

Press the **OK** button to save the changes made to the feeder.

Related Options

Insert feeder IDs inside feeder lines: Sets the default for the *Feeder ID* setting.

There are several other options that affect the overall appearance of the one-line diagram. See the **One-Line Diagram Options** section for more information.

Change Layer System

The `Change Layer System` command is used to change the layer system for distribution equipment and feeders

on the drawing.

To change the layer system of distribution equipment or feeders, go to

Ribbon: `DM Electrical->One-Line Diagram->` ▯▯ `Change Layer System`

Pulldown Menu: `DM Electrical->One-Line Diagram->Change Layer System`

You will be prompted to select one or more distribution equipment or feeders.

Select distribution equipment or feeder to change layer system of:

After you make your selections, the **Change Layer System** dialog box will appear.

See the **Layer System** section for more information.

Change Linked Database Record

The `Change Linked Database Record` command is used to change which distribution equipment or feeder is represented by a graphic on the drawing. This can be useful when copying a block on the one-line diagram, as the copy will not have an equipment linked to it.

To change a database record displayed on the one-line diagram, go to

Ribbon: `DM Electrical->One-Line Diagram->` ▯ᴰᴮ `Change Linked Database Record`

Pulldown Menu: `DM Electrical->One-Line Diagram->Change Linked Database Record`

You will be prompted to select a distribution equipment or feeder.

Select distribution equipment or feeder to link to a new database record:

If you select a distribution equipment, the **Change One-Line Diagram Item** dialog box will appear. Select a distribution equipment from the list and press the **OK** button.

Match Labels and Graphics

The `Match Labels and Graphics` command is used to change the labels and graphics of one or more distribution equipment or feeders to match an existing equipment or feeder on the drawing.

To match labels and graphics, go to

Ribbon: `DM Electrical->One-Line Diagram->` 🔻 `Match Labels and Graphics`

Pulldown Menu: `DM Electrical->One-Line Diagram->Match Labels and Graphics`

You will be prompted to select a distribution equipment or feeder from which the graphics and labels will be copied.

Select source distribution equipment or feeder:

You will then be prompted to select one or more distribution equipment or feeders, depending upon your source, to change.

Select destination <Source type>:

Press **ENTER** to change the labels and graphics of the selected distribution equipment or feeders.

Insert Meter

The `Insert Meter` command is used to insert a meter on the one-line diagram. Meters are for informational purposes only and are not otherwise used.

To insert a meter on a drawing, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⊕ `Insert Meter`

Pulldown Menu: `DM Electrical->One-Line Diagram->Insert Meter`

Insert Meter Dialog Box
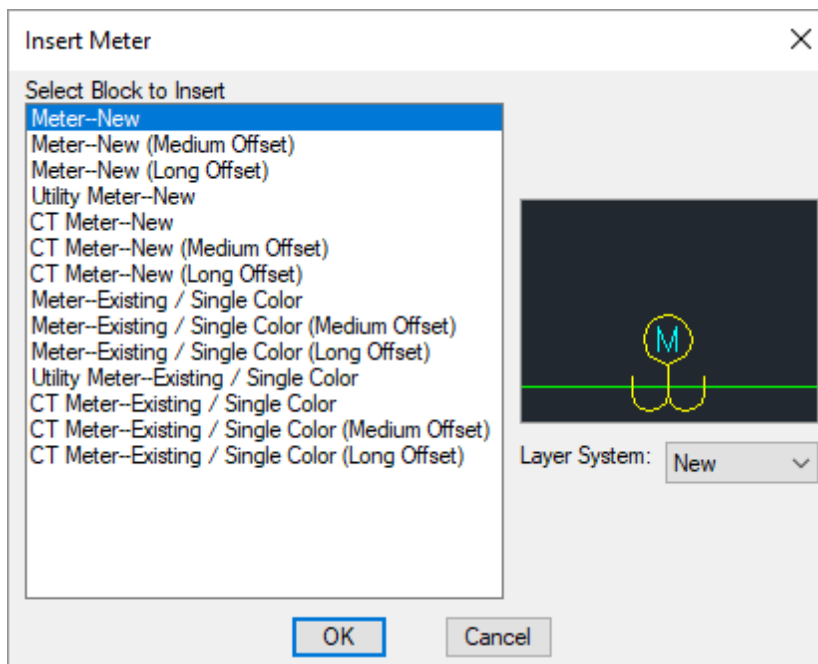
Select Block to Insert: The block to be used on the drawing for the meter.

Layer System: See the **Layer System** section for more information.

Press the **OK** button. You will be prompted to select a feeder or bus onto which the meter will be inserted.

Select feeder or bus to insert meter on:

After you select the feeder or bus, the meter will be inserted on the drawing.

Remove Meter

To remove a meter from a drawing, go to

Ribbon: `DM Electrical->One-Line Diagram->` ✗ `Remove Meter`

Pulldown Menu: `DM Electrical->One-Line Diagram->Remove Meter`

You will be prompted to select the feeder or bus to which the meter is connected.

Select feeder or bus to remove meter from:

After you select the feeder or bus, the connected meter will be removed from the drawing.

Show Bus

To show the bus in a distribution equipment, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⚑ `Show Bus`

Pulldown Menu: `DM Electrical->One-Line Diagram->Show Bus`

You will be prompted to select the distribution equipment in which the bus is to be shown.

Select distribution equipment to insert bus on:

Select the distribution equipment and the bus will be shown in the graphic.

Hide Bus

To hide the bus in a distribution equipment, go to

Ribbon: `DM Electrical->One-Line Diagram->` ✗ `Hide Bus`

Pulldown Menu: `DM Electrical->One-Line Diagram->Hide Bus`

You will be prompted to select the distribution equipment in which the bus is to be hidden.

Select distribution equipment to remove bus from:

Select the distribution equipment and the bus will be hidden.

Only the bus graphic will be hidden. Any other text or labels will remain.

Flip Bus

To flip the bus in a distribution equipment, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⊞ `Flip Bus`

Pulldown Menu: `DM Electrical->One-Line Diagram->Flip Bus`

You will be prompted to select the distribution equipment in which the bus is to be flipped.

Select distribution equipment to flip bus on:

Select the distribution equipment and the bus will be flipped.

For vertical bus bars, if the feeder was being fed to the top of the distribution equipment, it will now be fed to the bottom. If the feeder was being fed to the bottom of the distribution equipment, it will now be fed to the top.

Insert Panel Tab

The `Insert Panel Tab` command is used to change the appearance of custom panel blocks on the one-line diagram. It is for drafting purposes only and is not otherwise used.

To insert a panel tab on a distribution equipment, go to

Ribbon: `DM Electrical->One-Line Diagram->` `Insert Panel Tab`

Pulldown Menu: `DM Electrical->One-Line Diagram->Insert Panel Tab`

You will be prompted to select the distribution equipment on which to insert the panel tab.

Select distribution equipment to add tab to:

Select the distribution equipment and the tab will appear on the block as shown below.
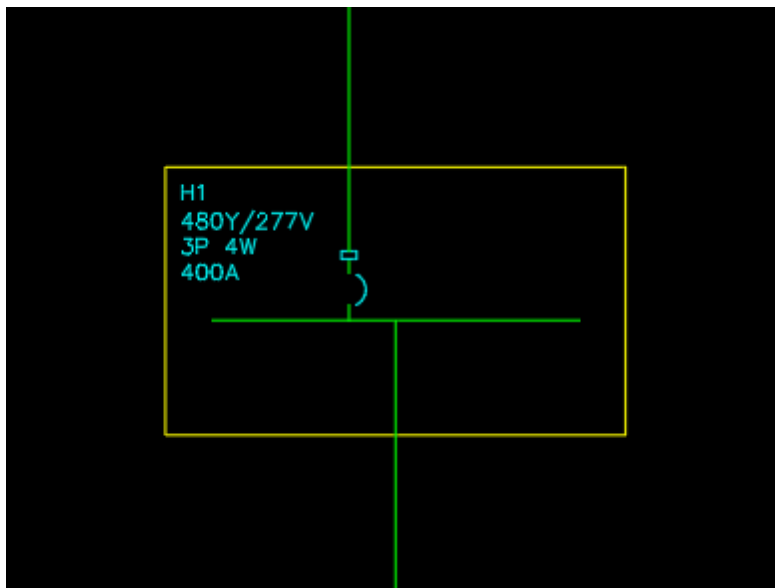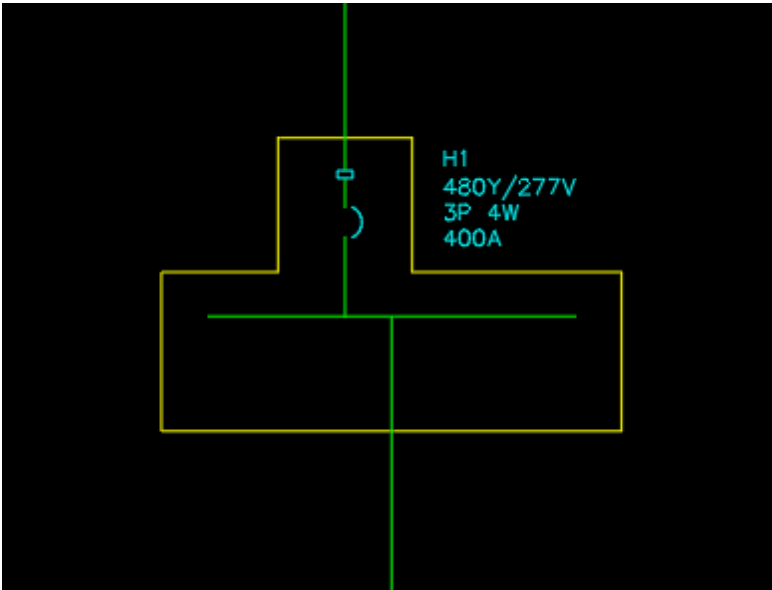
Remove Panel Tab

To remove a panel tab from a distribution equipment, go to

Ribbon: `DM Electrical->One-Line Diagram->` `Remove Panel Tab`

Pulldown Menu: `DM Electrical->One-Line Diagram->Remove Panel Tab`

You will be prompted to select the distribution equipment from which the panel tab is to be removed.

Select distribution equipment to remove tab from:

After you select the distribution equipment, the tab will be removed from the graphic.

Insert and Modify Labels

The `Insert and Modify Labels` command is used to insert, edit, and remove the labels on distribution equipment, feeders, and equipment connections on the one-line diagram.

To insert and modify labels, go to

Ribbon: `DM Electrical->One-Line Diagram->` `Insert and Modify Labels`

Pulldown Menu: `DM Electrical->One-Line Diagram->Insert and Modify Labels`

You will be prompted to select the distribution equipment or feeder label to edit.

Select distribution equipment or feeder to edit:

Select the distribution equipment, feeder, or equipment connection to be labeled.

Insert and Modify Labels Dialog Box

Checkbox: Whether the label corresponding to the row is inserted on the drawing.

Field: A description of the value that will be inserted.

Value: The text that will appear on the drawing. It includes the prefix, the value from the database, and the suffix. The prefix and suffix can be changed in this dialog box. The value from the database must be changed in the device definition itself.

Prefix: Text that is added before the value associated with the field in the database.

Suffix: Text that is added after the value associated with the field in the database.

Alignment: The alignment of the text within the text boundary.

Scale: The scale factor of the text.

Press the **OK** button when you are finished changing the notes for the distribution equipment, feeder, or equipment connection. Any labels that were removed or changed will be updated on the drawing.

You will be prompted to insert any new labels on the drawing.

Select insertion point or [Next label/<Group with other label>]:

Insertion point: Specify a point on the drawing. The top left corner of the label will be inserted at that point.

Next label: If you checked multiple labels to be inserted, type **N** to cycle to the next label.

toggle Snap to feeder: If you selected a feeder, type **s** to toggle whether the label snaps to the feeder when inserting the label.

Group with other label: Press **ENTER** or type **G** to group the label with another label on the drawing. You will be prompted to select a label to group with. See the **Group Labels** section for more information.

Insert below last label: After placing the first label, if you checked multiple labels to be inserted, press **ENTER** or type **I** to insert the label below the last label inserted.

Related Options

Insert labels inside feeder lines: Sets whether labels are inserted inside or to the side of feeders.

"Main Lugs Only" label: Sets the default value for the *Main Disconnect Type*, *Main Trip Amps*, and *Main Trip Amps and Poles* fields when the main disconnect for the distribution equipment is set to **Main Lugs Only**.

There are several other options that affect the overall appearance of the one-line diagram. See the **One-Line Diagram Options** section for more information.

Move Label

The `Move Labels` command is used to move labels on distribution equipment, feeders, and equipment connections on the one-line diagram.

To move a label, go to

Ribbon: `DM Electrical->One-Line Diagram->` ≡→ `Move Labels`

Pulldown Menu: `DM Electrical->One-Line Diagram->Move Labels`

You will be prompted to select the label to move.

Select label to move:

You can then specify a new location for the selected label. If you selected a feeder label, type **s** to toggle whether the label snaps to the feeder when moving the label.

Rotate Label

The `Rotate Labels` command is used to rotate labels on distribution equipment, feeders, and equipment connections on the one-line diagram.

To rotate a label, go to

Ribbon: `DM Electrical->One-Line Diagram->` 🔄 `Rotate Labels`

Pulldown Menu: `DM Electrical->One-Line Diagram->Rotate Labels`

You will be prompted to select the label to rotate.

Select label to rotate:

You can then specify a rotation angle for the selected label.

Group Labels

The `Group Labels` command is used to group together labels on distribution equipment, feeders, and equipment connections on the one-line diagram. Labels in a group can be modified all at once using the **Move Label** and **Rotate Label** commands and the *Alignment* and *Scale* columns of the **Insert and Modify Labels** dialog box.

To group labels together, go to

Ribbon: `DM Electrical->One-Line Diagram->` ▐≡ `Group Labels`

Pulldown Menu: `DM Electrical->One-Line Diagram->Group Labels`

You will be prompted to select a label, which will be positioned at the top of the group.

Select label in correct location:

You can then select one or more labels, including other groups of labels, to add to the group.

Remove Label from Group

To remove a label from a group, go to

Ribbon: `DM Electrical->One-Line Diagram->` ▐✖ `Remove Label from Group`

Pulldown Menu: `DM Electrical->One-Line Diagram->Remove Label from Group`

You will be prompted to select the group from which the label will be removed.

Select label group to remove label from:

The **Remove Label from Group** dialog box will appear. Select the label you want to remove and press the **OK** button.

You will then be prompted to specify a location for the selected label.

Specify insertion point or [Group with other label/<Remove label>]:

> *Insertion point:* Specify a point on the drawing. The top left corner of the label will be inserted at that point.
>
> *Group with other label:* Type G to group the label with another label on the drawing. You will be prompted to select a label to group with.

Remove label: Press **ENTER** or type **R** to remove the label.

Reorder Labels in Group

To reorder labels in a group, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⬛ `Reorder Labels in Group`

Pulldown Menu: `DM Electrical->One-Line Diagram->Reorder Labels in Group`

You will be prompted to select the group of labels you want you reorder.

Select label group to reorder:

Reorder Labels in Group Dialog Box
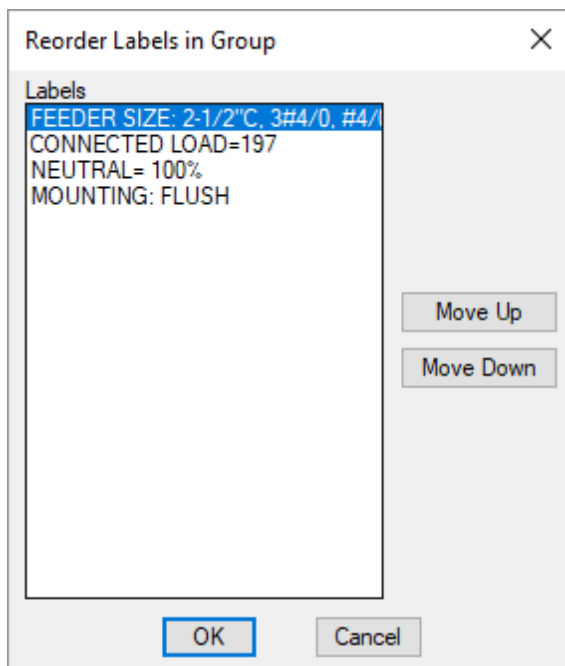
Select a label from the list. Press the **Move Up** and **Move Down** buttons to move the selected label.

Press the **OK** button to save your changes and update the label group.

Insert Leader

The `Insert Leader` command is used to add a leader to a label or group of labels.

To insert a leader, go to

Ribbon: `DM Electrical->One-Line Diagram->` ◤ `Insert Leader`

Pulldown Menu: `DM Electrical->One-Line Diagram->Insert Leader`

You will be prompted to specify a label to which to add the leader.

Select label to add a leader to:

You will then be prompted to specify an end point for the leader.

Specify end point of leader:

Specify a point on the drawing. The leader will be drawn from the selected label to that point.

Remove Leader

The `Remove Leader` command is used to remove a leader from a label or group of labels.

To remove a leader, go to

Ribbon: `DM Electrical->One-Line Diagram->` `Remove Leader`

Pulldown Menu: `DM Electrical->One-Line Diagram->Remove Leader`

You will be prompted to select the label from which the leader is to be removed.

Select label to remove a leader from:

After you select the label, the leader will be removed.

Redraw Feeder

The `Redraw Feeder` command is used to redraw feeders on the one-line diagram.

To redraw a feeder, go to

Ribbon: `DM Electrical->One-Line Diagram->` `Redraw Feeder`

Pulldown Menu: `DM Electrical->One-Line Diagram->Redraw Feeder`

You will be prompted to select a feeder or downstream distribution equipment.

Select one-line diagram feeder or distribution equipment being fed to redraw:

If you select a feeder, you will then be prompted to select a start point at the upstream distribution equipment from which to begin drawing the feeder. If you select a distribution equipment, you will be prompted to select a start point at the downstream distribution equipment from which to begin drawing the feeder.

Specify start point at <upstream/downstream> distribution equipment or [Reverse direction / Offset from other feeder / <use Default starting point>]:

   Start point: Specify a point on the drawing. The feeder will start at that point.

Reverse direction: Type **R** to toggle between specifying a start point at the upstream or downstream distribution equipment.

Offset from other feeder: Type **O** to offset the feeder from another feeder. You will be prompted to select a feeder from which to offset.
Only available if you specify a starting point at the upstream distribution equipment.

Use default starting point: Press **ENTER** or type **D** to use the default feeder starting point for the distribution equipment.

After you have selected a starting point, you will then be prompted to specify additional points to which the feeder will be drawn.

Specify next point or [Undo / Finish / Offset from other feeder]:

Next point: Specify a point on the drawing. The feeder will extend in a straight horizontal or vertical line to a point parallel to the specified point.

Undo: Type **U** to return to the previous command prompt.

Finish: Type **F** to automatically connect the feeder to the downstream distribution equipment.
Only available if you specified a starting point at the upstream distribution equipment.

Offset from other feeder: Type **O** to offset the feeder being redrawn from another feeder. You will be prompted to specify the feeder from which the redrawn feeder is being offset. Temporarily, object snap points for the selected feeder will be offset from the feeder itself.
Type **N** to undo this command.

After you have redrawn the feeder, you will be prompted to specify a location on the feeder for the feeder ID block to be inserted.

Specify feeder block insertion location or [<Default location>]:

Insertion location: Specify a point on the drawing along the feeder. The feeder ID block will be inserted at that point.

Default location: Press **ENTER** or type **D** to insert the feeder ID block at the default location along the feeder.

Insert Feeder

The `Insert Feeder` command is used to insert feeders on the one-line diagram. Feeders inserted using this command are for informational purposes only. To insert feeders that correctly show how they are connected to distribution equipment and other devices, use the **Redraw Feeder** command.

To insert a feeder, go to

Ribbon: `DM Electrical->One-Line Diagram->` 🖿 `Insert Feeder`

Pulldown Menu: `DM Electrical->One-Line Diagram->Insert Feeder`

You will be prompted to select an upstream distribution equipment to which the feeder will be connected.

Select upstream distribution equipment to connect to:

Insert Feeder Dialog Box

Layer System: The layer system used for the feeder when it is inserted on the drawing. See the **Layer System** section for more information.

OCP Block: The block used for the overcurrent protection on the feeder. The OCP block can be moved along the feeder.

Block at Start of Feeder: The block inserted at the start of the feeder. The location of this block cannot be moved.

Block at End of Feeder: The block inserted at the end of the feeder. The location of this block cannot be moved.

Press the **OK** button. You will then be prompted to select a start point at the upstream distribution equipment from which to begin drawing the feeder.

Specify start point at upstream distribution equipment or [Reverse direction / Offset from other feeder / <use Default starting point>]:

> *Start point:* Specify a point on the drawing. The feeder will start at that point.

> *Reverse direction:* Type ʀ to toggle between specifying a start point at the upstream or downstream distribution equipment.

Offset from other feeder: Type **O** to offset the feeder from another feeder. You will be prompted to select a feeder from which to offset.
Only available if you specify a starting point at the upstream distribution equipment.

Use default starting point: Press **ENTER** or type **D** to use the default feeder starting point for the distribution equipment.

After you have selected a starting point, you will then be prompted to specify additional points to which the feeder will be drawn.

Specify next point or [Undo / Finish / Offset from other feeder]:

Next point: Specify a point on the drawing. The feeder will extend in a horizontal or vertical line to a point parallel to the specified point.

Undo: Type **U** to return to the previous command prompt.

Finish: Type **F** to complete the feeder.
Only available if you specified a starting point at the upstream distribution equipment.

Offset from other feeder: Type **O** to to offset the feeder being redrawn from another feeder. You will be prompted to specify the feeder from which the redrawn feeder is being offset. Temporarily, object snap points for the selected feeder will be offset from the feeder itself.
Type **N** to undo this command.

Align Feeder

The `Align Feeder` command is used to align parallel segments of different feeders on the one-line diagram.

To align feeders, go to

Ribbon: `DM Electrical->One-Line Diagram->` `Align Feeder`

Pulldown Menu: `DM Electrical->One-Line Diagram->Align Feeder`

You will be prompted to select a feeder segment to which the other feeder segments will be aligned.

Select feeder segment in correct location:

You will then be prompted to select one or more feeder segments to align to the first feeder segment.

Select feeder segment to align with first segment:

Selected feeder segments will be moved to align with the first segment.

Offset Feeder

The `Offset Feeder` command is used to set the amount of space between a feeder segment and parallel segments of other feeders on the one-line diagram.

To offset feeders, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⬚ `Offset Feeder`

Pulldown Menu: `DM Electrical->One-Line Diagram->Offset Feeder`

You will be prompted to select a feeder segment from which the other feeder segments will be offset.

Specify feeder segment in correct location or [set Distance]:

> **Feeder segment:** Specify a feeder segment.

> **Set distance:** Set the distance by which other feeder segments will be offset.

You will then be prompted to select a feeder segment to offset from the first segment.

Select feeder segment to offset from first segment or [set Distance]:

You will then be prompted to specify on which side of the first segment the selected segment will be offset.

Select point on side of first segment to offset:

Specify a point on either side of the first segment. The selected feeder segment will be offset on that side.

You can select the same feeder segment multiple times to increase the offset. If you select a different segment, it will be offset from the previously selected feeder segment. Press **ENTER** to finish the command.

Related Options

Feeder offset multiplier: Sets the offset distance between feeders when the diagram is generated.

There are several other options that affect the overall appearance of the one-line diagram. See the **One-Line Diagram Options** section for more information.

Swap Feeder

The `Swap Feeder` command is used to transpose two parallel feeder segments on the one-line diagram.

To swap feeders, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⬚ `Swap Feeder`

Pulldown Menu: `DM Electrical->One-Line Diagram->Swap Feeder`

You will be prompted to select a feeder segment.

Select first feeder segment to swap:

You will then be prompted to select a feeder segment to transpose with the first feeder segment.

Select second feeder segment to swap:

The two feeder segments will be transposed. The other segments of each feeder will be redrawn if necessary.

Add Segment to Middle of Feeder

To add a segment to the middle of a feeder, go to

Ribbon: `DM Electrical->One-Line Diagram->` `Add Segment to Middle of Feeder`

Pulldown Menu: `DM Electrical->One-Line Diagram->Add Segment to Middle of Feeder`

You will be prompted to identify the location on the feeder where you want a segment to be added.

Select point on feeder to insert segment at:

Select the feeder where the segment is to be added. A new segment will be inserted. The appearance of the feeder will not change, but you will have new grips that you can use to drag the feeder into the correct shape.

Add Segment to End of Feeder

To add a segment to the end of a feeder, go to

Ribbon: `DM Electrical->One-Line Diagram->` `Add Segment to End of Feeder`

Pulldown Menu: `DM Electrical->One-Line Diagram->Add Segment to End of Feeder`

You will be prompted to identify the feeder where a segment is to be added at the end.

Select feeder near end:

Select the feeder near the end where the segment is to be added. A segment will be added to the end of the feeder. The feeder will move slightly to accommodate this new section.

Remove Segment from Feeder

To remove a segment from a feeder, go to

Ribbon: `DM Electrical->One-Line Diagram->` `Remove Segment from Feeder`

Pulldown Menu: `DM Electrical->One-Line Diagram->Remove Segment from Feeder`

You will be prompted to identify the segment to be removed.

Select segment on feeder to remove:

The feeder segment will be removed.

Insert Break in Feeder

To insert a break in a feeder, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⁻⁻ `Insert Break in Feeder`

Pulldown Menu: `DM Electrical->One-Line Diagram->Insert Break in Feeder`

You will be prompted to select where on the feeder to insert the break.

Specify point on feeder to insert break at:

A break will be inserted on the selected location on the feeder.

The break can be moved to a new location or made larger or smaller using the grips on the wire.

Related Options

Wire break block: Sets the block used for the break in the feeder.

Remove Break in Feeder

To remove a break from a feeder, go to

Ribbon: `DM Electrical->One-Line Diagram->` ✗ `Remove Break in Feeder`

Pulldown Menu: `DM Electrical->One-Line Diagram->Remove Break in Feeder`

You will be prompted to identify the broken feeder to be rejoined.

Select feeder near break to remove:

The break in the wire will be removed.

Insert Feeder ID

The `Insert Feeder ID` command is used to insert ID tags for feeders on the drawing.

Each row in the feeder schedule can be assigned a number. The feeder ID will contain the corresponding number. Inserting feeder IDs is an alternative to labeling each feeder with the full feeder callout. The numbers in the feeder IDs will update as rows are added and removed from the feeder schedule. See the **Insert Feeder Schedule** section for more information about inserting the feeder schedule on the drawing.

To insert a feeder ID, go to

Ribbon: `DM Electrical->One-Line Diagram->` ⟨ID⟩ `Insert Feeder ID`

Pulldown Menu: `DM Electrical->One-Line Diagram->Insert Feeder ID`

Select Feeder ID Dialog Box
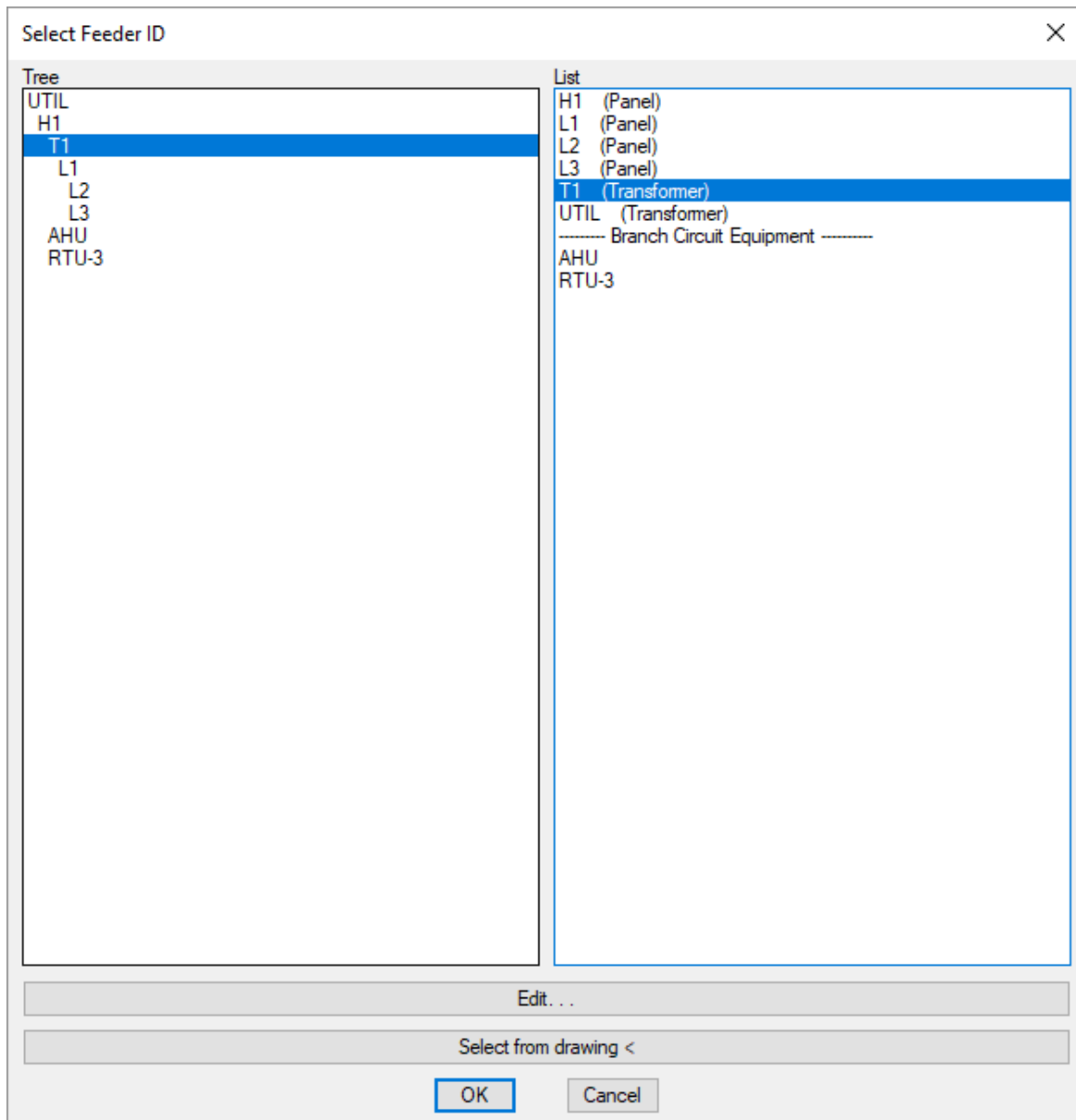
Select from Tree / List

All distribution equipment and equipment connections in the current project are listed in a tree corresponding to how the equipment are connected to each other, and alphabetically by callout.

Press the **Edit** button to open the corresponding dialog box for the selected equipment.

Press the **OK** button to insert the ID for the upstream feeder connected to the selected equipment on the drawing.

You will be prompted for the insertion point.

Select feeder ID location:

The feeder ID will be inserted at that point.

Once the feeder ID is inserted, it can be moved or deleted using standard CAD commands.

Select from Drawing

Press the **Select from Drawing** button to insert the feeder ID for a feeder on the one-line diagram.

You will be prompted to select a feeder on the drawing.

Select feeder:

You will then be prompted to specify a location along the feeder for the ID to be inserted.

Specify feeder ID location:

The feeder ID will be inserted at that point.

Once the feeder ID is inserted, it can be moved or deleted using standard CAD commands.

RT: Import from Revit

This section only applies if your one-line diagram is based upon a Revit model imported using Design Master Electrical RT.

The `Import from Revit` command is used to import information from a Revit project into the current project. This command is used following the `Export One-Line` command in Revit to ensure the information has successfully been transferred.

To import project information from Revit, go to

Ribbon: `DM Electrical->One-Line Diagram->` `RT: Import from Revit`

Pulldown Menu: `DM Electrical->One-Line Diagram->RT: Import from Revit`

The information will be imported from your Revit project.

You will not need to run this command again after the information has been transferred. The project information is updated automatically each time you use a one-line diagram command.

RT: Edit Distribution Equipment

This section only applies if your one-line diagram is based upon a Revit model imported using Design Master Electrical RT.

The `Edit` command is used to change the blocks of distribution equipment and equipment connections on the drawing.

To edit distribution equipment and equipment connections, go to

Ribbon: `DM Electrical->One-Line Diagram->` ✏️ `RT: Edit Distribution Equipment`
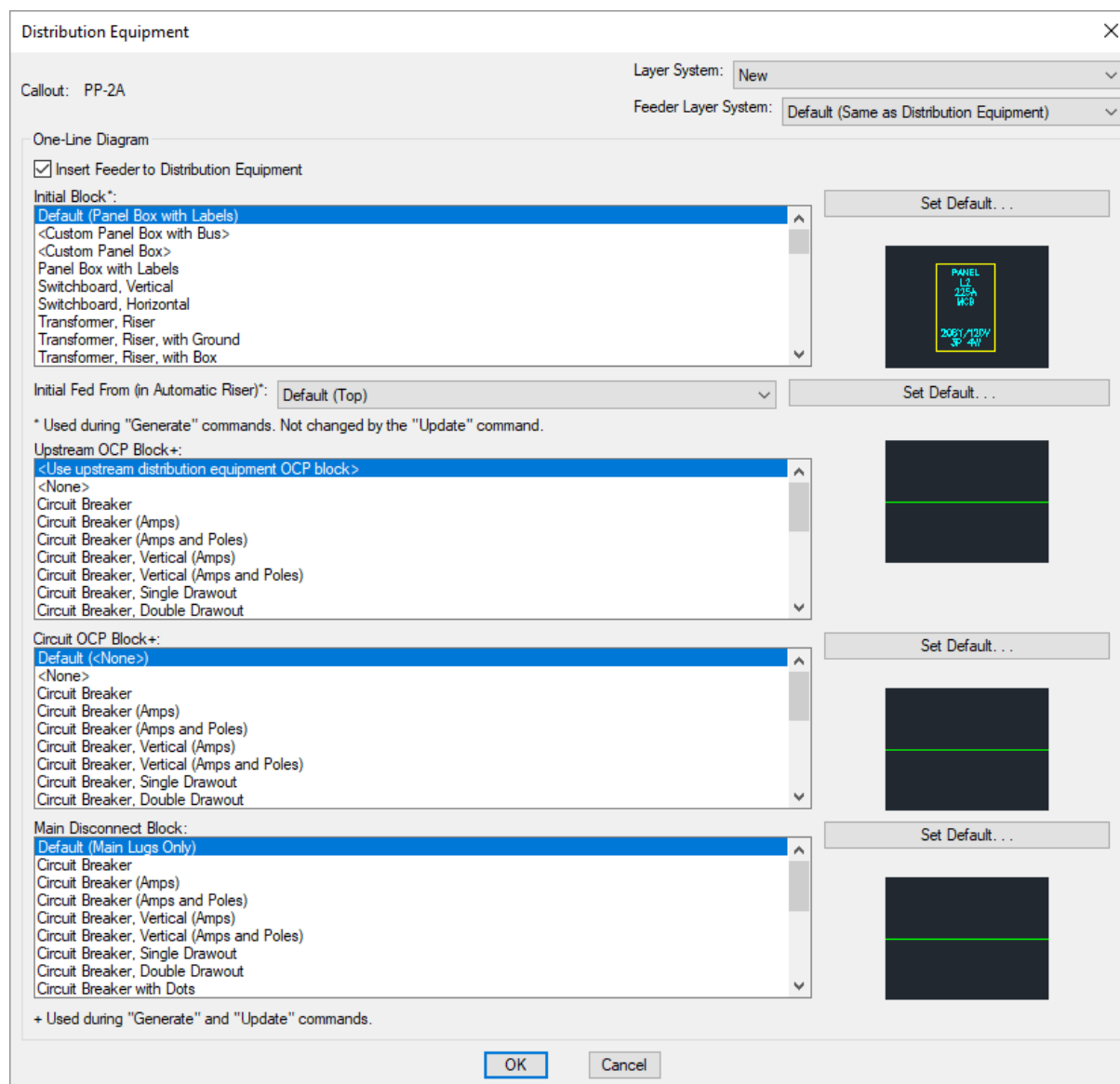
Pulldown Menu: `DM Electrical->One-Line Diagram->RT: Edit Distribution Equipment`

You will be prompted to select the distribution equipment or equipment connection to be changed.

Select distribution equipment or equipment connection to edit:

Specify a distribution equipment or equipment connection. Press **ENTER** to open the **Edit Distribution Equipment** dialog box to select the distribution equipment from a list.

Distribution Equipment Dialog Box

This dialog box sets the blocks used for the distribution equipment and its connections on the one-line diagram.

The **Set Default** button allows you to change the default for each block in the **One-Line Diagram** options section.

Callout: The name of the distribution equipment or equipment connection.

Layer System: The layer system used when the device is inserted on the drawing. See the **Layers** section for more information.

Do not change the layer directly. If you do, it will likely be changed back at some point in the future. Use this dialog box or the **Change Layer System** command.

Feeder Layer System: The layer system used for the feeders when the device is inserted on the drawing. See the **Layers** section for more information.

Do not change the layer directly. If you do, it will likely be changed back at some point in the future. Use this dialog box or the **Change Layer System** command.

Insert Feeder to Distribution Equipment: Whether the upstream feeder going into the distribution equipment is included on the one-line diagram.

Initial Block: The block used for the distribution equipment itself when it is generated. Changing the initial block does not affect blocks already on the one-line diagram.

Initial Fed From: Whether the feeder is drawn to the top or bottom of the block when it is generated. Changing this setting does not affect blocks already on the one-line diagram.

Upstream OCP Block+: The block for the upstream OCP. **<Use upstream distribution equipment OCP block>** uses the same OCP block as the upstream distribution equipment. Changing this setting will affect blocks already on the one-line diagram.

Upstream OCP Block 2+: The block for the second upstream OCP. Applies only to distribution equipment with two feeders.

Circuit OCP Block+: The block for downstream OCP. Changing this setting will affect blocks already on the one-line diagram.

Main Disconnect Block: The block for the main disconnect. Only visible if *Initial Block* is set to **<Custom Panel Box with Bus>**. Not available if *Main Disconnect Type* is set to **Main Lugs Only**. Does not apply to transformers.

Circuiting and Homeruns

This section describes how to connect devices to circuits on distribution equipment and how to draw loops and homeruns to connected devices.

Training Videos

- **The Circuiting Command**
- **Wire Sizing**

Circuiting

The `Circuiting` command is used to manage all of the devices and loads connected to circuits on your distribution equipment.

To manage the circuits on your distribution equipment, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` `Circuiting`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Circuiting`

Circuiting Dialog Box

The **Circuiting** dialog box is divided into two main sections. The left section displays information about and circuits associated with the active distribution equipment. Select the circuit you want to configure from the list. The *Circuit Information* section on the right that allows you to modify the selected circuit.

Distribution Equipment: The active distribution equipment. The circuits listed are associated with the distribution equipment displayed. Changing the equipment will display new circuits in the circuit lists.
Information about the active distribution equipment is displayed below this field. The size, voltage, total connected and calculated load in kVA, and connected load in amps on each phase are all displayed.

Recalculate Panel Load Total: Press this button to recalculate the load total on the active distribution equipment. The total connected and calculated load in kVA and connected load in amps on each phase will be updated to match the current loads connected to the equipment and any distribution equipment connected to it.

Move Circuits: Press this button to open the **Move Circuits** dialog box with the currently selected circuit active.

See the **Move Circuits** section for more information.

Insert Circuit: Press this button to insert a new circuit in the active distribution equipment. A single-pole circuit will be inserted in the circuit list at the location of the active circuit. The active circuit and all of the circuits below it will be shifted down.
If there are no spaces available below the active circuit, no circuit will be inserted.

Remove Circuit: Press this button to remove an unused circuit from the active distribution equipment. All of the circuits below the selected circuit will be shifted up in the circuit list. A new unused circuit will be inserted at the bottom of the circuit list. This button cannot be used if the circuit has any devices connected to it or any fixed loads set in it.

Create Tandem Breaker: Press this button to convert the selected circuit to a tandem breaker.

Remove Tandem Breaker: Press this button to convert a tandem breaker to a circuit.

Circuit Lists: There are two lists of circuits in the **Circuiting** dialog box. Panels will display odd circuits in the list on the left and even circuits in the list on the right. All other distribution equipment types will display their circuits in the list on the left. Only one circuit in the two lists can be selected at the same time. The selected circuit is used in the rest of the dialog box.

Circuit Devices: Press this button to connect devices on the drawing to the currently selected circuit. See the **Circuit Devices** section for more information.

Circuit to Phase: Press this button to connect devices on the drawing to a specific phase of the currently selected circuit. See the **Circuit to Phase** section for more information.

Draw Homerun and Loops: Press this button to draw a homerun and loops on devices on the drawing. See the **Draw Homerun and Loops** section for more information.

Draw Homerun: Press this button to draw a homerun on a device on the drawing. See the **Draw Homerun Only** section for more information.

Circuit Information

Circuit Information: Displays information about the selected circuit. Includes the circuit callout, circuit description, connected and calculated loads in kVA and amps, and the size of the wire.

Change # of Poles: Press this button to change the number of poles for the currently selected circuit. See the **Change # of Poles** section for more information.

Breaker Size: The size of the breaker for the selected circuit. The list of breaker sizes available can be customized. See the **Wire Sizing** section for more information.
- **Size Automatically:** The breaker size is chosen automatically based upon the load connected to the circuit. The size is based upon 125% of the continuous load, and 100% or 125% of the noncontinuous load, depending upon the setting of the *Size breakers to 100% of noncontinuous load* option. See the **Circuits, Load Types, and Wire Sizes** options section for more information.
- **None***:* The circuit does not have a breaker. The breaker column in the distribution equipment schedule will display a slash and the number of poles (/1 or /2 or /3). No breaker size will be displayed.
- **Specific Size***:* The selected breaker size is used for the circuit.

Conductor Amps: Sets the size of the hot wires for the selected circuit.

- **Size Automatically:** The hot wires are sized based upon the *Breaker Size*.
- **None***:* The circuit does not have a conductor wire. The wire callout for the circuit will be blank.
- **Specific Breaker Size***:* The hot wires are sized to match the breaker size chosen from the list.

Neutral: Sets the size of the neutral wire for the selected circuit. If the distribution equipment does not have a neutral, this field is disabled and set to **None**.
- **Size Automatically:** The neutral wire is the same size as the phase wires as specified in the *Conductor Amps* field.
- **None***:* No neutral wire is included in the circuit. If the devices on the circuit require a neutral wire, this option is not available.
- **Specific Breaker Size:** The neutral wire is sized to match the wire size chosen from the list.

Ground: The size of the ground wire for the selected circuit.
- **Size Automatically:** The ground wire is sized based upon the *Equipment Ground* setting for the larger of the two breaker sizes specified in the *Breaker Size* and *Conductor Amps* fields. The ground wire will never be larger than the conductor wires.
- **None***:* No ground wire is included in the circuit.
- **Specific Breaker Size:** The ground wire is sized to match the wire size chosen from the list.

Conduit Size: The size of the conduit for the selected circuit. The conduit fill percentage is displayed next to the *Conduit Size* field. Changing the conduit size will update the fill percentage.
- **Size Automatically:** The conduit is sized automatically based upon the wires in the circuit. All conduits are sized using a 40% conduit fill (*NEC Table 1*).
- **None***:* No conduit is included for the circuit.
- **Specific Breaker Size:** The conduit is sized to match the conduit size chosen from the list.

MCC Starter Size: The motor control center starter size. The value is displayed in the motor control center schedule. Applies only to motor control centers.

MCC Description: The motor control center description. The value is displayed in the motor control center schedule. Applies only to motor control centers.

Layer System: The layer system of the selected circuit. This layer system controls the layer that is used for the circuit callouts, homerun, and loops on the plan. If the selected circuit is shown on the one-line diagram, this layer system will also control the layer of the connected feeder. See the **Layer System** section for more information.
- **Default:** The circuit will use the default circuit layer system. You can change the default layer system using the *Default circuit layer system* setting in the **Circuits, Load Types, and Wire Sizes** options section.
- **<Layer System>:** The circuit will use the selected layer system.

Underground: Whether the circuit is underground. Different layers are used for aboveground and underground loops. The default layers for underground loops use a dashed linetype. The layers can be changed using the **Layers** commands.

General Lighting Circuit: Whether the circuit is considered a general lighting circuit during load calculations.

Starting Phase: Sets the starting phase of the circuit. This value can be used on single-pole and two-pole circuits to balance the loads on distribution equipment.
This field is disabled for panels. The starting phase for each circuit on a panel is fixed.

Description: An additional description for the circuit that is shown on the distribution equipment schedule. This value is added before any other descriptions on the circuit from connected devices. To change the description on the circuit from the devices, change the device definition.

Press the **More** button to open the **More Circuit Description Settings** dialog box. If the selected circuit has more than one pole, the following fields will be available for each phase:

> **Circuit Description Prefix:** Text that is added before the circuit description in the **Circuiting** dialog box.

> **Circuit Description Suffix:** Text that is added after the circuit description in the **Circuiting** dialog box.

> **Circuit Description Replacement:** Text that overrides the circuit description in the **Circuiting** dialog box. This will also override the *Circuit Description Prefix* and *Circuit Description Suffix*.

Set Circuit Information: See the **Set Circuit Information** section for more information.

Select Circuit from Drawing: Press this button to select a circuit based upon a device on the drawing. The dialog box will close and you will be prompted to select a device.

Select device on circuit to make current:

After you select a device, the **Circuiting** dialog box will be opened. The circuit the selected device is connected to will be active.

Find Circuit on Drawing: Press this button to locate devices on the selected circuit. If there are any on the current drawing, they will be highlighted. If there are devices on another drawing but not the current drawing, you will have the option to open the other drawing. This button cannot be used on circuits that do not have any devices connected to them.

Recalculate Circuit: Press this button to recalculate the selected circuit. The load on the circuit will be recalculated based upon the connected devices. The breaker and wire size will be recalculated based upon the newly calculated load.

Recalculate All Circuits on Panel: Press this button to recalculate all of the circuits on the active distribution equipment. The loads on the circuits will be recalculated based upon the connected devices. The breaker and wire sizes will be recalculated based upon the newly calculated loads.

Uncircuit Devices: Press this button to disconnect devices on the drawing from the circuit they are connected to. See the **Uncircuit Devices** section for more information.

Compress Circuits: Press this button to move all of the spaces in the distribution equipment to the bottom of the circuit lists. The order of all of the circuits with loads on them will be the same. Any unused circuits between circuits with loads will be moved.

Related Options

There are several options that affect default values, labels, and behaviors when circuiting devices. See the **Circuits, Load Types, and Wire Sizes Options** section for more information.

Circuit Devices

This section describes the **Circuit Devices** button on the **Circuiting** dialog box. The **Circuit Devices** button connects devices on the drawing to a circuit on a distribution equipment.

If a device is already connected to a circuit, the **Circuit Devices** button can be used to remove it from the

previous circuit and connect it to a new circuit in one action. Using the **Uncircuit Devices** button is not needed.

When you press the button, the **Circuiting** dialog box will close. The circuit that was selected in the *Circuit List* starts as the current circuit. The current circuit can be changed during the command by incrementing the circuit, explained below.

You will be prompted to select the devices to be circuited. The current circuit, the load connected to the circuit, the size of the breaker on the circuit, and the starting phase of the circuit are all listed on the command line.

Select devices to be circuited:
Current Circuit: <Circuit> Connected KVA: <Load> Breaker Amps: <Breaker Size> Phase: <Phase Letter>
Press ENTER once to increment the circuit, twice to return to circuiting dialog.

> **Select device:** Select the devices to be connected to the current circuit. The devices will be connected if the voltage matches the circuit.
>
> A circuit label listing the panel and circuit number will be inserted next to the selected devices. The circuit label will be updated or removed when the homerun and loops are drawn for the circuit.
>
> If you select a device with multiple loads, you will be prompted to specify which loads you want to connect to the current circuit.
>
> If the circuit is a space and the device has more poles than the circuit, the software will attempt to join together circuits below the current circuit to make a circuit large enough for the device. If the circuits below are not spaces, they cannot be joined with the current circuit. The device will not be connected to the current circuit.
>
> If the circuit is a space and the device has fewer poles than the circuit, the circuit will be split into multiple single-pole circuits. The device will be connected to the first circuit. The other circuits will be left as single-pole spaces in the distribution equipment.
>
> **Increment circuit:** Press ENTER once to increment to the next circuit. On panels, the next circuit will be on the same side of the panel as the current circuit. The increment function will not move between the odd and even sides of the panel.
>
> **Return to circuiting dialog:** Press ENTER a second time without circuiting a device to return to the **Circuiting** dialog box.

Circuit to Phase

This section describes the **Circuit to Phase** button on the **Circuiting** dialog box. The **Circuit to Phase** button connects devices on the drawing to a specific phase of a circuit on a distribution equipment. There are two common uses for this button:

- **Shared Neutrals on Multiwire Branch Circuits:** *NEC 2008 210.4b* requires multiwire branch circuits that share a neutral to have the breakers all trip at the same time. Using this button, you can connect single-pole devices to a specific phase of a two-pole or three-pole circuit. All of the devices on the circuit will share a common neutral and have a common trip.
- **208V and 480V Site Lighting on Three-Phase Circuits:** It is common to connect two-pole site lights to three-pole circuits, rotating the phases that are connected to keep the load balanced. Using this button, you can connect two-pole light fixtures to specific phases of a three-pole circuit.

When you press the button, the **Circuiting** dialog box will close. The circuit that was selected in the *Circuit List* starts as the current circuit. The current circuit can be changed during the command by incrementing the circuit,

explained below.

The **Select Circuit Phase** dialog box will appear. You are prompted to select the phase of the circuit to which devices will be connected.

Select Circuit Phase

A (H1-7)
B (H1-9)
C (H1-11)

OK   Cancel
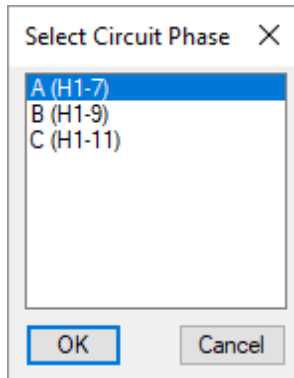
After you select the phase, you will be prompted to select the devices to be circuited. The current circuit, the load connected to the circuit, the size of the breaker on the circuit, and the starting phase of the circuit are all listed on the command line.

Select devices to be circuited:
Current Circuit: <Circuit> Connected KVA: <Load> Breaker Amps: <Breaker Size> Phase: <Phase Letter>
Press ENTER for circuiting options, twice to return to circuiting dialog.

Select device: Select the devices to be connected to the selected phase on the current circuit. The devices will be connected if the voltage matches the circuit.

A circuit label listing the panel and circuit number will be inserted next to the selected devices. The circuit label will be updated or removed when the homerun and loops are drawn for the circuit.

If you select a device with multiple loads, you will be prompted to specify which loads you want to connect to the current circuit.

If the circuit is a space and the device has more poles than the circuit, the software will attempt to join together circuits below the current circuit to make a circuit large enough for the device. If the circuits below are not spaces, they cannot be joined with the current circuit. The device will not be connected to the current circuit.

If the device has fewer poles than the circuit, the device is connected to the selected phase on the current circuit. The number of poles on the circuit is not changed.

Circuiting options: Press **ENTER** once to change the current phase or circuit.

[Set phase/Increment circuit/<Return to circuiting dialog>]:

Set phase: Type **s** to select a new phase using the **Select Circuit Phase** dialog box, shown above.

Increment circuit: Type **I** to increment to the next circuit. On panels, the next circuit will be on the same side of the panel as the current circuit. The increment function will not move between the odd and even sides of the panel.

Return to circuiting dialog: Press ENTER a second time or type R to return to the **Circuiting** dialog box.

Change # of Poles

This section describes the **Change # of Poles** button on the **Circuiting** dialog box and other dialog boxes. The **Change # of Poles** button is used to change the number of poles for a selected circuit.

To comply with *NEC 2008 210.4b*, this button can be used to merge three single-pole circuits into one three-pole circuit. If there are devices on the single-pole circuits, they will remain on the corresponding phases of the three-pole circuit. The circuits will now share a common neutral and a common trip.

When you press the **Change # of Poles** button, the **Change Number of Poles** dialog box will appear.
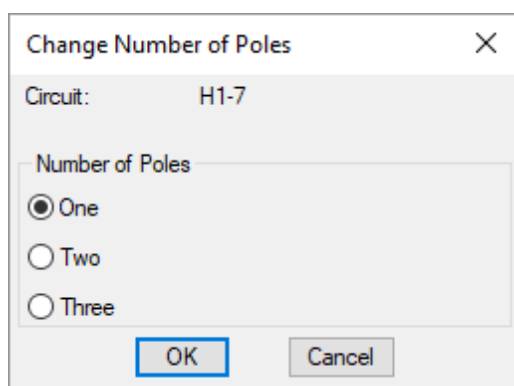
Circuit: The selected circuit.

Number of Poles: Select the number of poles for the circuit. Options that are not allowed for the current circuit will be disabled.

Set Circuit Information

This section describes the **Set Circuit Information** button on the **Circuiting** dialog box. The **Set Circuit Information** button is used to set how the circuit length is calculated, the circuit description, and the fixed load types and connections on the circuit.

When you press the **Set Circuit Information** button, the **Set Circuit Information** dialog box will appear.

Current Circuit: The selected circuit. Use the **Previous** and **Next** buttons to select a different circuit.

Length Calculation

Circuit Length Calculation Method: Sets how the circuit length is calculated. The values you can set vary depending upon the setting of this field.

- **Default:** The default calculation method for the project. The default *Circuit length calculation method* setting can be changed in the **Distribution Equipment** options section.
- **Right Angles:** Calculate the circuit length using line segments at right angles to one another. The direction of the segments is determined by the *Building Angle*.
- **Straight Line:** Calculate the circuit length using a straight line.
- **Custom:** Set the circuit length manually.

Average Circuit Length: The calculated average length of the circuit to each device in feet and inches. This can be set manually when *Circuit Length Calculation Method* is set to **Custom**.

Building Angle: Sets the building angle relative to the alignment point when *Circuit Length Calculation Method* is set to **Right Angles**. The default *Building angle* setting can be changed in the **Distribution Equipment** options section. See the **How Building Angle Affects Calculations** article in the knowledge base for more information about this setting.

Farthest Device Length: The length of the circuit to the farthest device in feet and inches.

Make-Up

Branch Circuit Make-Up: The extra length of wire included in branch circuit lengths to account for branch circuit connections. It is used in the fault calculation, voltage drop calculation, and takeoffs.
- **Default:** The default make-up length for the distribution equipment. This value can be changed in the **Schedule and Length Settings** dialog box for the distribution equipment.
- **Custom:** Set the make-up length manually.

Add Branch Circuit Make-Up For Each Device: Whether the length specified in *Branch Circuit Make-Up* is added for each device on the circuit.
- **Default:** The default setting for the distribution equipment. This setting can be changed in the **Schedule and Length Settings** dialog box for the distribution equipment.
- **Yes:** The make-up added to the branch circuit length is multiplied for each device on the circuit.
- **No:** The make-up is only added to the branch circuit length once, regardless of the number of devices on the circuit.

Circuit Description Prefix: Text that is added before the circuit description in the **Circuiting** dialog box.

Circuit Description Suffix: Text that is added after the circuit description in the **Circuiting** dialog box.

Circuit Description Replacement: Text that overrides the circuit description in the **Circuiting** dialog box. This will also override the *Circuit Description Prefix* and *Circuit Description Suffix*.

Fixed Loads

Fixed loads are loads on a circuit that are not associated with a device on a drawing and must be specified manually. Loads are input in kVA. Existing loads are often modeled using fixed loads. Fixed load values are added to the load of any devices connected to the circuit.

See the **Electrical Load Type Explanations** article on the knowledge base for more information about each load type and when it should be used. See the **Inserting Fixed Loads Without Using the Mouse** article in the knowledge base for keyboard shortcuts available for entering fixed loads.

Lighting: Sets the fixed lighting load on the circuit.

Largest Motor: Sets the fixed largest motor load on the circuit. If a circuit has multiple motors connected to it, specify the load of the largest motor here. This will not necessarily be the largest motor on the distribution equipment or in the project. When the load on a distribution equipment is calculated, the largest of all of the largest motor loads will be used as the largest motor on the distribution equipment.

Other Motors: Sets the fixed other motor loads on the circuit. This load can represent multiple motors. The total can be larger than the largest motor. If a largest motor load is included on the circuit, this value will always be added to the other motors load for all of the distribution equipment it is connected to. If no largest motor load is included on the circuit, this value is treated as the largest motor load.

Receptacles: Sets the fixed non-dwelling receptacle load on the circuit.

Continuous: Sets the continuous load on the circuit.

Heating: Sets the fixed space-heating load on the circuit.

Cooling, Largest Motor: Sets the fixed load for the largest motor used for cooling on the circuit. If a circuit has multiple motors connected to it, specify the load of the largest motor here. This will not necessarily be the largest motor on the distribution equipment or in the project. When the load on a distribution equipment is calculated,

the largest of all of the largest motor loads will be used as the largest motor on the distribution equipment.

Cooling, Other Motors: Sets the fixed loads for other motors used for cooling on the circuit. This load can represent multiple motors. The total can be larger than the largest motor. If a largest motor load is included on the circuit, this value will always be added to the other motors load for all of the distribution equipment it is connected to. If no largest motor load is included on the circuit, this value is treated as the largest motor load.

Heating and Cooling Motor: Sets the fixed heating and cooling motor load on the circuit.

Noncontinuous: Sets the noncontinuous load on the circuit.

Kitchen: Sets the fixed kitchen load on the circuit.

> **Kitchen (Multiplier):** Sets the number of pieces of kitchen equipment on the circuit. The kitchen demand factor in *NEC Table 220.56* is based upon the number of pieces of kitchen equipment.

Diversified: Sets the fixed diversified load on the circuit.

> **Diversified (Multiplier):** Sets the multiplier for the calculated load of the diversified load.

Custom Load 1: Sets the fixed connected load for *Custom Load Type 1* on the circuit. The label on the dialog box will match the *Custom Load Type 1 Name* option. See the **Circuits, Load Types, and Wire Sizes** options section for more information.

> **Custom Load 1 (Demand):** Sets the fixed calculated load for *Custom Load Type 1* on the circuit. This field is enabled if the *Custom Load Type 1 Calculation method* option is set to **Variable calculated load**.

> **Custom Load 1 (Multiplier):** Sets the demand factor used to calculate the calculated load for *Custom Load Type 1* on the circuit. This field is enabled if the *Custom Load Type 1 Calculation method* option is set to **Variable calculated load**.

Custom Load 2: Sets the fixed load for *Custom Load Type 2* on the circuit. The label on the dialog box will match the *Custom Load Type 2 Name* option. See the **Circuits, Load Types, and Wire Sizes** options section for more information.

> **Custom Load 2 (Demand):** Sets the fixed calculated load for *Custom Load Type 2* on the circuit. This field is enabled if the *Custom Load Type 2 Calculation method* option is set to **Variable calculated load**.

> **Custom Load 2 (Multiplier):** Sets the demand factor used to calculate the calculated load for *Custom Load Type 2* on the circuit. This field is enabled if the *Custom Load Type 2 Calculation method* option is set to **Variable calculated load**.

Small Appliance: Sets the fixed small appliance load on the circuit.

Laundry: Sets the fixed laundry load on the circuit.

Appliance, Continuous: Sets the continuous load for appliances fastened in place on the circuit.

> **Appliance, Continuous (Multiplier):** Sets the number of continuous appliances on the circuit.

Appliance, Noncontinuous: Sets the noncontinuous load for appliances fastened in place on the circuit.

Appliance, Noncontinuous (Multiplier): Sets the number of noncontinuous appliances on the circuit.

Appliance Motor: Sets the fixed load for appliances that are also motors on the circuit.

Electric Dryer: Sets the fixed electric dwelling unit dryer load on the circuit.

Electric Dryer (Multiplier): Sets the number of electric dwelling unit dryers on the circuit.

Electric Cooking: Sets the fixed household electric cooking appliance load on the circuit.

Electric Dryer (Multiplier): Sets the number of pieces of household electric cooking appliances on the circuit.

Dwelling Unit Heating: Sets the fixed dwelling unit space-heating load on the circuit.

Heating (Multiplier): Sets the number of pieces of heating equipment on the circuit.

Dwelling Unit Cooling, Largest Motor: Sets the fixed load for the largest motor used for cooling a dwelling unit on the circuit. If a circuit has multiple motors connected to it, specify the load of the largest motor here. This will not necessarily be the largest motor on the distribution equipment or in the project. When the load on a distribution equipment is calculated, the largest of all of the largest motor loads will be used as the largest motor on the distribution equipment.

Dwelling Unit Cooling, Other Motors: Sets the fixed loads for other motors used for cooling a dwelling unit on the circuit. This load can represent multiple motors. The total can be larger than the largest motor. If a largest motor load is included on the circuit, this value will always be added to the other motors load for all of the distribution equipment it is connected to. If no largest motor load is included on the circuit, this value is treated as the largest motor load.

Dwelling Unit Heating and Cooling Motor: Sets the fixed dwelling unit heating and cooling motor load on the circuit.

Marina / Mobile Home / RV: Sets the fixed load for marina receptacles, mobile home lots, or RV sites on the circuit.

Marina / Mobile Home / RV (Multiplier): Sets the number of receptacles, lots, or sites on the circuit.

Clear Circuit Descriptions and Loads: Clears all fixed loads and description fields for the circuit.

Related Options

Circuit length calculation method: Sets the default for the *Circuit Length Calculation Method* setting.

Building angle: Sets the default value for the *Building Angle* field.

Custom Load Types 1 and 2: Configures settings for *Custom Load Type 1* and *Custom Load Type 2*.

Uncircuit Devices

This section describes the **Uncircuit Devices** button on the **Circuiting** dialog box. The **Uncircuit Devices** button

disconnects devices on the drawing from the circuit to which they are connected. The circuit that is currently selected in the circuit list is not used in this command.

You do not need to use this button to when moving devices from one circuit connection to another. Instead, use the **Circuit Device** button to connect the device to the new circuit. The device will be removed from the circuit it is currently connected to before being connected to the new circuit.

You do not need to use this button before deleting a device. When the device is deleted, it will be removed from the circuit it is connected to.

Only use this button when you want to disconnect a device from all circuits but still have it on your drawing.

When the button is pressed, the **Circuiting** dialog box is closed. You are prompted to select the devices to disconnect from a circuit.

Select objects:

All of the devices that you select will be disconnected from the circuit to which they are connected. The circuit labels, homeruns, and loops associated with these devices will be removed from the drawing.

You can uncircuit any device on the drawing. You are not limited to devices on the selected circuit or the selected distribution equipment in the **Circuiting** dialog box.

Move Circuits

The `Move Circuits` command is used to rearrange circuits on distribution equipment.

To move circuits between distribution equipment, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` `Move Circuits`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Move Circuits`

Move Circuits Dialog Box

The **Move Circuits** dialog box is divided into left and right sections that correspond to two separate distribution equipment. The duplicate buttons work the same in both sections and each will only affect the selected circuit for that section. The buttons that span the entire dialog box affect both sections.

Distribution Equipment: Sets the active distribution equipment in each section. The circuits in the circuit list correspond to the active distribution equipment. The voltage of the distribution equipment and the selected circuit are listed below this field.

Insert Circuit: Press this button to insert a new circuit in the active distribution equipment. A single-pole circuit will be inserted in the circuit list at the location of the selected circuit. The selected circuit and all of the circuits below it will be shifted down.

If there are no spaces available below the selected circuit, no circuit will be inserted.

Compress Circuits: Press this button to move all of the spaces in the active distribution equipment to the bottom of the circuit lists. The order of all of the circuits with loads on them will be the same. Any unused circuits between circuits with loads will be moved.

Remove Circuit: Press this button to remove an unused circuit from the active distribution equipment. All of the circuits below the selected circuit will be shifted up in the circuit list. A new unused circuit will be inserted at the bottom of the circuit list. This button cannot be used if the circuit has any devices connected to it or any fixed loads set in it.

Change # of Poles: Press the button to change the number of poles for the selected circuit. See the **Change # of**

147

Circuiting: Press this button to open the **Circuiting** dialog box with the selected circuit in the left circuit list active. See the **Circuiting** section for more information.

Circuit List: Sets the active circuit on the active distribution equipment. For panels, the odd circuits are shown at the top of the list and the even circuits are at the bottom of the list. For all other distribution equipment, all of the circuits are shown in the list in order.

Swap: Press this button to swap the selected circuits in the left and right circuit lists. The two circuits must have the same voltage and number of poles. The connected devices and fixed loads on each circuit will be swapped. Any circuit labels on the drawing will automatically be updated with the new circuit number.

Copy: Press this button to copy the wire size and breaker customization, as well as fixed loads, from the selected circuit in the left circuit list to the selected circuit in the right circuit list. All of the customized wire size and breaker settings, as well as the fixed loads, will be copied.

After the fixed loads are copied, the next circuit in the right circuit list will be selected. This allows you to continue pressing the **Copy** button to copy the same fixed loads to multiple circuits in a row without selecting each circuit manually.

Merge: Press this button to merge the connected devices and fixed loads from the selected circuit in the right circuit list with the selected circuit in the left circuit list. The two circuits must have the same voltage and number of poles. The connected devices and fixed loads will be removed from the selected circuit in the right circuit list and connected to the circuit in the left circuit list.

Draw Homerun and Loops

The `Draw Homerun and Loops` command is used to draw a homerun and loops between devices that are connected to a circuit. This command can be used to draw homeruns and loops for a single circuit or for a combined circuit.

Loops will be drawn on the shortest possible path to minimize wire lengths.

If you run the command on devices that already have loops drawn between them, any loops that already exist on the drawing will not be modified. Any changes in the position of loops or loop breaks will be retained. Any loops that are no longer needed will be erased.

To draw both the homerun and the loops, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` `Draw Homerun and Loops`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Draw Homerun and Loops`

You will be prompted to select the device from which to draw the homerun.

Select device from which to draw homerun:

You will then be prompted to select other circuited devices to add to the homerun.

Select other devices with circuits to add to homerun:

If you select additional devices on different circuits, a combined homerun that includes all of the circuits will be inserted. Circuits on a combined homerun must all have the same wire size. You cannot combine circuits that use different size wires. The neutral will be shared based upon the **Share Neutral on Combined Homeruns** setting.

If *Select Preferred Circuiting Direction* is set to **Yes**, you will be prompted to specify the preferred circuiting angle. See the **Select Preferred Circuiting Direction** section for more information.

Input preferred circuiting angle / <None>:

If *Select Loop Center Point* is set to **Yes**, you will be prompted to specify the center point for the arcs. See the **Select Loop Center Point** section for more information.

Arc center point / <None>:

Loops will be inserted between all of the devices on the selected circuits.

Right click or press space to change HR arc direction.

Position the homerun on the drawing. Right-click or press **SPACE** to flip the arc direction of the homerun.

Related Options

Hot or neutral first: Sets whether the hot or neutral tick marks are drawn closest to the homerun arrowhead.

Ground location: Sets whether the ground tick mark is inserted closest to or furthest from the homerun arrowhead.

Homerun wire callouts and tick marks: Sets when wire callouts and/or tick marks are displayed on homeruns.

Display tick marks on loops: Sets whether tick marks are displayed on loops.

Multiple arrowheads on combined homeruns: Sets whether combined homeruns display an arrowhead for each circuit.

Homerun arrow block: Sets the block used for homerun arrowheads

This option can be further customized with the **Homerun arrow length** option.

Tick Mark Blocks options: Set the blocks used for tick marks and the space between tick marks.

Set Looping Options

The `Set Looping Options` command is used to set options used when drawing circuit loops and homeruns.

To change the options used when drawing loops and homeruns, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` ^{CIR OPTS} `Set Looping Options`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Set Looping Options`

Looping Options Dialog Box

Draw Loops Between Circuited Devices: Controls whether loops are drawn between devices on the circuit and how circuit callouts are displayed. This setting controls a number of related options, detailed in the table below. If the cell is green, the feature for that column is enabled. If the cell is red, the feature for that column is disabled. For example, option 2 enables drawing loops between devices and disables all circuit callouts, while option 7 enables all features.

Loops are not required to calculate voltage drop and wire lengths in takeoffs, but the lengths will be estimated. If you want more accurate calculations but don't want loops on the final printed drawing, turn loops and circuit

callouts on and insert them on a non-plotting layer.

Option #	Draw Loops Between Devices	Circuit Callout	Panel Name in Callout	Callout on Combined Homeruns	Callout on Switches
1	🟩	🟥	🟥	🟩	🟥
2	🟩	🟥	🟥	🟥	🟥
3	🟩	🟩	🟩	🟩	🟥
4	🟩	🟩	🟥	🟩	🟥
5	🟥	🟩	🟩	🟩	🟥
6	🟥	🟩	🟥	🟩	🟥
7	🟩	🟩	🟩	🟩	🟩
8	🟩	🟩	🟥	🟩	🟩
9	🟥	🟩	🟩	🟩	🟩
10	🟥	🟩	🟥	🟩	🟩

Share Neutrals on Combined Homeruns: Whether neutrals are shared on combined homeruns. This must be set to **No** if you are using the 2008 NEC.

To share neutrals in accordance with the 2008 NEC, a common trip for the circuits on the combined homerun is required.

One way to create a common trip for the circuits is to create three single-pole circuits and join them using the **Change # of Poles** button on the **Circuiting** dialog box. See the **Change # of Poles** section for more information.

The other way to create a common trip for the circuits is to create a three-pole circuit and connect single-pole devices to the circuit using the **Circuit to Phase** button on the **Circuiting** dialog box. See the **Circuit to Phase** section for more information.

Select Preferred Circuiting Direction: Whether the preferred circuiting angle prompt is used when drawing homeruns and loops.

The preferred circuiting angle is used to guide how loops are drawn between devices. When possible, loops are drawn parallel to the circuiting direction specified.

Select Loop Center Point: Whether the arc center point prompt is used when drawing homeruns and loops.

The center point is used to help automatically draft the loops in an organized manner. All of the loops that are inserted will arc away from the center point.

Loop to Same Circuit First on Combined Homeruns (Light Fixtures): For combined homeruns, whether loops are drawn to the closest light fixture on the same circuit or the closest light fixture on any circuit.

Loop to Same Circuit First on Combined Homeruns (Other Devices): For combined homeruns, whether loops are drawn to the closest device on the same circuit or the closest device on any circuit.

Related Options

Draw loops: Sets the default for the *Draw Loops Between Circuited Devices* setting.

Loop to same circuit first on combined homeruns: Sets the defaults for the *Loop to Same Circuit First on Combined Homeruns* settings.

Share neutrals on combined homeruns: Sets the default for the *Share Neutrals on Combined Homeruns* setting.

Draw Homerun Only

The `Draw Homerun Only` command is used to draw only the homerun for a circuit. This command can be used to move the homerun from one device to another without affecting the loops that are already inserted on the drawing.

To draw a homerun only without drawing the loops associated with the circuit, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` ↗ `Draw Homerun Only`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Draw Homerun Only`

You will be prompted to select the device from which to draw the homerun.

Select device from which to draw homerun:

You will then be prompted to select other devices with circuits to add to the homerun.

Select other devices with circuits to add to homerun:

If you select additional devices on different circuits, a combined homerun that includes all of the circuits will be inserted. Circuits on a combined homerun must all have the same wire size. You cannot combine circuits that use different size wires. The neutral will be shared based upon the **Share Neutral on Combined Homeruns** setting.

Right click or press space to change HR arc direction.

Position the homerun on the drawing. Right-click or press `SPACE` to flip the arc direction of the homerun.

Draw Additional Homerun

To draw an additional homerun, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` [icon] `Draw Additional Homerun`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Draw Additional Homerun`

You will be prompted to select the device from which to draw the homerun.

Select device from which to draw homerun:

You will then be prompted to position the homerun on the drawing

Right click or press space to change HR arc direction.

Right-click or press **SPACE** to flip the arc direction of the homerun.

Automatic Homerun

To draw multiple homeruns on a drawing simultaneously, go to

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Automatic Homerun`

You will be prompted to identify the devices for which homeruns will be drawn.

Select devices to homerun:

You will then be prompted to specify the insertion angle for the homeruns.

Select angle for homeruns to be inserted at / <Point toward panels>:

> ***Select angle for homeruns to be inserted at:*** All of the homeruns will be drawn pointing in the direction specified.

> ***Point toward panels:*** All of the homeruns will be drawn pointing toward the panel they are connected to.

Homeruns and loops will be drawn for all of the circuited devices you select. Any devices you did not select that are circuited to a selected device will have loops drawn to them. The homerun will be drawn from the device closest to the panel the circuit is connected to.

Related Options

Hot or neutral first: Sets whether the hot or neutral tick marks are drawn closest to the homerun arrowhead.

Ground location: Sets whether the ground tick mark is inserted closest to or furthest from the homerun arrowhead.

Homerun wire callouts and tick marks: Sets when wire callouts and/or tick marks are displayed on homeruns.

Display tick marks on loops: Sets whether tick marks are displayed on loops.

Multiple arrowheads on combined homeruns: Sets whether combined homeruns display an arrowhead for each circuit.

Homerun arrow block: Sets the block used for homerun arrowheads

This option can be further customized with the **Homerun arrow length** option.

Tick Mark Blocks options: Set the blocks used for tick marks and the space between tick marks.

Calculate Voltage Drop

The `Calculate Voltage Drop` command is used to calculate and display the voltage drop on a circuit.

To calculate the voltage drop of a circuit, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` **VOLT DROP** `Calculate Voltage Drop`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Calculate Voltage Drop`

You will be prompted to select a device on the circuit for which the voltage drop is to be calculated.

Select device on circuit:

After you select a device, a **Design Master Electrical** dialog box will appear that displays the voltage drop on the circuit. The voltage drop is also printed on the command line.
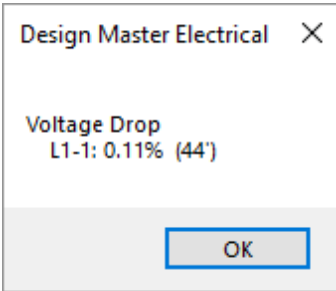
The calculations for the distance and voltage drop are dependent upon whether loops have been drawn for the devices on the circuit.

Loops Are Drawn

The voltage drop is calculated based upon the loops on the drawing. The load on each loop is adjusted based upon the specific devices that are connected to the loop.

The distance displayed is the length of the longest run of wire on the circuit. It does not necessarily correspond to the path of the highest voltage drop.

The distance between devices is calculated based upon the *Circuit Length Calculation Method* for the circuit. See the **Set Circuit Information** section for more information.

No Loops Are Drawn

The distance displayed is the average length of the circuit to each device, weighted by the load on each device. The voltage drop is calculated based upon this distance.

Draw Tick Marks

The `Draw Tick Marks` command is used to update the tick marks on existing loops on the drawing. The new tick marks will account for any changes made to the path of the loops by adding or removing loops manually.

To draw tick marks on existing loops, go to

Ribbon: DM Electrical->Circuiting and Homeruns-> DRAW TICKS Draw Tick Marks

Pulldown Menu: DM Electrical->Circuiting and Homeruns->Draw Tick Marks

You will be prompted to select a device on the circuit where tick marks are to be updated.

Select device on circuit:

The tick marks on the loops connected to the selected device will be updated.

<div style="background-color:#d3d3d3;">

Related Options

Hot or neutral first: Sets whether the hot or neutral tick marks are drawn closest to the homerun arrowhead.

Ground location: Sets whether the ground tick mark is inserted closest to or furthest from the homerun arrowhead.

Display tick marks on loops: Sets whether tick marks are displayed on loops.

Tick Mark Blocks options: Set the blocks used for tick marks and the space between tick marks.

</div>

Modify Circuit Label

The `Modify Circuit Label` command is used to insert, modify, or remove the circuit label on devices on the drawing. Changing the label does not affect what the device is connected to. It only changes how the label is displayed on the drawing.

To modify the circuit label on a device, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` CIR LABEL `Modify Circuit Label`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Modify Circuit Label`

Modify Circuit Label Dialog Box
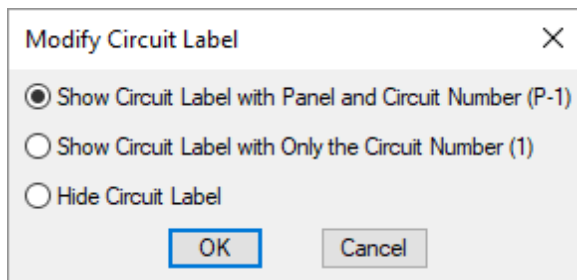
Show Circuit Label with Panel and Circuit Number (P-1): The circuit label will include the distribution equipment callout and the circuit number, separated by a dash. This label is the default label that is inserted when devices are first connected to a circuit.

Show Circuit Label with Only the Circuit Number (1): The circuit label will include only the circuit number. The distribution equipment callout is not shown.

Hide Circuit Label: No circuit label is shown on the device.

Select the desired option and press the **OK** button.

You will be prompted to specify which devices on the drawing to modify the circuit labels for.

Select objects:

The circuit labels on the selected objects will be updated to match the chosen setting. If the device is not connected to a circuit, no label will be displayed.

Move Loop or Homerun

To move a homerun or loop after a circuit has been looped, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` ⬆ `Move Loop or Homerun`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Move Loop or Homerun`

You will be prompted to select the loop or homerun to move.

Select loop or homerun to move:

Move the loop or homerun to the new location.

Insert Tick Marks

The `Insert Tick Marks` command is used to manually insert tick marks on a loop or homerun. These tick marks will replace the tick marks that already exist on the loop.

To manually insert tick marks, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` ⊞ `Insert Tick Marks`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Insert Tick Marks`

Circuit Tick Marks Dialog Box

This dialog box allows the user to specify how tick marks are displayed on the drawing. The far left column is the

first tick mark closest toward the homerun.

Hot: Inserts the hot in the order specified.

Neutral: Inserts the neutral in the order specified.

Ground: Inserts the ground in the order specified.

Iso Ground: Inserts the isolated ground in the order specified.

Switch Leg: Inserts the switch leg in the order specified.

Traveller: Inserts the traveler in the order specified.

Space: Inserts a space. Spaces can be inserted between tick marks to allow for grouping and easier readability.

Reverse: Press this button to reverse the order of the tick marks. Spaces will not be affected.

Reset All to "Space": Press this button to set all of the columns to **Space**.

Set the order for the tick marks to be inserted and press the **OK** button.

You will be prompted to identify the loop to which to add tick marks.

Select loop to which to add tick marks:

Ticks marks will be inserted on the loop you select. Any tick marks that already exist on the loop will be removed. You will be prompted to selected additional loops to which to add tick marks.

Related Options

Tick Mark Blocks options: Set the blocks used for tick marks and the space between tick marks.

Break Circuit

To break a circuit loop graphic without uncircuiting the devices connected by the loop, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` ⊣⊢ `Break Circuit`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Break Circuit`

You will be prompted to identify the first break point on the loop to be broken. Select the loop at the point where the break should begin.

Select circuit to break at first break point:

You will then be prompted to select on the loop where the break should end.

Select second break point:

The loop will be broken at the two selected points. The broken loops will be grouped with the device they are connected to. To remove the pieces of the broken loop without deleting the devices, use the **Erase Part of Device** command.

Related Options

Circuit break block: Sets the block used for the break in the circuit.

Join Circuit

To rejoin a circuit that was broken using the **Break Circuit** command, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` ⌢ `Join Circuit`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Join Circuit`

You will be prompted to select a broken loop to rejoin.

Select circuit to join:

The two pieces of the broken loop will be removed and replaced with a connected loop between the two devices.

Loop Devices Manually

The `Loop Devices Manually` command is used to manually force a loop between two devices. The two devices will be connected without regard to whether they are connected to the same circuit on a distribution equipment.

To manually insert a loop between two devices, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` ⌃ `Loop Devices Manually`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Loop Devices Manually`

You will be prompted to identify the first device to loop.

Select device to loop from:

You will then be prompted to identify the device to which the first device is to be looped.

Select device to loop to:

Select the next device. A loop will be drawn between the two devices.

Select device to loop to:

Continue to insert loops. Each time you select a device, a loop between the selected device and the last device will be drawn. Press **ENTER** to finish the command.

Loops inserted manually will not have tick marks. To insert them manually, see the **Insert Tick Marks** section. To insert them automatically, see the **Draw Ticks Marks** section.

The voltage drop calculated for the circuit will need to be updated to account for the new loops. See the **Calculate Voltage Drop** section for more information.

Highlight Uncircuited Devices

The `Highlight Uncircuited Devices` command is used to highlight all of the devices on the current drawing that have not yet been circuited.

To highlight uncircuited devices, go to

Ribbon: `DM Electrical->Circuiting and Homeruns->` FIND UNCIR `Highlight Uncircuited Devices`

Pulldown Menu: `DM Electrical->Circuiting and Homeruns->Highlight Uncircuited Devices`

A line will be drawn from the origin to each device that is not circuited. The lines can be erased using standard CAD commands when you no longer need them.

Running the command a second time on a drawing will remove any lines pointing to devices that are now circuited.

Modify Devices

This section describes the commands available for modifying devices on the drawing.

Query Device

The `Query Device` command allows you to select any device and view and edit the information associated with it. It will work with distribution equipment, light fixtures, receptacles, equipment, switches, junction boxes, low voltage devices, conduits, and cable trays.

To query a device, go to

Ribbon: `DM Electrical->Devices->` DEV Q `Query Device`

Pulldown Menu: `DM Electrical->Modify Devices->Query Device`

You will be prompted to select a device to query.

Select device to query:

After you select a device, a dialog box will open displaying the information about the selected device. This dialog box can be used to both view and edit the information related to the device.

The specific dialog boxes are described in the following sections:
- **Querying Distribution Equipment**
- **Querying Light Fixtures**
- **Querying Receptacles**
- **Querying Equipment**

Array Device (2 Point)

The `Array Device (2 Point)` command is used to array devices on the drawing in a straight line from the first device to a second point.

Use this command instead of the standard CAD **ARRAY** command.

To array devices using the 2-point method, go to

Ribbon: DM Electrical->Devices-> Array Device (2 Point)

Pulldown Menu: DM Electrical->Modify Devices->Array Device (2 Point)

You will be prompted to select the devices to array.

Select devices to array:

You will then be prompted to specify a second point for the array. The distance between the specified point and the first device will be divided evenly among the devices in the array.

End point of array:

You will then be prompted to specify the number of devices in the array, including the selected device.

Number of devices in array (including selected device):

The devices in the array will be generated in the space between the selected device and the end point.

Array Device (Distance / Angle)

The `Array Device (Distance/Angle)` command is used to array devices on the drawing in a straight line from the first device in a set direction with a set distance between each device.

Use this command instead of the standard CAD **ARRAY** command.

To array a device using the distance/angle method, go to

Ribbon: DM Electrical->Devices-> Array Device (Distance/Angle)

Pulldown Menu: DM Electrical->Modify Devices->Array Device (Distance/Angle)

You will be prompted to select the devices to array.

Select devices to array:

You will then be prompted to set an offset distance and angle. The distance between each device in the array is based upon the distance from the first device to the specified point.

Offset distance and angle:

You will then be prompted to specify the number of devices in the array, including the selected device.

Number of devices in array (including selected device):

The devices in the array will be generated in the direction of the specified point from the selected device.

Copy and Increment Circuits

The `Copy and Increment Circuits` command allows you to copy a device that is circuited and have the copied device also be circuited.

To copy a device in this manner, go to

Ribbon: DM Electrical->Devices-> ++ Copy and Increment Circuits

Pulldown Menu: DM Electrical->Modify Devices->Copy and Increment Circuits

You will be prompted to select the devices to be copied.

Select objects:

You will then be prompted to specify a location for the new devices.

Specify base point:
Specify second point:

The **Increment Circuits** dialog box will appear.

Increment Circuits Dialog Box
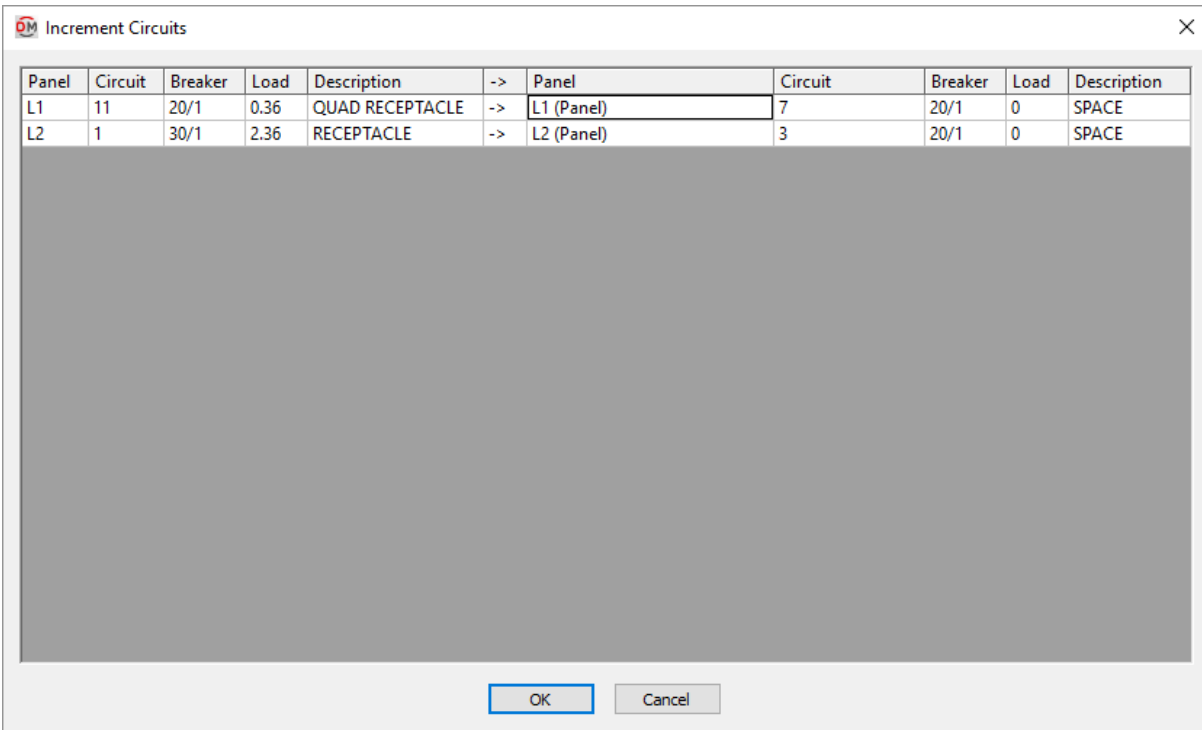
The **Increment Circuits** dialog box is broken into two sections, separated by arrows (->).

The left section lists information about the circuits connected to the devices being copied. Each circuit being copied is listed. The associated *Panel*, *Breaker*, *Load*, and *Description* for the circuit are also displayed.

The right section lists the circuit to which the new devices will be connected. You can change the *Panel*, *Circuit*, and *Description* columns.

Insert Note

To insert a note on a device, go to

Ribbon: `DM Electrical->Devices->` INS NOTE `Insert Note`

Pulldown Menu: `DM Electrical->Modify Devices->Insert Note`

You will be prompted to specify the device to which the note will be added.

Select device to add note to:

Insert Device Note Dialog Box
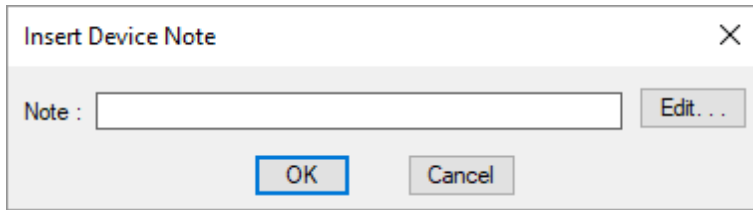
Enter the note into the *Note* field and press the **OK** button.

If the note is longer than one line, press the **Edit** button and enter in the entire note.

See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the note.

Edit Note

To edit a note on a device, go to

Ribbon: `DM Electrical->Devices->` **EDIT NOTE** `Edit Note`

Pulldown Menu: `DM Electrical->Modify Devices->Edit Note`

You will be prompted to specify the note to be edited.

Select device note to edit:

The dialog box that is displayed depends upon the type of note selected. When selecting a general note inserted using the **Insert Note** command, the **Edit Device Note** dialog box will be displayed.

Edit Device Note Dialog Box
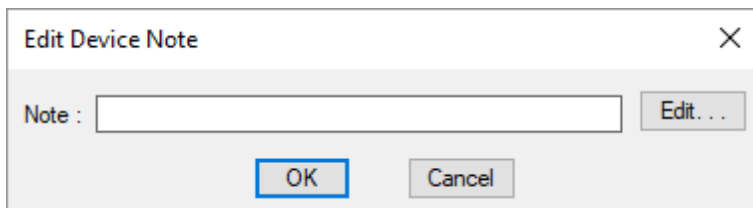
Enter the revised note into the *Note* field and press the **OK** button.

If the revised note is longer than one line, press the **Edit** button and enter in the entire note.

The revised note will be displayed on the drawing in the same location.

Editing Other Types of Notes

Selecting other types of notes will display different dialog boxes. If you can change the note, a dialog box with the note value with be displayed. A larger dialog box for the device may open. You will then need to find and change the field for the note you want to edit.

If you cannot directly change the note, nothing will happen. To change these values, you need to modify the source of the information directly. For example, to change a light fixture callout, query the light fixture and change the type there. To change a circuit callout, connect the device to a different circuit.

Insert Device Elevation Label

The `Insert Device Elevation Label` command is used to insert a label on a distribution equipment or branch circuit device that displays the elevation of the device.

To insert an elevation label on a distribution equipment or branch circuit device, go to

Ribbon: `DM Electrical->Devices->` INS ELEV `Insert Device Elevation Label`

Pulldown Menu: `DM Electrical->Modify Devices->Insert Device Elevation Label`

You will be prompted to specify the device to which the label will be added.

Select device to add elevation label to:

You will then be prompted to specify the location of the device elevation label on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the label.

Related Options

Block: Sets the block used for the elevation label.

Label: Sets how the elevation is displayed on the label.

Move Note

The `Move Note` command can be used to move any note associated with a device on the drawing.

To move a note, go to

**Ribbon:** `DM Electrical->Devices->` MOVE NOTE `Move Note`

Pulldown Menu: `DM Electrical->Modify Devices->Move Note`

You will be prompted to specify the note to be moved.

Select device note to move:

See the **Inserting Notes and Leaders on the Drawing** section for more information about moving the note.

Rotate Note

The `Rotate Note` command can be used to rotate notes and labels on the drawing, including:
- Distribution equipment labels (in plan view only)
- Circuit tags

- Receptacle labels
- Light fixture callouts (when part of the light fixture block)
- Equipment disconnect labels
- Low voltage device labels
- Switch labels
- Notes inserted using the **Insert Note** command
- Elevation labels

To rotate a note, go to

Ribbon: `DM Electrical->Devices->` ↺ `Rotate Note`

Pulldown Menu: `DM Electrical->Modify Devices->Rotate Note`

You will be prompted to specify the note to be rotated.

Select device note to rotate:

You will then be prompted to specify a rotation angle for the note.

Specify rotation angle / <Reset to 0>:

Specify a rotation angle, or press **ENTER** to reset the note to the default rotation angle of 0.

Erase Note

To erase a device note, go to

Ribbon: `DM Electrical->Devices->` ✕ `Erase Note`

Pulldown Menu: `DM Electrical->Modify Devices->Erase Note`

You will be prompted to specify the note to be erased.

Select device note to erase:

The note and associated leader will be erased.

Some notes cannot be erased using this command. These notes will need to removed by changing the definition of the device the note is associated with or changing the option that displays the note.

Erase Part of Device

Devices are created by grouping multiple standard CAD entities. There are times when entities that are part of the device are not properly removed. Using the standard CAD **ERASE** command will erase the entire device. Use the `Erase Part of Device` command instead to remove the entity that is no longer needed.

To erase part of a device, go to

Ribbon: `DM Electrical->Devices->` ✕ `Erase Part of Device`

Pulldown Menu: `DM Electrical->Modify Devices->Erase Part of Device`

You will be prompted to specify the part of a device to be erased.

Select entity in group to erase:

The entity will be erased.

Match Note Values and Locations

The `Match Note Values and Locations` command is used to copy the locations and values of notes from one device to another.

Arrange notes on one device using standard CAD commands and the **Move Note** and **Rotate Note** commands, then use this command to easily copy the settings to other devices.

This command will change the values of the notes on the other devices to match the first device you select. To change the note locations without changing the note values, use the **Match Note Locations** command.

To copy note values and locations to other devices, go to

Ribbon: `DM Electrical->Devices->` 📇 `Match Note Values and Locations`

Pulldown Menu: `DM Electrical->Modify Devices->Match Note Values and Locations`

You will be prompted to specify the device with the note you want to copy.

Select device to use as source:

You will then be prompted to specify the devices to which the note will be added in the same relative location.

Select objects:

The notes will be arranged to match the first device you selected.

Match Note Locations

The `Match Note Locations` command is used to copy the location of notes from one device to another.

Arrange notes on one device using standard CAD commands and the **Move Note** and **Rotate Note** commands, then use this command to easily copy the locations to other devices.

This command will change the location of the notes on the other devices to match the first device you select, but will not make any changes to the values. To change the note values as well, use the **Match Note Values and Locations** command.

To copy note locations to other devices, go to

Ribbon: `DM Electrical->Devices->` 📇 `Match Note Locations`

Pulldown Menu: `DM Electrical->Modify Devices->Match Note Locations`

You will be prompted to specify the device with the note you want to copy.

Select device to use as source:

You will then be prompted to specify the devices to which the note will be added in the same relative location.

Select objects:

The notes will be arranged to match the first device you selected.

Light Fixtures

This section describes how to use light fixtures.

Common Light Fixtures Information

The values that can be specified for a light fixture are listed below.

Callout: The name of the light fixture.

Include in Schedule: Whether the light fixture type is displayed in the light fixture schedule that is inserted on the drawing. Most light fixture types will have this box checked. It is commonly unchecked for existing light fixture types and light fixture types that are being used to model other types of devices in your project.

Loads 1, 2, and 3: Light fixtures can have up to three different loads assigned to them. Each load can be connected to a separate circuit. If the light fixture will always be connected to a single circuit, only one load should be used. If the light fixture could be connected to multiple circuits, multiple loads should be used. A common use for multiple loads is emergency lighting, where some lamps are connected to a normal circuit and other lamps are connected to an emergency circuit.

Each load can have the following values set:

> **Load Description:** The description shown in the panel schedule for this light. "LIGHTING" is the default, but it can be changed.

> **Number of Lamps:** The number of lamps (bulbs) in the light fixture.

> **Lamp Description:** Description of the lamp if needed.

> **Input VA:** Load of the fixture including the ballast, if applicable.

> **Watts/Lamp:** The wattage of each lamp.

> **Lumens/Lamp:** The number of initial lumens per lamp.

> **Egress Mode Lumens/Lamp:** The number of initial lumens per egress mode lamp.

> **Lamp Depreciation:** The depreciation value of each lamp (Lumens / lamp x lamp depreciation = actual

lumens). Lamp depreciation may include fixture dirt or lens depreciation, where applicable.

Ballast: The type of ballast on the light fixture. Default ballasts are available in the list. To specify a ballast that is not included in the list, select **Custom** and enter the value in the field below.

Volts: The voltage of the light fixture.

Mounting: How the light fixture is mounted. Default mounting types are available in the list. To specify a mounting that is not included in the list, select **Custom** and enter the value in the field below.

Default Elevation: The default elevation when this type of light fixture is inserted on the drawing. Elevation can be modified for individual light fixtures during and after insertion.

Arm Length: The length of the arm of the fixture. In photometric calculations, the light source is moved this distance from the insertion point of the light fixture block. See the **Distance Input Format** section for more information about specifying this length.

Number of Heads: The number of heads on the fixture.
- **One:** Light is distributed once in the direction of the insertion angle of the light fixture.
- **Two, 90 degree offset:** Light is distributed in the direction of the insertion angle of the light fixture and at 90 degrees counterclockwise.
- **Two, 180 degree offset:** Light is distributed in the direction of the insertion angle of the light fixture and at 180 degrees.
- **Three:** Light is distributed in the direction of the insertion angle of the light fixture, at 90 degrees clockwise, and at 90 degrees counterclockwise.
- **Four:** Light is distributed in the direction of the insertion angle of the light fixture, at 90 degrees clockwise, at 180 degrees, and at 90 degrees counterclockwise.

IES Rotation: The counterclockwise rotation of the IES file relative to the fixture in degrees. Use this value to rotate the IES file if it does not point in the same direction as your light fixture block.

General Scale Factor: Multiplies the light output by the number specified. For use with IES files for LED light fixtures.

Egress Scale Factor: Multiplies the egress mode light output by the number specified. For use with IES files for LED light fixtures.

Energy Code Settings: The settings in this area are for energy code calculations. In particular, they are used when exporting light fixture information to COMCheck.

Exempt: Whether this type of light fixture is exempt from being counted in energy calculations. Light fixtures that have this box checked will not be exported to COMCheck. This will vary by state, but emergency or exit lights are often exempt. This setting can be overridden on an individual light fixture basis on the drawing if necessary. See the **Exempt Light Fixtures** section for more information.

Watts: The load to export to COMCheck. If this value is not specified, the *Input VA* is exported instead.

Fixture Type: The type of fixture to export to COMCheck.

Description: The description to export to COMCheck.

Schedule Description: A description of the type of light fixture. This description will appear on the light fixture

schedule on the drawing.

Model 1, **Model 2**, **Model 3:** A description of the model for the light fixture. In the light fixture schedule, the three models can be shown in a single column or in three separate columns. See the **Edit Light Fixture Schedule List** section for more information.

Note 1, Note 2, Note 3: These notes will appear on the light fixture schedule on the drawing. See the **Schedule Notes** section for more information about setting the default values for these notes.

Insert Light Fixture

To insert a light fixture on a drawing, go to

Ribbon: `DM Electrical->Devices->` `Insert Light Fixture`

Pulldown Menu: `DM Electrical->Light Fixtures->Insert Light Fixture`

Insert Light Fixture Dialog Box

Group: The active light fixture group. The items listed in the *Callout* list are based upon the active light fixture group. The list of groups available is defined in the **Light Fixture Project Schedule** command.

Callout: The type of light fixture to be inserted. The list of light fixtures available is defined in the **Light Fixture Project Schedule** command.

The selected callout controls the values displayed in the top-right of the dialog box. See the **Common Light Fixtures Information** section for more information about these values. To change these values, you must change the definition of the light fixture type in the project schedule.

Elevation: The elevation of the light fixture. See the **Elevation** section for more information.

The elevation is used as the mounting height of the light fixture when calculating point-by-point photometrics. See the **Point-by-Point Photometrics** section for more information.

Layer System: See the **Layer System** section for more information.

171

Exempt: Whether the light fixture is exempt from energy calculations.

- **Default:** The exempt setting is based upon the value in the light fixture project schedule.
- **Yes:** The light fixture is exempt from energy calculations. This value overrides the setting in the light fixture project schedule.
- **No:** The light fixture is not exempt from energy calculations. This value overrides the setting in the light fixture project schedule.

See the **Exempt Light Fixtures** section for more information.

General Light: Whether the light fixture is included in general point-by-point photometric calculations. See the **Point-by-Point Photometrics** section for more information.

Egress Light: Whether the light fixture is included in egress point-by-point photometric calculations. See the **Point-by-Point Photometrics** section for more information.

Tilt: The angle in degrees at which the light fixture is tilted. The rotation is in the direction of the insertion angle of the light fixture.

If the light fixture has multiple heads, the *Tilt* will be applied to each head in its oriented direction.

Roll: The angle in degrees at which the light fixture is rolled. The rotation is perpendicular to the direction of the insertion angle of the light fixture.

If the light fixture has multiple heads, the *Roll* will be applied to each head perpendicular to its oriented direction.

Inserting the Light Fixture on the Drawing

To insert a light fixture on the drawing, press the **OK** button. The settings specified in the dialog box will be used for the inserted light fixture. You will be prompted to identify where on the drawing the light fixture is to be inserted.

Specify insertion point:

You will then be prompted for the rotation angle. If the light fixture block is not able to be rotated, this prompt will be skipped.

Specify rotation angle <0>:

You will then be prompted to insert another light fixture of the same type. Continue to insert light fixtures, or press **ENTER** to finish the command.

Related Options

Insert callout as part of each light fixture: Sets whether the *Callout* is automatically inserted along with the fixture.

Insert callout after each light fixture: Sets whether you are prompted to insert a light fixture callout after inserting a light fixture on the drawing.

This option can be further customized with the **LF callout block** and **LF callout short label width** options.
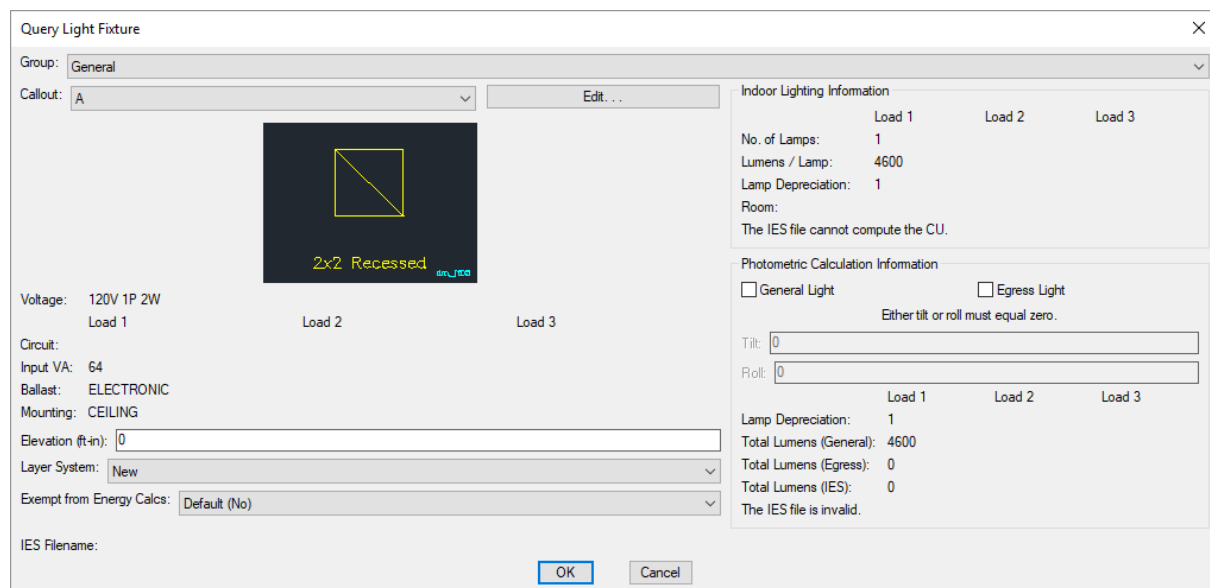
Querying Light Fixtures

To modify a light fixture that is inserted on the drawing, go to

Ribbon: `DM Electrical->Modify Devices->` `Query Device`

Pulldown Menu: `DM Electrical->Modify Devices->Query Device`

Select a light fixture on the drawing. The information in the database regarding the selected light fixture will be displayed in the **Query Light Fixture** dialog box.

Query Light Fixture Dialog Box

Changes can be made to the light fixture in the same way as when it was inserted. See the **Insert Light Fixture** section for more information.

Circuit: This field displays the circuit that the load on the light fixture is connected to. Use the **Circuiting** command to connect the light fixture to a new circuit.

Indoor Lighting Information

The information displayed in this area of the dialog box is used when performing indoor lighting calculations with

the light fixture.

No. of Lamps: Set using the *Number of Lamps* field in the **Light Fixture Project Schedule** command.

Lumens / Lamp: Set using the *Lumens / Lamp* field in the **Light Fixture Project Schedule** command.

Lamp Depreciation: Set using the *Lamp Depreciation* field in the **Light Fixture Project Schedule** command.

Room: The room the light fixture is inserted in on the drawing.

Photometric Calculation Information

The information displayed in this area of the dialog box is related to the point-by-point lighting calculations for the queried light fixture. See the **Point-by-Point Photometrics** section for more information about the calculation.

Lamp Depreciation: Set using the *Lamp Depreciation* field in the **Light Fixture Project Schedule** command.

Total Lumens (General): The product of the *Lumens / Lamp* and *Number of Lamps* fields set in the **Light Fixture Project Schedule** command.

Total Lumens (Egress): The product of the *Egress Mode Lumens / Lamp* and *Number of Lamps* fields set in the **Light Fixture Project Schedule** command.

Total Lumens (IES): The total lumens set in the IES file for the fixture.

If a number is displayed, the light output from the IES file will be scaled based upon the ratio of the *Total Lumens (General)* or *Total Lumens (Egress)* value to the *Total Lumens (IES)* value.

If **Absolute** is displayed, the IES file uses absolute photometry. The total lumens for the IES file are not provided. The light output for the fixture will not be scaled based upon the *Total Lumens (General)* or *Total Lumens (Egress)* values.

Edit Multiple Light Fixtures

To edit multiple light fixtures, go to

Ribbon: DM Electrical->Devices-> [LTG EM] Edit Multiple Light Fixtures

Pulldown Menu: DM Electrical->Light Fixtures->Edit Multiple Light Fixtures

You will be prompted to select the light fixtures to be edited.

Select lights to edit:

The **Edit Multiple Light Fixtures** dialog box will appear.

Edit Multiple Light Fixtures Dialog Box

Changes can be made to the light fixture in the same way as when it was inserted. See the **Insert Light Fixture** section for more information.

Insert Light Fixture Callout

The `Insert Light Fixture Callout` command is used to insert a single light fixture callout label on the drawing.

The callout label is linked to the light fixture it is inserted on. Changing the callout of the light fixture in the database will change the callout label on the drawing. Erasing the light fixture from the drawing will erase the callout label.

The callout label is not moved with the light fixture. If the light fixture is moved to a new location, the callout label must be moved separately.

Light fixture callout labels can also be inserted with each light fixture that is inserted on the drawing. See the *Insert Callout as Part of Each Light Fixture* option in the **Light Fixtures** options section for more information.

To insert a light fixture callout, go to

Ribbon: `DM Electrical->Devices->` `Insert Light Fixture Callout`

Pulldown Menu: `DM Electrical->Light Fixtures->Insert Light Fixture Callout`

You will be prompted to select the light fixture to be labeled.

Select light fixture:

You will then be prompted to specify where the callout will be inserted on the drawing.

Specify insertion point:

The light fixture callout will be inserted at the selected location.

Related Options

LF callout block: Sets the blocks used for the light fixture callout.

This option can be further customized with the **LF callout short label width** option.

Calculate Lighting Load

The `Calculate Lighting Load` command is used to display the total load in kVA on a selected group of light fixtures.

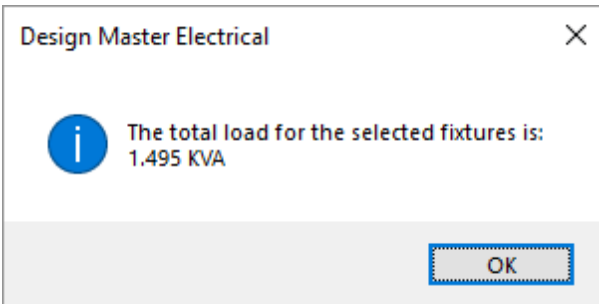To determine the lighting load for select light fixtures, go to

Ribbon: `DM Electrical->Devices->` `Calculate Lighting Load`

Pulldown Menu: `DM Electrical->Light Fixtures->Calculate Lighting Load`

You will be prompted to select light fixtures.

Select objects:

A **Design Master Electrical** dialog box with the lighting load displayed will appear, as shown below.

Press the **OK** button to close the dialog box.

Trace Architect's Light Fixtures

The `Trace Architect's Light Fixtures` command is used to trace a light fixture layout from another drawing that does not have Design Master entities.

Before using this command, create light fixtures using the **Light Fixture Project Schedule** command.

This command should be run on the drawing from which you are copying the light fixture layout. Do not run this command on the drawing you are using for your electrical design. Both drawings must be in the same folder.

To trace a light fixture layout, go to

Ribbon: `DM Electrical->Devices->` LTG TRACE `Trace Architect's Light Fixtures`

Pulldown Menu: `DM Electrical->Light Fixtures->Trace Architect's Light Fixtures`

Background Mapper Dialog Box
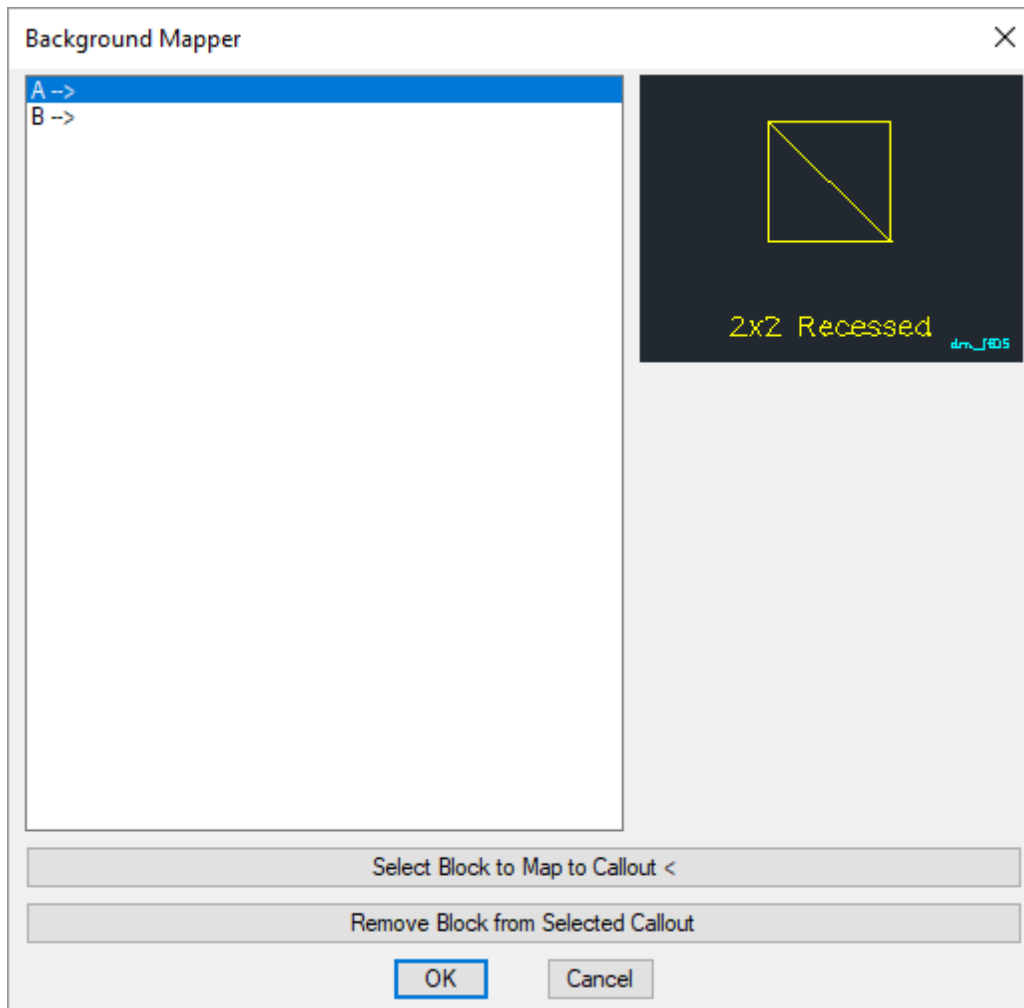
The list in the top-left of the dialog displays all of the light fixtures currently defined in the project schedule. Each callout can be mapped to a specific block on the drawing. Callouts that are currently mapped will list the block to which they are currently mapped.

Select Block to Map to Callout: Press this button to select the block on the drawing that will be mapped to the selected light fixture callout. If the callout is already mapped to a block, it will be overwritten with the newly selected block.

Remove Block from Selected Callout: Press this button to remove the block to which the selected callout is mapped. The callout will no longer be mapped to a block.

Tracing the Light Fixtures

After all of the callouts have been properly mapped to blocks, press the **OK** button.

You will be prompted to select the location of the alignment point on the drawing. Make sure you select the location on this drawing that corresponds to the alignment point on the destination drawing.

Select alignment point location:

You will then be prompted to specify the north direction on the current drawing.

Select north direction:

You will then be prompted to select the blocks to actually map. This allows you to map part of a drawing if the entire light fixture layout is not needed.

Select blocks to map:

You will then be prompted to select the drawing to which the light fixtures will be mapped. Make sure no one, including yourself, has this drawing open or the command will fail.

Select Drawing to Map Devices to Dialog Box
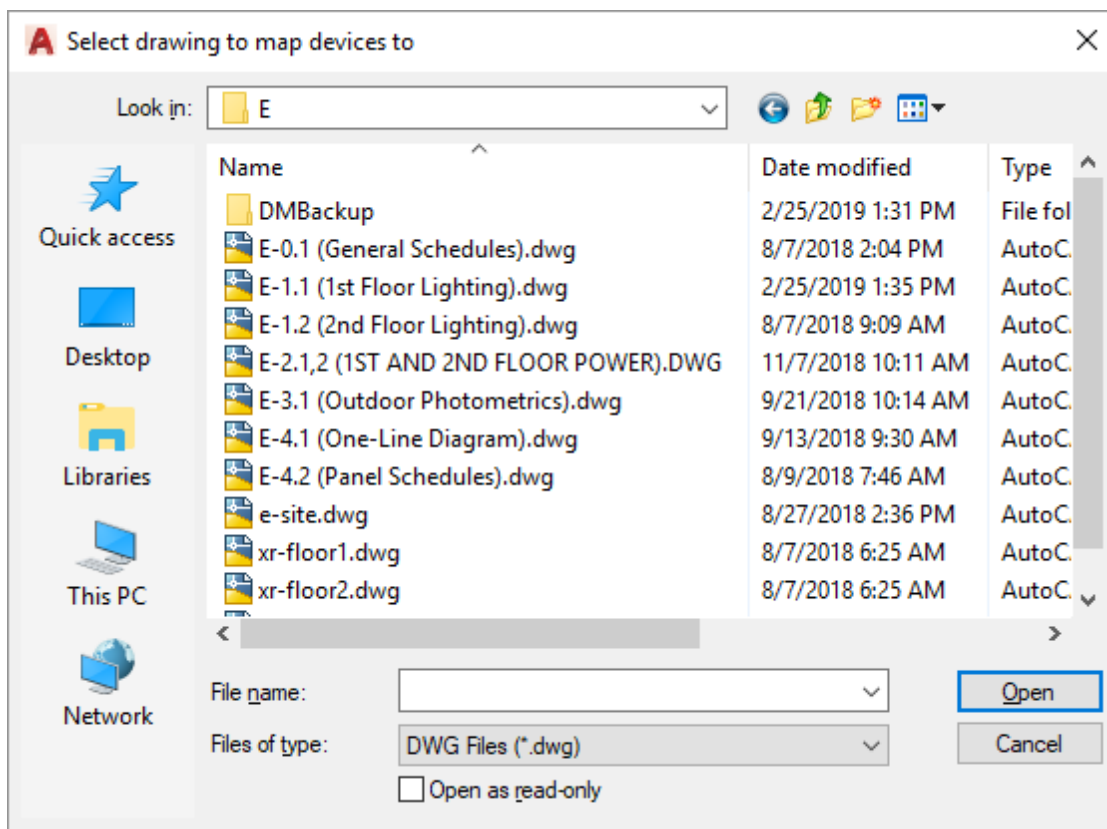
The selected drawing will be opened and the light fixtures will be inserted on it.

When inserting the light fixtures, the software checks whether a light fixture already exists in each location. If there is one, a new light fixture will not be inserted at that location. This feature prevents duplication of light fixtures on your drawings.

It is possible to take an updated plan from your architect and run this command on it again. Light fixtures that have not moved will not be duplicated. New light fixtures or light fixtures that have been moved will be reinserted on the drawing. You will need to manually erase any light fixtures that have been removed.

Light Fixture Blocks Not Inserted Correctly

When using this command, the light fixture blocks that are inserted may not match the blocks that the architect inserted. They may be offset slightly or rotated at a different angle.

This occurs when your block and the architect's block have a different insertion point and/or rotation angle. If you insert both blocks at the exact same location with a rotation angle of 0, they will not match.

The solution is to create a new block or redefine your existing block so that it matches the architect's block. Use standard CAD commands to create or modify the block so that it looks correct, then use the **Create Plan View Block from This Drawing** to configure the block for use as a light fixture.
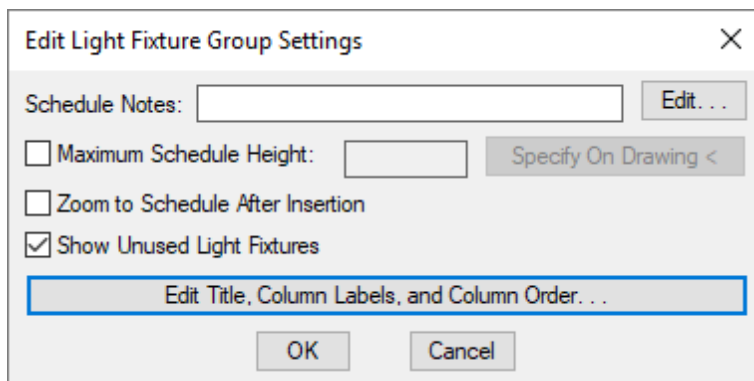
Insert Light Fixture Project Schedule

A light fixture schedule that lists all of the light fixtures used in the current project can be inserted on the drawing.

To insert this schedule onto the drawing, go to

Ribbon: `DM Electrical->Devices->` 📊 `Insert Light Fixture Project Schedule`

Pulldown Menu: `DM Electrical->Light Fixtures->Insert Light Fixture Project Schedule`

Edit Light Fixture Group Settings Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Show Unused Light Fixtures: Whether light fixtures in the schedule that have not been inserted on a drawing will be displayed in the schedule.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the light fixture schedule on the drawing. See the **Edit Light Fixture Project Schedule List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.
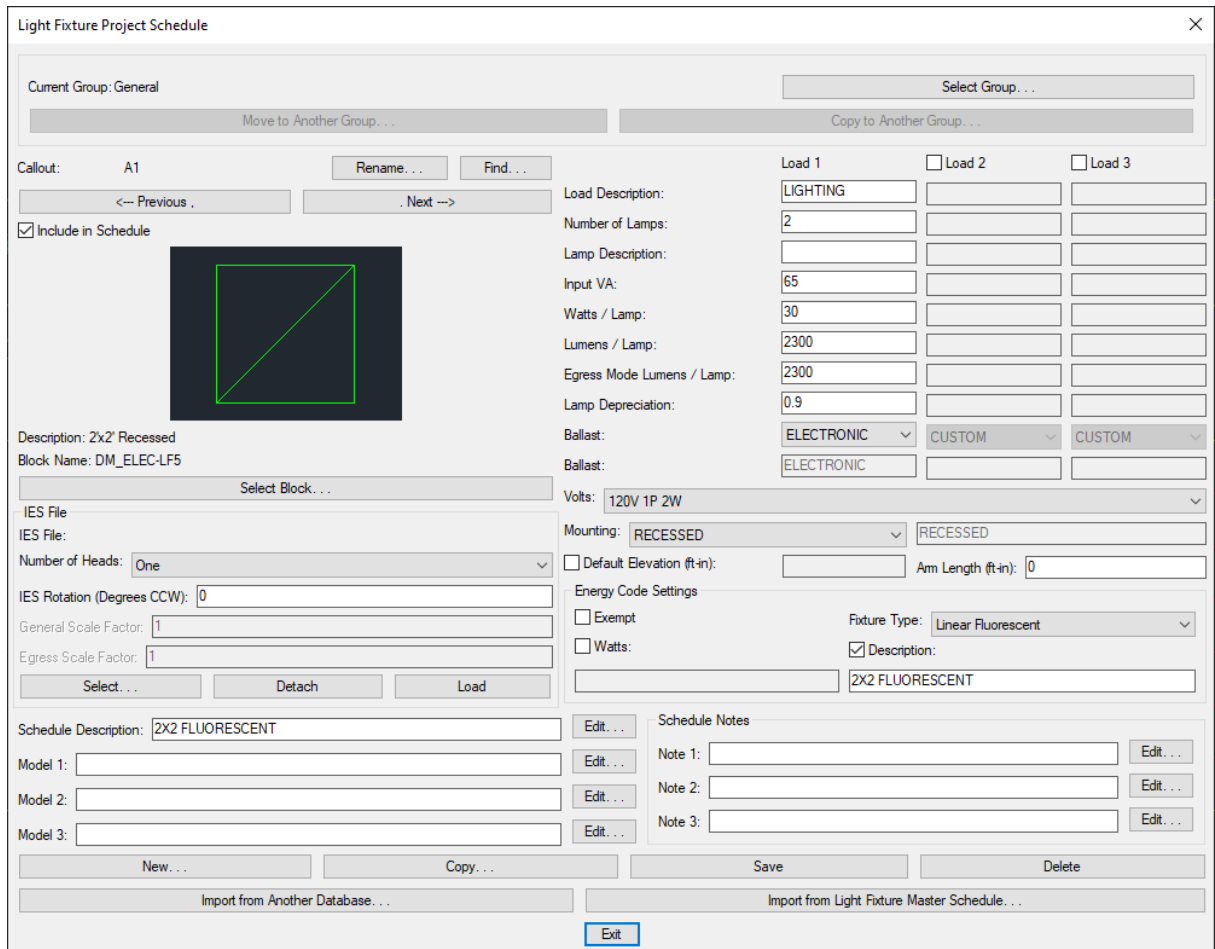
Light Fixture Project Schedule

The light fixture project schedule contains the light fixture types used in the current project.

To create and modify light fixture types, go to

Ribbon: `DM Electrical->Devices->` **LTG** **P** `Light Fixture Project Schedule`

Pulldown Menu: `DM Electrical->Light Fixtures->Light Fixture Project Schedule`

Light Fixture Project Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.
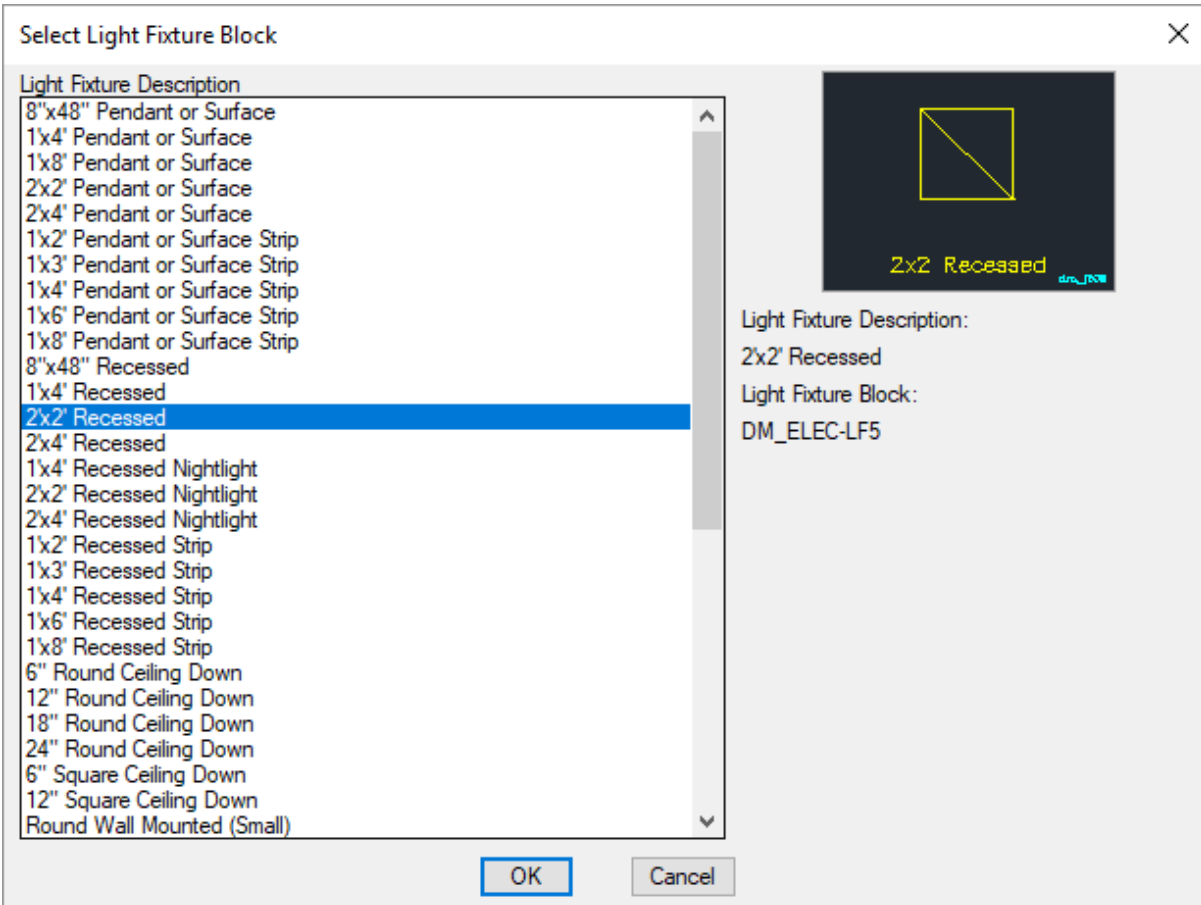
See the **Common Light Fixture Information** section for more information about the values that can be specified for light fixtures using this dialog box.

See the **Import from Another Database** and **Import from Master Schedule** sections for more information about importing light fixtures to the current project from other locations.

Current Group: The active light fixture group is listed here. All of the light fixture types that are created and modified will be associated with this group.
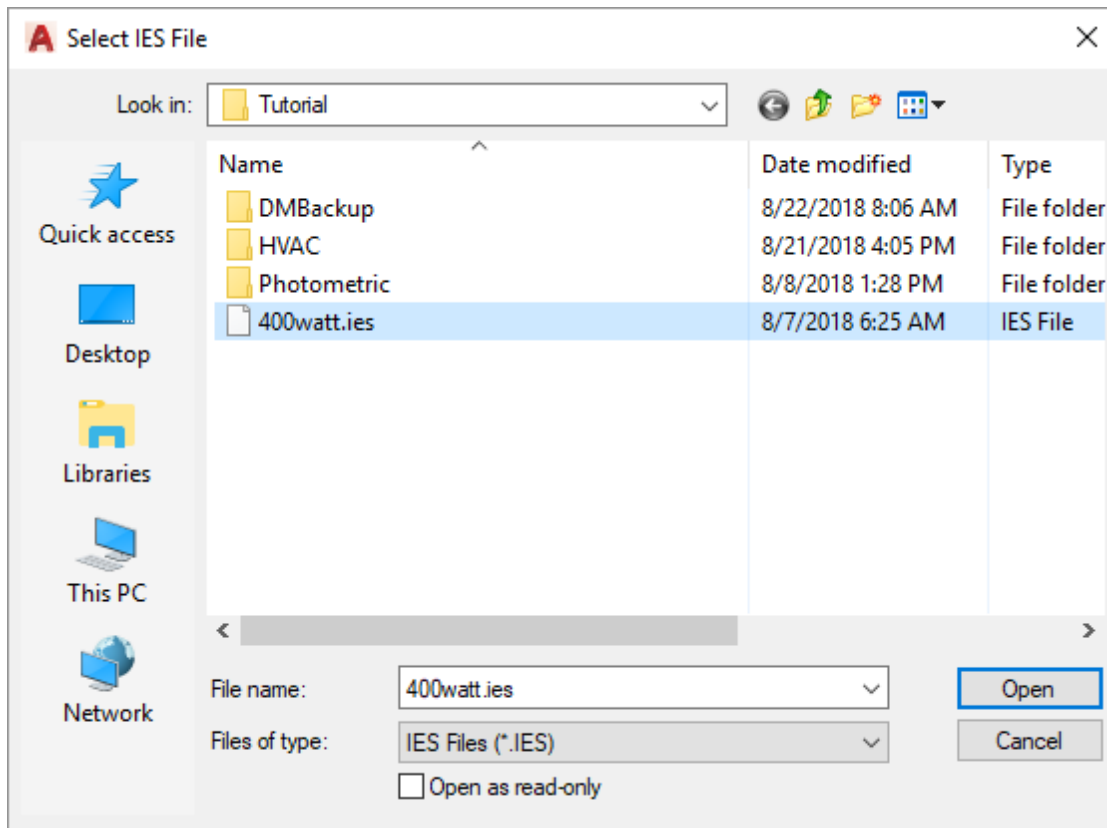
Select Group: Press this button to open the **Light Fixture Project Schedule Groups** dialog box. The group selected will become the current group. See the **Schedule Groups** section for more information.

Select Block: Press this button to open the **Select Light Fixture Block** dialog box. This dialog box allows you to choose the block used to represent the light fixture on the drawing.
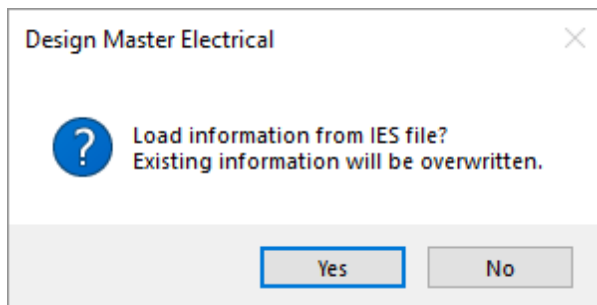

You can add more blocks to this list using the **Light Fixture Blocks->Edit Project List** command.

Select: Press this button to select the IES file for the light fixture. The **Select IES File** dialog box will appear.

Select the IES file for this light fixture and press the **Open** button. The IES file will be attached to this light fixture and you will be asked if the information is to be added to the schedule.

Press the **Yes** button to add the information to the schedule. The information contained in the IES file varies depending upon what the manufacturer provides. You will need to review the information that is loaded to confirm it is loaded into the correct field and that it contains useful text.
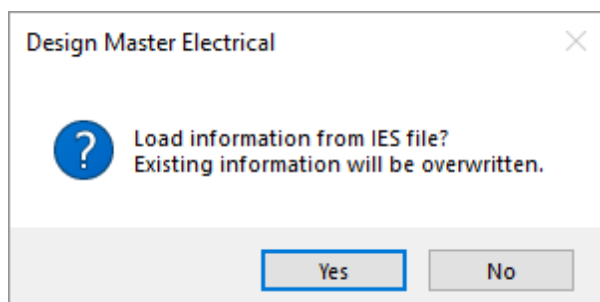
Press the **No** button to only add the information from the IES file used for lighting calculations. The other information in the light fixture schedule will not be filled in based upon the information in the IES file.

Once an IES file is attached to a light fixture, the file path is displayed in the *IES File* field of the dialog box.

Detach: Press this button to detach the IES file from the light fixture. The light fixture will have no IES file associated with it after pressing this button.

Load: Press this button to load the IES file information into the schedule. The information contained in the IES file varies depending upon what the manufacturer provides. You will need to review the information that is loaded to confirm it is loaded into the correct field and that it contains useful text.

You will be asked if the information is to be added to the schedule.

Press the **Yes** button.

All of the information from the IES file will be loaded into the schedule.

Related Options

Copy IES files to project folder: Sets whether loaded IES files are copied to the project folder.

Light Fixture Master Schedule

The light fixture master schedule contains the light fixture types that could be used by a company on a project. See the **Master and Standards Databases** section for more information about using master databases.

To create and modify callouts in the master schedule, go to

Ribbon: `DM Electrical->Devices->` `Light Fixture Master Schedule`

Pulldown Menu: `DM Electrical->Light Fixtures->Light Fixture Master Schedule`

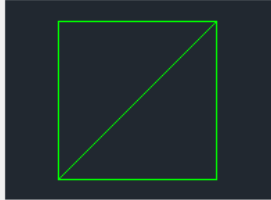## Light Fixture Master Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Common Light Fixture Information** section for more information about the values that can be specified for light fixtures using this dialog box.

See the **Import from Project Schedule** section for more information about importing light fixtures from a project to the master database.

Current Group: The active light fixture group is listed here. All of the light fixture types that are created and modified will be associated with this group.

Select Group: Press this button to open the **Light Fixture Master Schedule Groups** dialog box. The group selected will become the current group. See the **Schedule Groups** section for more information.

Select Block: Press this button to open the **Select Light Fixture Block** dialog box. This dialog box allows you to choose the block used to represent the light fixture on the drawing.

You can add more blocks to this list using the **Light Fixture Blocks->Edit Master List** command.

Select: Press this button to select the IES file for the light fixture. The **Select IES File** dialog box will appear.

Select the IES file for this light fixture and press the **Open** button. The IES file will be attached to this light fixture and you will be asked if the information is to be added to the schedule.

Press the **Yes** button to add the information to the schedule. The information contained in the IES file varies depending upon what the manufacturer provides. You will need to review the information that is loaded to confirm it is loaded into the correct field and that it contains useful text.
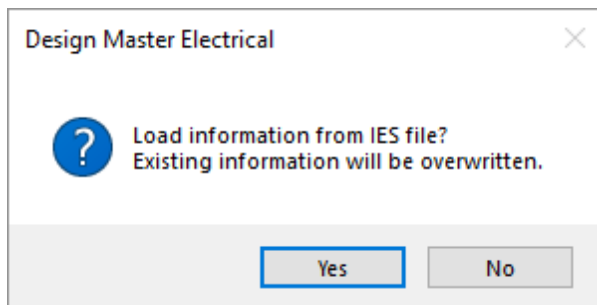
Press the **No** button to only add the information from the IES file used for lighting calculations. The other information in the light fixture schedule will not be filled in based upon the information in the IES file.

Once an IES file is attached to a light fixture, the file path is displayed in the *IES File* field of the dialog box.

Detach: Press this button to detach the IES file from the light fixture. The light fixture will have no IES file associated with it after pressing this button.

Load: Press this button to load the IES file information into the schedule. The information contained in the IES file varies depending upon what the manufacturer provides. You will need to review the information that is loaded to confirm it is loaded into the correct field and that it contains useful text.
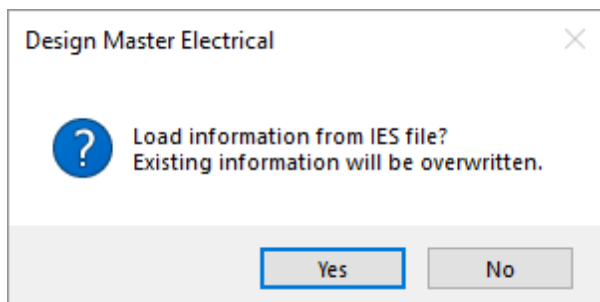
You will be asked if the information is to be added to the schedule.

Press the **Yes** button.

All of the information from the IES file will be loaded into the schedule.

Diffuser Coordination

If your company is using both Design Master Electrical and Design Master HVAC, it is possible to insert the diffusers from Design Master HVAC on your electrical drawings for coordination with your lighting plan.

The **Coordinate Light Fixtures with Diffusers** command inserts the diffusers from the HVAC project on your drawing.

The **Erase Coordination Diffusers** command removes the diffusers from your drawing.

# Coordinate Light Fixtures with Diffusers

To coordinate the light fixtures on an electrical drawing with the diffusers in a Design Master HVAC project, go to

Ribbon: DM Electrical->Devices-> ⊠ Coordinate Light Fixtures with Diffusers

Pulldown Menu: DM Electrical->Light Fixtures->Diffuser Coordination->Coordinate Light Fixtures with Diffusers

Select HVAC Database to Transfer Diffuser From Dialog Box
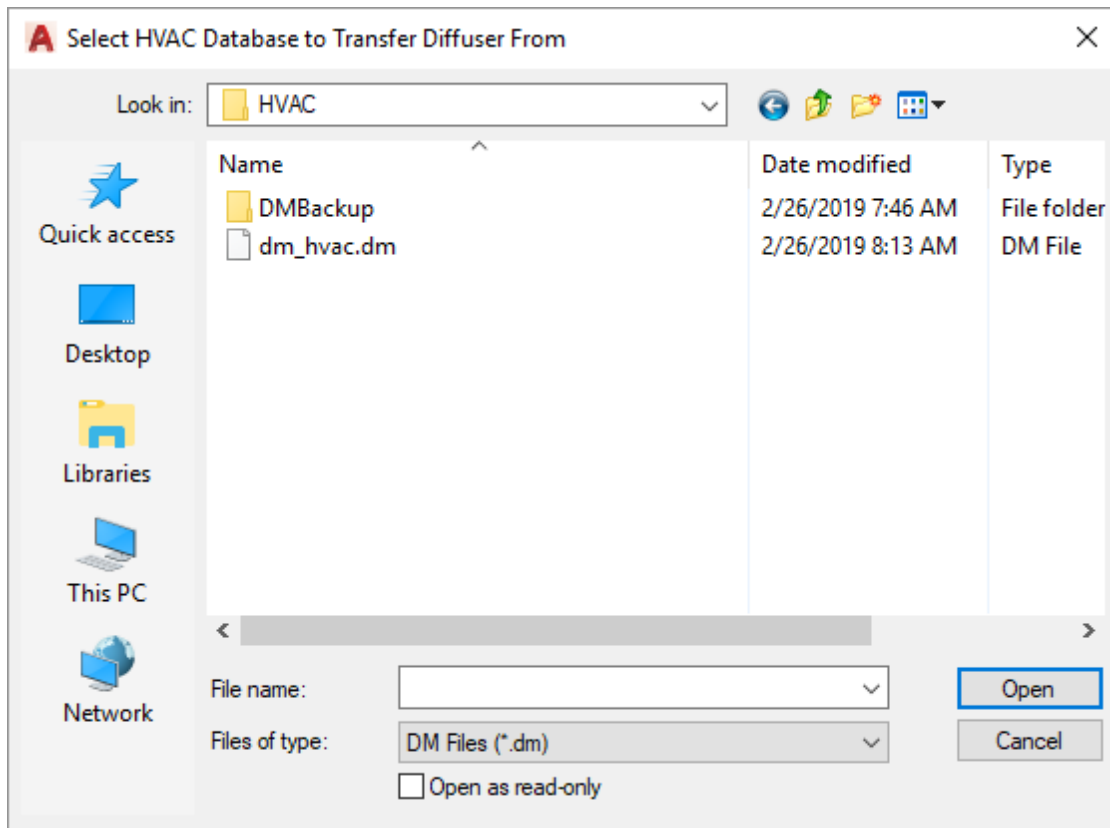
Select the HVAC database from the project you are working on and press the **Open** button.

The database file name will start with "dm_hvac". The extension will be ".dm". There may be a project name included at the end of the file name, depending upon how the project was originally set up.

The database file will be in the same folder as the drawings with the diffusers on them. The specific location of the database will vary depending upon how your company arranges its project folders.

After you press the **Open** button, the **Diffuser Coordination** dialog box will appear.
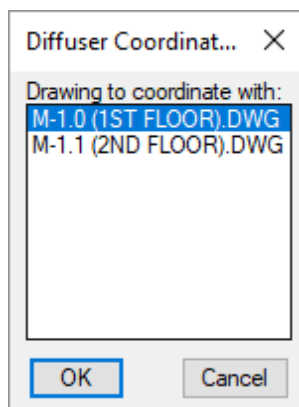
Select the drawing with which you would like to coordinate and press the **OK** button. The diffusers from the selected database will be inserted on the current drawing.

Erase Coordination Diffusers

To erase the diffusers that were placed on the electrical drawing using the **Coordinate Light Fixtures with Diffusers** command, go to

Ribbon: `DM Electrical->Devices->` ✕ `Erase Coordination Diffusers`

Pulldown Menu: `DM Electrical->Light Fixtures->Diffuser Coordination->Erase Coordination Diffusers`

Diffusers imported from any HVAC drawing will be erased from the drawing.
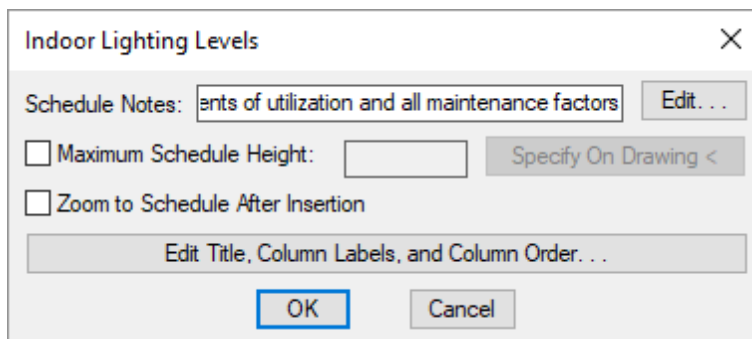
Insert Indoor Lighting Level Report

The `Insert Indoor Lighting Level Report` command is used to display the lighting levels in each room in the project.

To insert the indoor lighting level report on the drawing, go to

Ribbon: `DM Electrical->Devices->` 🔲 `Insert Indoor Lighting Level Report`

Pulldown Menu: `DM Electrical->Light Fixtures->Insert Indoor Lighting Level Report`

Indoor Lighting Levels Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. See the **Edit Indoor Lighting Level Schedule List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.
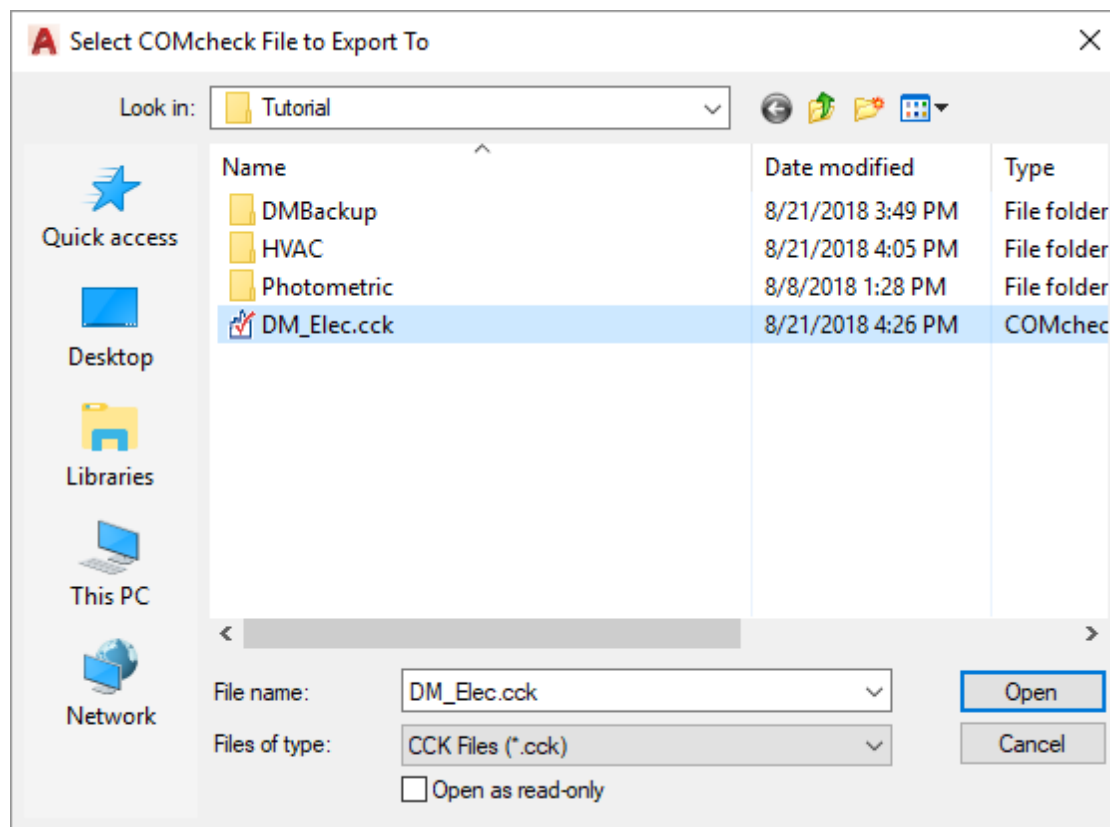
Export to COMcheck

The `Export to COMcheck` command is used to export lighting information for your project to a COMcheck file. See the **COMcheck Lighting Export** article in the knowledge base for more information about which data is exported and how it is used.

To export to COMcheck, go to

Ribbon: `DM Electrical->Devices->` `Export to COMcheck`

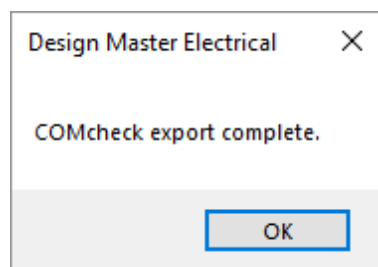**Pulldown Menu:** `DM Electrical->Light Fixtures->Export to COMcheck`

Select COMcheck File to Export To Dialog Box

Select the COMcheck file for the project you are working on and press the **Open** button.

If the project is exported successfully, a **Design Master Electrical** dialog box will appear confirming the export.

Receptacles

This section describes how to use receptacles.

Receptacles can be used to model loads that are not receptacles. Any electrical load that is repeated throughout a building that is not a light fixture is a good candidate to be modeled as a receptacle.

Common Receptacle Information

The values that can be specified for a receptacle are listed below. Some values can be set in the schedule only. Others can have a default set in the schedule that can be changed on individual receptacles.

Schedule-Only Settings

This section describes receptacle values that can only be set in the schedule. These cannot be changed on individual receptacles. If you need to insert a receptacle that has a different setting for one of these values, you must create a new record in the schedule.

Include in Schedule: Whether the receptacle type is displayed in the receptacle schedule that is inserted on the drawing. Most receptacle types will have this box checked. It is commonly unchecked for existing receptacle types and receptacles that are being used to model other types of devices in your project.

Volts: The voltage for the receptacle type.

NEMA Type: The NEMA rating for the receptacle. NEMA ratings are standards that refer to the environment where the receptacle is safe to use. This value is not displayed on the drawing. It is included as a column in the receptacle schedule on the drawing.

Explosion-proof: Whether the receptacle is explosion-proof. If this is checked, an explosion-proof label is inserted on the drawing. The *Explosion-proof* text can be changed in the **Receptacles** options section.

Weatherproof: Whether the receptacle is weatherproof. This typically corresponds to a NEMA 3R rating. If this is checked, a weatherproof label is inserted on the drawing. The *Weatherproof* text can be changed in the **Receptacles** options section.

GFCI: Whether the receptacle is a GFCI outlet. If this is checked, a GFCI label is inserted on the drawing. The *GFCI* text can be changed in the **Receptacles** options section.

Ground: Whether the receptacle requires a ground wire. If this is checked, a ground label is inserted on the drawing. The *Ground* text can be changed in the **Receptacles** options section.

Isolated Ground: Whether the receptacle requires an isolated ground wire. If this is checked, an isolated ground label is inserted on the drawing. The *Isolated Ground* text can be changed in the **Receptacles** options section.

Individual Receptacle Settings

This section describes receptacle values that be changed for individual receptacles. The value in the receptacle schedule is the default value.

Subscript Note: This note is inserted with the receptacle on the drawing. This note can also be displayed in the receptacle schedule if the *Show Subscript on Symbols* box is checked when inserting the schedule. See the **Insert Receptacle Project Schedule** section for more information.

Note with Arrow #1, Note with Arrow #2: These notes will appear on the drawing when the receptacle is inserted. These notes will not appear on the receptacle schedule.

Load 1, Load 2, Load 3, Load 4: Receptacles can have up to four separate loads on them. Each load can be assigned a different load type and connected to a different circuit. See the **Load Information** section for more information.

Schedule Note 1, Schedule Note 2, Schedule Note 3: These notes will appear on the receptacle schedule. See the **Schedule Notes** section for more information.

Insert Receptacle

To insert a receptacle on a drawing, go to

Ribbon: `DM Electrical->Devices->` 🔲 `Insert Receptacle`

Pulldown Menu: `DM Electrical->Receptacles->Insert Receptacle`
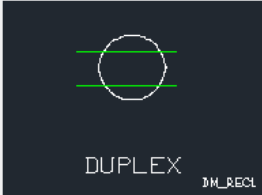
Insert Receptacle Dialog Box

Receptacle Type: The type of receptacle to be inserted. The list of receptacles available is defined in the **Receptacle Project Schedule** command.

The default values for the receptacle are specified based upon the receptacle type you choose.

Some of these values cannot be changed in the individual receptacle. To use a different value, you must choose a different receptacle type.

Other values can be overridden in the individual receptacle. To do this, check the box next to the value you want you change. You can then specify the value that will apply to the receptacles you will insert on the drawing.

Elevation: The elevation of the receptacle. See the **Elevation** section for more information.

Layer System: See the **Layer System** section for more information.

Override Schedule Load: Check these boxes to override the load defined in the schedule. If this box is checked, the load fields below it will be enabled and can be changed.

See the **Common Receptacle Information** section for more information about the other values in this dialog box.

Inserting the Receptacle on the Drawing

To insert a receptacle on the drawing, press the **OK** button. The settings specified in the dialog box will be used for the inserted receptacle. You will be prompted to identify where on the drawing the receptacle is to be inserted.

Specify insertion point:

You will then be prompted for the rotation angle. If the receptacle block is not able to be rotated, this prompt will be skipped.

Specify rotation angle <0>:

If there are any notes included with the receptacle, you will then be prompted to insert the notes on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the notes.

You will then be prompted to insert another receptacle of the same type. Continue to insert receptacles, or press **ENTER** to finish the command.

Related Options

Receptacles options: Sets the label inserted on the drawing with the receptacle if *Explosion-proof*, *Weather-proof*, *GFCI*, or *Isolated Ground* is set to **Yes**.

Insert 3D blocks on drawing: Sets whether a 3D block is inserted on the drawing along with the 2D block.
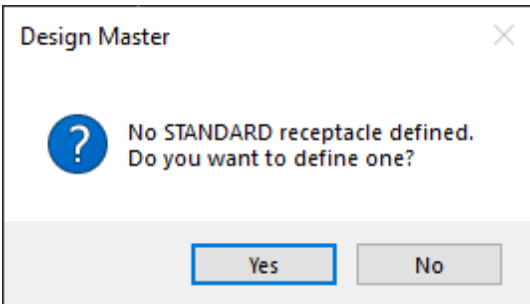
Insert Standard Receptacle

The `Insert Standard Receptacle` command is used to quickly insert a typical receptacle on the drawing without using the **Insert Receptacle** dialog box.

To insert a standard receptacle on a drawing, go to

Ribbon: DM Electrical->Devices-> ᴵᴺˢ STND Insert Standard Receptacle

Pulldown Menu: DM Electrical->Receptacles->Insert Standard Receptacle

When you run the command, it will attempt to find a receptacle definition in the in the **Receptacle Project Schedule** with the **STANDARD** callout. If one does not exist, you will be prompted to create one.

Press the **Yes** button to open the the **Receptacle Project Schedule** and define a standard receptacle.

Once you have defined the receptacle, press the **Save** button to save your changes, then press the **Exit** button to insert the receptacle on the drawing.

Inserting a Standard Receptacle on the Drawing

You will be prompted to identify where on the drawing the receptacle is to be inserted.

Specify insertion point:

You will then be prompted for the rotation angle. If the receptacle block is not able to be rotated, this prompt will be skipped.

Specify rotation angle <0>:

If there are any notes included with the receptacle, you will then be prompted to insert the notes on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the notes.

You will then be prompted to insert another receptacle of the same type. Continue to insert receptacles, or press **ENTER** to finish the command.

Related Options

Receptacles options: Sets the label inserted on the drawing with the receptacle if *Explosion-proof*, *Weather-proof*, *GFCI*, or *Isolated Ground* is set to **Yes**.

Insert 3D blocks on drawing: Sets whether a 3D block is inserted on the drawing along with the 2D block.

Querying Receptacles

To modify a receptacle that is inserted on the drawing, go to

Ribbon: DM Electrical->Devices-> Query Device

Pulldown Menu: DM Electrical->Modify Devices->Query Device

Select a receptacle on the drawing. The information in the database regarding the selected receptacle will be displayed in the **Query Receptacle** dialog box.
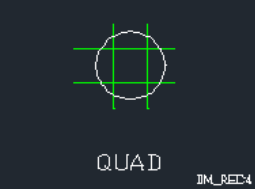
Query Receptacle Dialog Box

Changes can be made to the receptacle in the same way as when it was inserted. See the **Insert Receptacles** section for more information.

Current Circuit: This field displays the circuit that the load on the receptacle is connected to. Use the **Circuiting** command to connect the receptacle to a new circuit if necessary.

Edit Multiple Receptacles

To edit multiple receptacles, go to

Ribbon: DM Electrical->Devices-> [REC EM] Edit Multiple Receptacles

Pulldown Menu: DM Electrical->Receptacles->Edit Multiple Receptacles

You will be prompted to select the receptacles to be edited.

Select receptacles to edit:

The **Edit Multiple Receptacles** dialog box will appear.

Edit Multiple Receptacles Dialog Box

Changes can be made to the receptacles in the same way as when they were inserted. See the **Insert Receptacles** section for more information.

Trace Architect's Receptacles

The `Trace Architect's Receptacles` command is used to trace a receptacle layout from another drawing that does not have Design Master entities.

Before using this command, create receptacles using the **Receptacle Project Schedule** command.

This command should be run on the drawing from which you are copying the receptacle layout. Do not run this command on the drawing you are using for your electrical design.

To trace a receptacle layout, go to

Ribbon: `DM Electrical->Devices->` `Trace Architect's Receptacles`

Pulldown Menu: `DM Electrical->Receptacles->Trace Architect's Receptacles`

Background Mapper Dialog Box
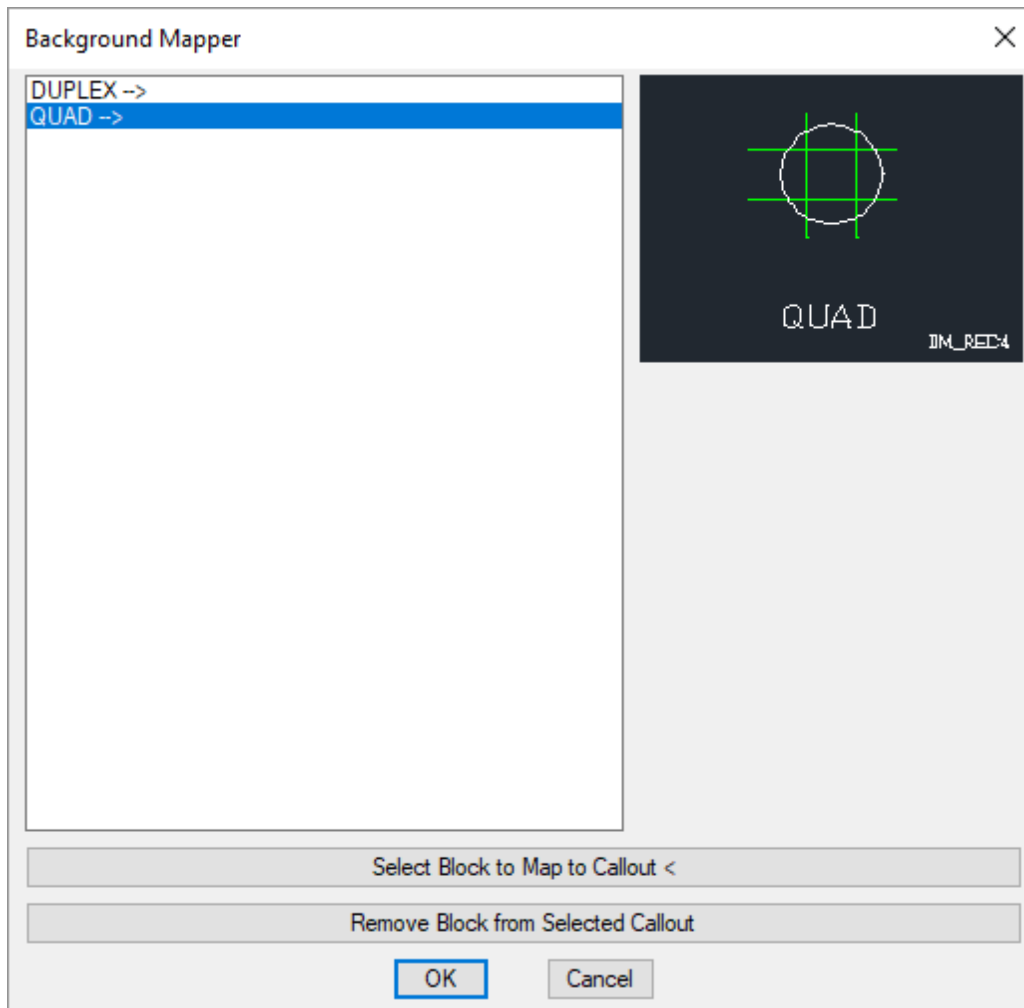
See the **Trace Architect's Light Fixtures** section for more information about how this command works. The **Background Mapper** dialog box works the same way in both commands.

Insert Receptacle Project Schedule

A receptacle schedule that lists all of the receptacles used in the current project can be inserted on the drawing.

To insert this schedule on the drawing, go to

Ribbon: `DM Electrical->Devices->` `Insert Receptacle Project Schedule`

Pulldown Menu: `DM Electrical->Receptacles->Insert Receptacle Project Schedule`

Edit Receptacle Group Settings Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Show Unused Receptacles: Whether receptacles in the schedule that have not been inserted on a drawing will be displayed in the schedule.

Show Subscript on Symbols: Whether subscript notes are displayed in the symbol column of the schedule. The subscript note is defined in the project schedule.

If this box is unchecked, the symbol column will display the receptacle symbol without the subscript note.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the receptacle schedule on the drawing. See the **Edit Receptacle Schedule Project List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Receptacles options: Set the values displayed in the *Features* column for each receptacle type if *Explosion-proof*, *Weather-proof*, *GFCI*, *Ground*, or *Isolated Ground* is set to **Yes**.

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Receptacle Project Schedule

The receptacle project schedule contains the receptacle types that can be used in the current project.

To create and modify receptacle types, go to

Ribbon: `DM Electrical->Devices->` REC `P  Receptacle Project Schedule`

Pulldown Menu: `DM Electrical->Receptacles->Receptacle Project Schedule`

Receptacle Project Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New, Copy, Save**, and **Delete** buttons.

See the **Common Receptacle Information** section for more information about the values that can be specified for receptacles using this dialog box.

See the **Import from Another Database** and **Import from Master Schedule** sections for more information about importing receptacles to the current project from other locations.

Current Group: The active receptacle group is listed here. All of the receptacle types that are created and modified will be associated with this group.

Select Group: Press this button to open the **Receptacle Project Schedule Groups** dialog box. The group selected will become the current group. See the **Schedule Groups** section for more information.

Select Block: Press this button to open the **Select Receptacle Block** dialog box. This dialog box allows you to choose the block used to represent the receptacle on the drawing.
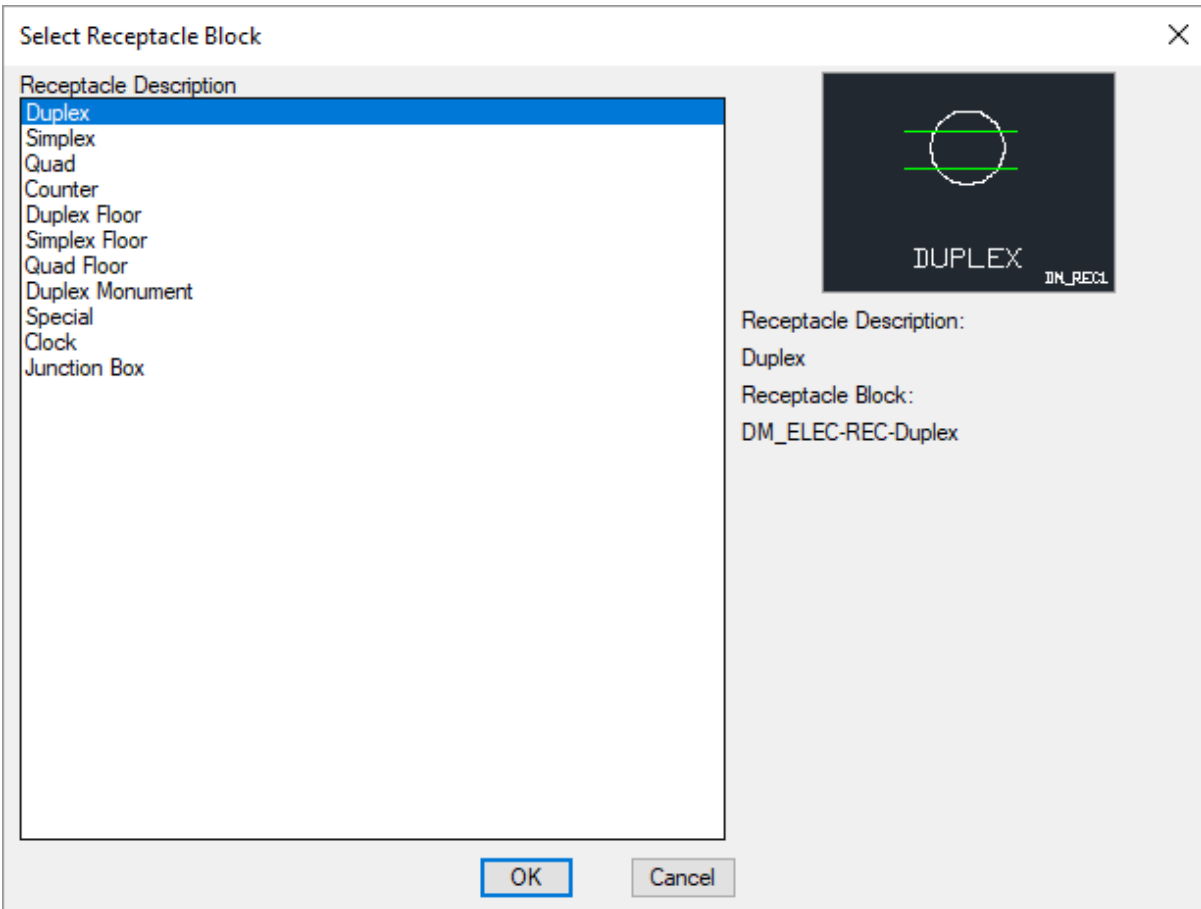
You can add more blocks to this list using the **Receptacle Blocks->Edit Project List** command.

Related Options

Receptacles options: Set the labels for the receptacle in the project schedule and on the drawing if *Explosion-proof*, *Weather-proof*, *GFCI*, *Ground*, or *Isolated Ground* is set to **Yes**.
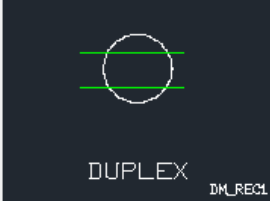
Receptacle Master Schedule

The receptacle master schedule contains the receptacle types that could be used by a company on a project. See the **Master and Standards Databases** section for more information about using master databases.

To create and modify receptacle types in the master schedule, go to

Ribbon: `DM Electrical->Devices->` `Receptacle Master Schedule`

Pulldown Menu: `DM Electrical->Receptacles->Receptacle Master Schedule`

Receptacle Master Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Common Receptacle Information** section for more information about the values that can be specified for receptacles using this dialog box.

See the **Import from Project Schedule** section for more information about importing receptacles from a project to the master database.

Current Group: The active receptacle group is listed here. All of the receptacle types that are created and modified will be associated with this group.

Select Group: Press this button to open the **Receptacle Master Schedule Groups** dialog box. The group selected will become the current group. See the **Schedule Groups** section for more information.

Select Block: Press this button to open the **Select Receptacle Block** dialog box. This dialog box allows you to choose the block used to represent the receptacle on the drawing.
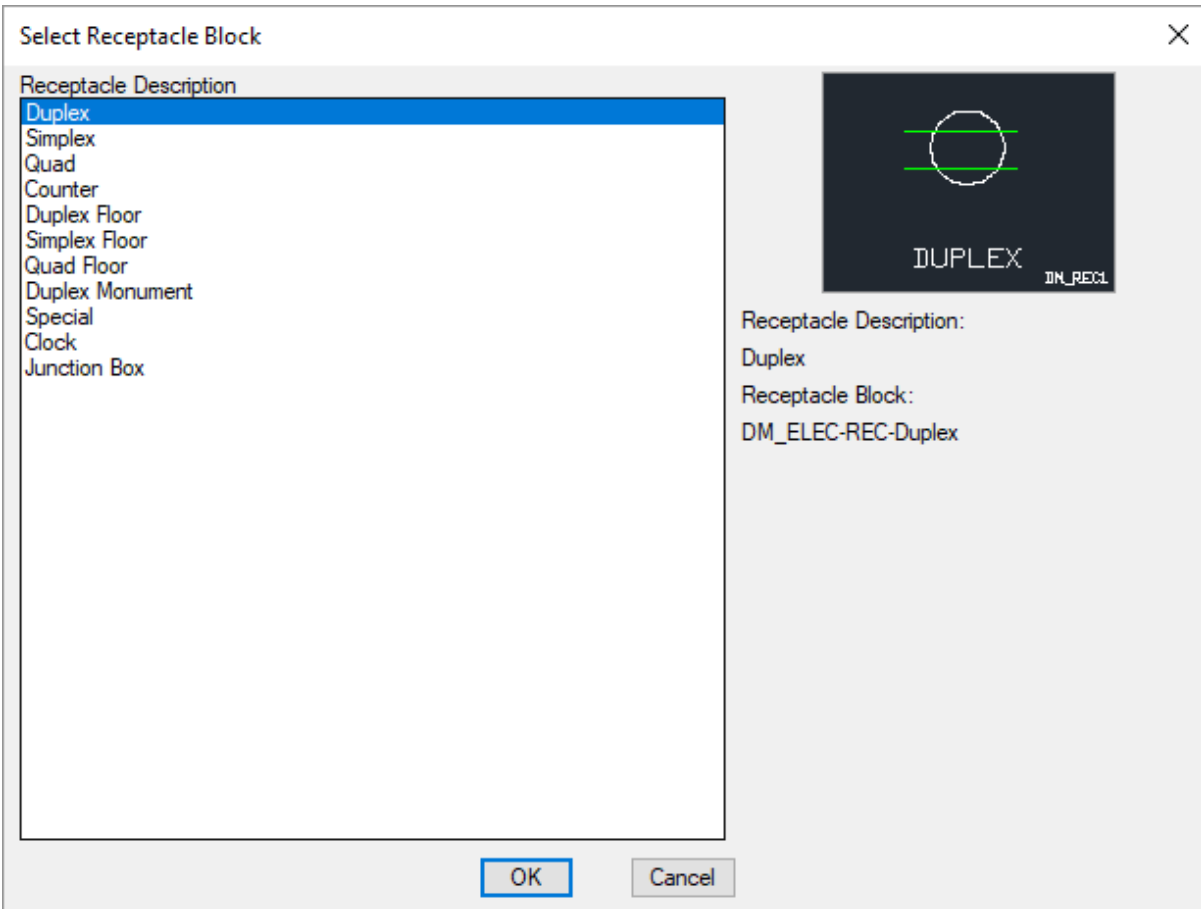
You can add more blocks to this list using the **Receptacle Blocks->Edit Master List** command.

Related Options

Receptacles options: Set the labels for the receptacle in the project schedule and on the drawing if *Explosion-proof*, *Weather-proof*, *GFCI*, *Ground*, or *Isolated Ground* is set to **Yes**.

Equipment

This section describes how to use equipment.

Equipment is used to model individual electrical connections that are typically not repeated in the design. Mechanical equipment and kitchen equipment are both examples of items that should be modeled as equipment.

Equipment can be inserted on the drawing or just exist in the database.

Training Videos

- **Equipment Connections**
- **Wire Sizing**

Common Equipment Information

The values that can be specified for equipment are listed below. All of these values must be set for each piece of equipment. Unlike receptacles and light fixtures, values cannot be pulled from a schedule.

Group: The group that the piece of equipment is in. All of the equipment in a group are displayed in a single list. For more information about equipment groups, see the **Equipment Projects Groups** section.

Equipment Callout: The name of the piece of equipment. The callout is used to label the equipment on the drawing and in the equipment list.

Panel Description: The value that is displayed in the circuit description for the circuit that the equipment is connected to. There are two options for this field:
- **Same as Equipment Callout:** The circuit description on the panel schedule will be the equipment callout. This is the default setting.
- **Custom:** The circuit description on the panel schedule will be set to a custom description, entered in the *Custom Panel Description* field. Custom descriptions are most useful when you need to shorten the label that is displayed on the panel schedule.

Circuit: The circuit to which the equipment is connected.

Total Connected Load (kVA): The sum of the four connected loads on the piece of equipment in kVA.

Total Demand Load (kVA): The sum of the four calculated loads on the piece of equipment in kVA.

Total Connected Load (Amps): The sum of the four calculated loads on the piece of equipment in amps. This load depends upon both the load in kVA and the voltage of the equipment connection.

Volts: The voltage for the piece of equipment.

Ground: Whether the equipment requires a ground wire.

Isolated Ground: Whether the equipment requires an isolated ground wire. If this is checked, an isolated ground wire is included in the wire callout for the branch circuit connected to this device.

Elevation: The elevation of the equipment connection. See the **Elevation** section for more information.

Layer System: See the **Layer System** section for more information.

Drawing Note 1, Drawing Note 2: These notes will be displayed on the drawing below the equipment callout.

Disconnect

This section describes information about the disconnect. Some of this information is displayed on the drawing and some is displayed in the schedule. Schedule-only fields that you do not plan to display can be ignored.

Type: The type of equipment connection and disconnect. The graphics that are displayed for the equipment on the drawing are dependent upon the equipment type. The type also determines whether a separate disconnect graphic is shown.

The types available in this list can be modified using the **Equipment Blocks** command.

Trip: The trip size of the disconnect, in amps.

Set Curve: Press this button to configure the curve for the disconnect. The **Select Curve** dialog box will appear.

Elevation: The elevation of the disconnect. This elevation is separate from the elevation of the equipment connection to allow for cases where the elevations are different. See the **Elevation** section for more information.

NEMA Rating: The NEMA rating of the disconnect. This value is displayed on the drawing as a note on the disconnect block.

Breaker and Wire Sizing

FLA/ BCSC: The full load amps or branch circuit selection current. This value is used to size the breaker according to *NEC 430.52* and *440.22*, depending upon the *Breaker Size* setting. It is also used in one of the *Wire Size* options.

MCA: The minimum circuit amps. This value is used to size the conductors to the equipment. It is not used in any of the calculations for the breaker size.

MOCP: The maximum overcurrent protection. This value is used when sizing motor-compressor breakers, but not when sizing motor breakers.

Breaker Size: How the breaker for the equipment is sized.
- **Motor-Compressor, <= MOCP (or loads if no MOCP):** Breaker will be sized to match the *MOCP*. The *FLA / BCSC* value will be ignored. If no *MOCP* value is specified, the breaker is sized based upon the connected load of the device. This is for *NEC 440.22(C)*.
- **Motor-Compressor, <= 175% of BCSC, <= MOCP:** Breaker will be sized to not exceed 175% of the *FLA / BCSC* value. The largest breaker that is less than 175% of the *FLA / BCSC* value will be used. If that is larger than the *MOCP*, the *MOCP* value will be used. This is for *NEC 440.22(A)* and *(C)*.
- **Motor-Compressor, <= 225% of BCSC, <= MOCP:** Breaker will be sized to not exceed 225% of the *FLA / BCSC* value. The largest breaker that is less than 225% of the *FLA / BCSC* value will be used. If that is larger than the *MOCP*, the *MOCP* value will be used. This is for *NEC 440.22(A) Exception no. 2* and *(C)*.
- **Motor, Dual Element Fuse, <= 175% of FLA:** Breaker will be sized to not exceed 175% of the *FLA / BCSC* value. The largest breaker that is less than 175% of the *FLA / BCSC* value will be used. This is for *NEC 430.52* dual element fuses.
- **Motor, Dual Element Fuse, <= 225% of FLA:** Breaker will be sized to not exceed 225% of the *FLA / BCSC* value. The largest breaker that is less than 225% of the *FLA / BCSC* value will be used. This is for *NEC 430.52(C)(1)(b)(2)*.
- **Motor, Inverse Time Breaker, <= 250% of FLA:** Breaker will be sized to not exceed 250% of the *FLA / BCSC* value. The largest breaker that is less than 250% of the *FLA / BCSC* value will be used. This is for *NEC 430.52* inverse time breakers.
- **Size based upon loads:** Breaker will be sized based upon the connected load of the device.
- **Specific Breaker Size:** The breaker is set to the specific size chosen from the list.

Set Curve: Press this button to configure the curve for the breaker. The **Select Curve** dialog box will appear.

Frame Size: How the frame for the equipment is sized.
- **Same as Breaker Size:** Frame will use the same size as the *Breaker Size*.
- **Specific Frame Sizes:** The frame is set to the specific size chosen from the list.

Wire Size: How the conductors for the equipment are sized.
- **Motor-Compressor, >= MCA (if no MCA, loads if motor, breaker if not):** Conductors will be sized to be

greater than the *MCA*. This is for *NEC 440.33* and *440.35*. If no MCA value is specified, the conductors are sized based upon the connected load of the device.
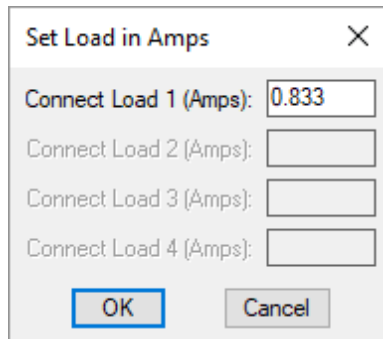
- **Single Motor-Compressor, >= 125% of BCSC:** Conductors will be sized to be greater than 125% of the *FLA / BCSC* value. This is for *NEC 440.32*.
- **<= MOCP:** Conductors will be sized to not exceed the *MOCP*.
- **Size based upon loads:** Conductors will be sized based upon the connected load of the device.
- **Size based upon breaker:** Conductors will be sized based upon the *Breaker Size*.
- **Specific Wire Sizes:** The conductors are sized to match the breaker size chosen from the list.

Loads

Load 1, Load 2, Load 3 and Load 4: An individual piece of equipment can have up to four separate assigned loads, each assigned to a different load type. See the **Load Information** section for more information.

When the *Horsepower* field is set, the connected kVA for the load is calculated based upon *NEC Tables 430.248* and *430.250*. If the *Connected* field is later changed, the *Horsepower* field is reset to be blank.

Set Load in Amps: Press this button to set the load on a piece of equipment in amps. The **Set Load in Amps** dialog box will appear.

Connected Load 1, Connected Load 2, Connected Load 3, Connected Load 4: The current on each load in amps. Each field will be disabled if the corresponding load is disabled in the main equipment dialog box.

Enter the load in amps for each connected load and press the **OK** button. The load will be converted to kVA based upon the *Volts* setting for the equipment and transferred to the corresponding *Connected* field.

Schedule Information

This section describes the **Schedule Information** button. The **Schedule Information** button is used to set information about the equipment that is displayed on the schedule. These values are not displayed or used anywhere else. Any fields you do not plan to display on the schedule can be ignored.

Show in List: Whether the equipment is displayed in the list on the drawing. See the **Insert Equipment List** section for more information about the equipment list.

Show in Fault Schedule: Whether the equipment is displayed in the fault schedule. See the **Insert Fault Schedule** section for more information about the fault schedule.

Show in Voltage Drop Schedule: Whether the equipment is displayed in the voltage drop schedule. See the **Insert Voltage Drop Schedule** section for more information about the voltage drop schedule.

Description: The description of the equipment.

Location: The location of the equipment.
- **Default:** The name of the room where the equipment is inserted will be displayed on the schedule.
- **Custom:** Enter the location in the field provided. This option is useful if you have not defined rooms in your project or want a location other than the room name displayed.

Sheet: The sheet on which the equipment is inserted.
- **Default:** The name of the drawing file where the equipment is inserted will be displayed on the schedule.
- **Custom:** Enter the sheet in the field provided.

Wire Callout: The callout of the branch circuit wire connected to the equipment.
- **Default:** The callout from the circuit is displayed on the schedule.
- **Custom:** Enter the wire callout in the field provided.

Control Wiring: Information about the control wiring for the equipment.

Provided By: Who is responsible for providing the equipment.

Installed By: Who is responsible for installing the equipment.

Schedule Note 1, Schedule Note 2, Schedule Note 3: These notes appear on the equipment schedule. See the **Schedule Notes** section for more information.

Equipment AIC Rating: The ampere interrupting capacity (AIC) rating for the device. This value is not automatically calculated. It is important to specifically choose this value for all of the equipment in the project.

When the **Custom** value is selected, a second *Equipment AIC Rating* field is available for manual input.

Fuse Size: The fuse size of the disconnect.

Fuse Type: The fuse type of the disconnect.

Description: The description of the disconnect.

Provided By: Who is responsible for providing the disconnect.

Installed By: Who is responsible for installing the disconnect.

Schedule Note 1, Schedule Note 2, Schedule Note 3: These notes appear on the equipment schedule to describe the disconnect. See the **Schedule Notes** section for more information.

Description: The description of the starter.

Thermal Overload: The thermal overload of the starter.

NEMA Size: The NEMA size of the starter.

NEMA Enclosure: The NEMA enclosure of the starter.

Provided By: Who is responsible for providing the starter.

Installed By: Who is responsible for installing the starter.

Schedule Note 1, Schedule Note 2, Schedule Note 3: These notes appear on the equipment schedule to describe the starter. See the **Schedule Notes** section for more information.

One-Line Diagram Settings

This section describes the **One-Line Diagram Settings** button. The **One-Line Diagram Settings** button is used to set the blocks for the equipment when it is displayed on the one-line diagram.

Show on One-Line Diagram: Whether the equipment is displayed on the one-line diagram. Most equipment will not have this box checked. When this box is not checked, the equipment will not be available to be inserted on the one-line diagram.

Initial Equipment Connection Block: The block used on the drawing for the equipment. The **Set Default** button allows you to change the *Default equipment connection block* in the **One-Line Diagram** options section.

Disconnect Block: The block used on the drawing for the disconnect. The **Set Default** button allows you to change the *Default equipment disconnect block* in the **One-Line Diagram** options section.

OCP Block: The block used on the drawing for the OCP.**<Use upstream distribution equipment OCP block>** uses the same OCP block as the upstream distribution equipment.

Create Equipment

To create equipment to place on the drawing or save in the project database, go to

Ribbon: `DM Electrical->Devices->` ⬡ `Create Equipment`

Pulldown Menu: `DM Electrical->Equipment->Create Equipment`

Create Equipment Dialog Box

See the **Common Equipment Information** section for more information about the values in this dialog box.

Creating Equipment on the Plan View

To create a piece of equipment to be added to the plan view, enter the information into the various fields of the dialog box and press the **Create on Plan View** button.

You will be prompted to insert the piece of equipment on the drawing.

Specify insertion point:

You will then be prompted for the rotation angle. If the equipment block is not able to be rotated, this prompt will be skipped.

Rotation angle <0>:

If the equipment has a disconnect block, you will then be prompted to insert the disconnect on the drawing.

Specify insertion point:
Rotation angle <0>:

You will then be prompted for the location of the equipment callout.

Right-click or press space to toggle the leader on or off, press enter to place the callout.

See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the callout.

Creating Equipment on the One-Line Diagram

To create a piece of equipment to be added to the one-line diagram, enter the information into the various fields of the dialog box and press the **Create on One-Line Diagram** button.

You will be prompted to insert the piece of equipment on the drawing.

Specify insertion point:

The equipment will be inserted on the drawing.

Creating Equipment in the Database

To create a piece of equipment in the database without inserting it on the drawing, enter the information into the various fields of the dialog box and press the **Create in Database** button.

The equipment will be added to the database and the following dialog box will appear.
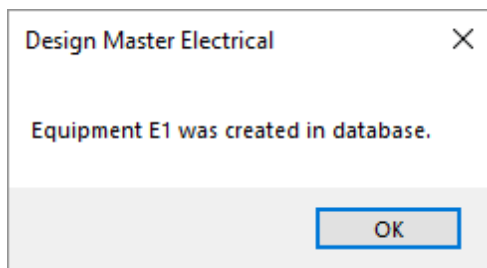
Related Options

Equipment callout: Sets the default information displayed in the equipment callout.

Insert 3D blocks on drawing: Sets whether a 3D block is inserted on the drawing along with the 2D block.
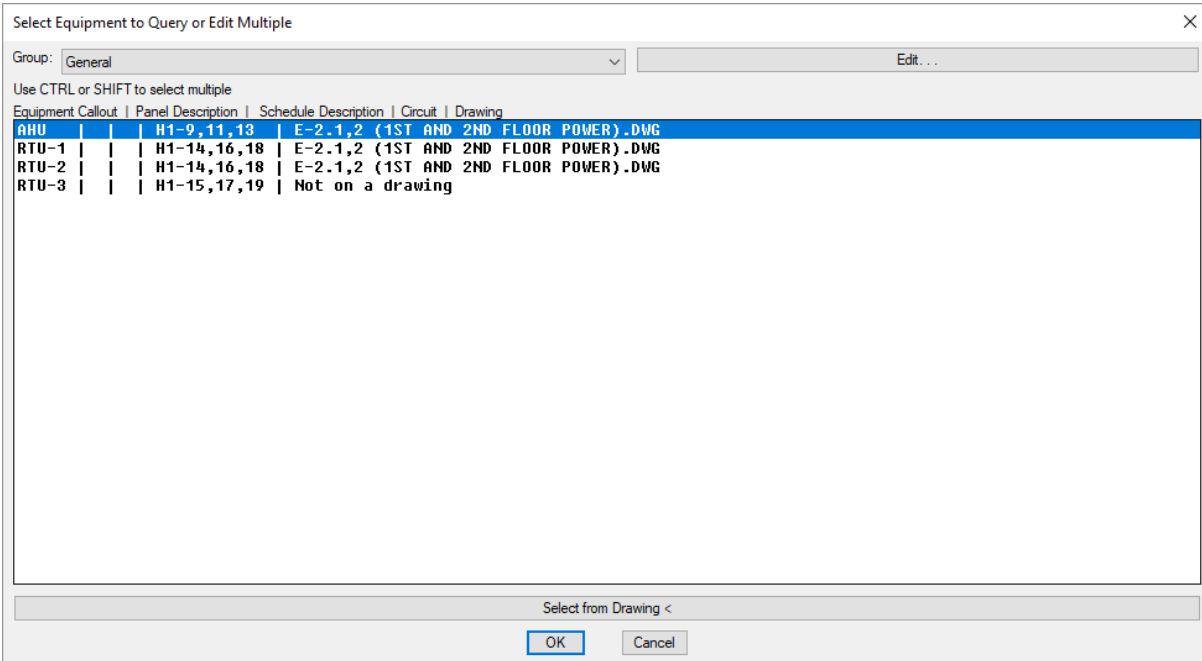
Query or Edit Multiple Equipment

To modify a piece of equipment that has been inserted on the drawing or in the database, go to

Ribbon: DM Electrical->Devices-> [EQU EM] Query or Edit Multiple Equipment

Pulldown Menu: DM Electrical->Equipment->Query or Edit Multiple Equipment

Select Equipment to Query or Edit Multiple Dialog Box

Group: A list of the equipment groups in the project. The equipment listed in the *Equipment* list are all part of the selected group. See the **Equipment Project Groups** section for more information.

Equipment: A list of all of the equipment in the database. You can select multiple items in the list using the `SHIFT` or `CTRL` keys.

Press the **OK** button to open the **Edit Multiple Equipment** dialog box.

Select from Drawing: Press this button to select an equipment on the drawing. The dialog box will close and you will be prompted to select an equipment.

Select objects:

The **Edit Multiple Equipment** dialog box will appear.

Edit Multiple Equipment Dialog Box

Changes can be made to the equipment settings when editing the equipment in the same way as when it was originally created. See the **Create Equipment** section for more information.

Circuit Equipment in Database

To circuit equipment in the database, go to

Ribbon: DM Electrical->Devices-> `CIR` Circuit Equipment in Database

Pulldown Menu: DM Electrical->Equipment->Circuit Equipment in Database

Circuit Equipment in Database Dialog Box

Group: A list of the equipment groups in the project. The equipment listed in the *Equipment* list are all part of the selected group. See the **Equipment Project Groups** section for more information.

Equipment: Select the equipment to circuit. The **Edit** button located below the list can be used to edit the list of *Equipment Callouts*.

Distribution Equipment: Select the distribution equipment to which the equipment will be conntected. The **Edit** button located below the list can be used to edit the list of *Distribution Equipment*. The **Not Circuited** option at the top of the list is used when the equipment is not connected to a distribution equipment.

Circuits: Select the circuit on the distribution equipment to which the equipment will be connected. For panels, the odd circuits are displayed in the left list and the even circuits are displayed in the right list. For all other types of distribution equipment, all of the circuits are displayed in the left list.

Circuit Equipment: Press this button to connect the selected equipment to the selected circuit.

If **Not Circuited** is selected in the *Distribution Equipment* list, the equipment will be disconnected from any distribution equipment it is connected to and be left uncircuited.

Insert Equipment on this Drawing

The `Insert Equipment on this Drawing` command is used to insert equipment on the drawing that currently exists in the database. Equipment that exists in the database can be created using the **Create Equipment** command, the **Remove Equipment from this Drawing** command, and the **Import Equipment from Another Project** command.

To insert a piece of equipment on the drawing that exists in the database, go to

Ribbon: `DM Electrical->Devices->` `Insert Equipment on this Drawing`

Pulldown Menu: `DM Electrical->Equipment->Insert Equipment on this Drawing`

Select Equipment to Insert on the Drawing Dialog Box
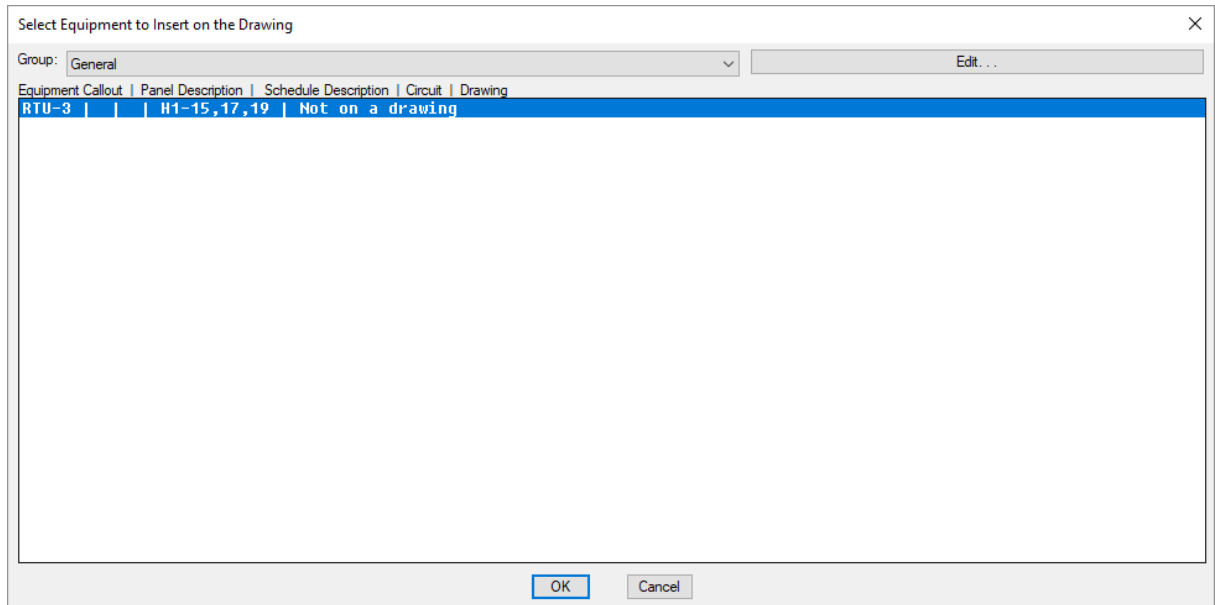
Group: A list of the equipment groups in the project. The equipment listed in the *Equipment* list are all part of the selected group. See the **Equipment Project Groups** section for more information.

A list of pieces of equipment in the group that are not are the drawing will be displayed. Equipment that is already inserted on the drawing will not be displayed, as it cannot be inserted again using this command.

Select the equipment to insert and press the **OK** button.

You will be prompted to insert the piece of equipment on the drawing.

Specify insertion point:

You will then be prompted for the rotation angle. If the equipment block is not able to be rotated, this prompt will be skipped.

Rotation angle <0>:

If the equipment has a disconnect block, you will then be prompted to insert the disconnect on the drawing.

Specify insertion point:
Rotation angle <0>:

You will then be prompted for the location of the equipment callout.

Right-click or press space to toggle the leader on or off, press enter to place the callout.

See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the callout.

Related Options

Equipment callout: Sets the default information displayed in the equipment callout.

Insert 3D blocks on drawing: Sets whether a 3D block is inserted on the drawing along with the 2D block.

Copy Equipment in Database

The `Copy Equipment in Database` command is used to copy equipment that is in the database but not on a drawing. To copy equipment on the drawing, use the standard CAD `COPY` command.

To copy equipment in the database, go to

Ribbon: `DM Electrical->Devices->` Copy Equipment in Database

Pulldown Menu: `DM Electrical->Equipment->Copy Equipment in Database`

Copy Equipment Dialog Box
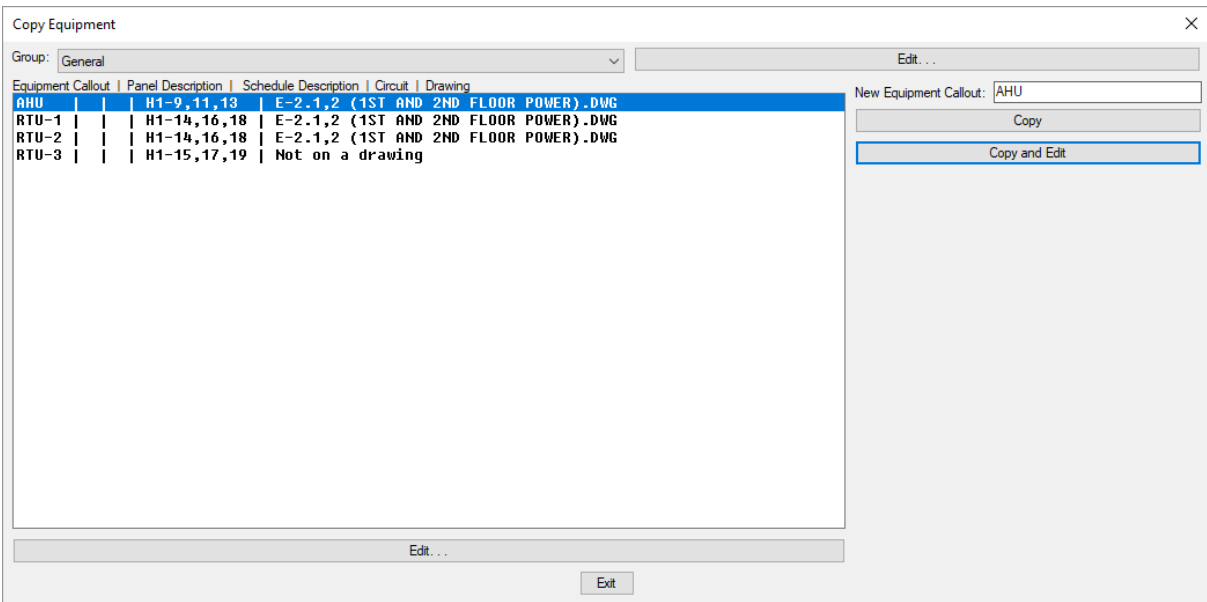
Group: A list of the equipment groups in the project. The equipment listed in the *Equipment* list are all part of the selected group. See the **Equipment Project Groups** section for more information.

Equipment: Select the equipment to copy. The **Edit** button located below the list can be used to edit the list of *Equipment Callouts*.

New Equipment Callout: The callout of the equipment that will be created. The default will be the same as the callout of the equipment that is being copied from.

Copy: Press this button to copy the selected equipment.

Copy and Edit: Press this button to copy the selected equipment and immediately start editing it. The **Edit Equipment** dialog box will open with the newly created equipment connection active. See the **Query or Edit Multiple Equipment** command for more information about editing the equipment.

Remove Equipment from this Drawing

The `Remove Equipment from this Drawing` command is used to remove equipment from a drawing while leaving it in the database.

To remove a piece of equipment from the drawing, go to

Ribbon: `DM Electrical->Devices->` Remove Equipment from this Drawing

Pulldown Menu: `DM Electrical->Equipment->Remove Equipment from this Drawing`

You will be prompted to select the equipment to remove from the drawing.

Select equipment to remove from drawing:

The selected equipment will be removed from the drawing but remain in the database. If it was connected to a circuit, it will remain connected.

You can use the **Insert Equipment on this Drawing** command to reinsert the equipment.

Delete Equipment from Database

The `Delete Equipment from Database` command is used to delete equipment that is in the database but not on a drawing. Equipment that is on a drawing can be deleted using the standard CAD **ERASE** command. Equipment not on the drawing can only be erased using the `Delete Equipment from Database` command.

To delete equipment that is in the database, go to

Ribbon: `DM Electrical->Devices->` Delete Equipment from Database

Pulldown Menu: `DM Electrical->Equipment->Delete Equipment from Database`

Delete Equipment from Database Dialog Box
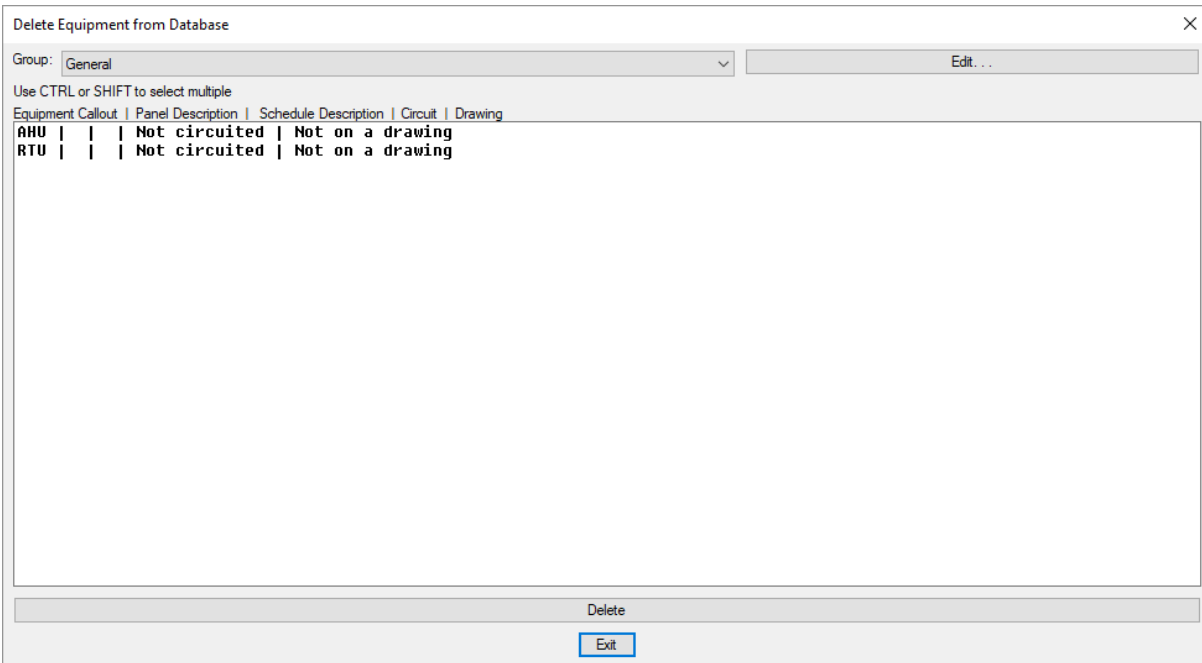
Group: A list of the equipment groups in the project. The equipment listed in the *Equipment* list are all part of the selected group. See the **Equipment Project Groups** section for more information.

Equipment: A list of all of the equipment in the database that is not on a drawing. You can select multiple items in the list using the `SHIFT` or `CTRL` keys.

Delete: Press this button to delete the selected equipment from the database. A **Design Master Electrical** dialog box will appear asking you to confirm the deletion.

Insert or Move Equipment Callout

To insert or move a callout for an equipment on the drawing, go to

Ribbon: `DM Electrical->Devices->` `Insert or Move Equipment Callout`

Pulldown Menu: `DM Electrical->Equipment->Insert or Move Equipment Callout`

You will be prompted to select the equipment for which the callout will be inserted or moved.

Select equipment or callout:

You will then be prompted for the location of the equipment callout.

Right-click or press space to toggle the leader on or off, press enter to place the callout.

See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the callout.

Related Options

Modify Equipment Callout

To change the information displayed in an equipment callout on the drawing, go to

Ribbon: `DM Electrical->Devices->` `Modify Equipment Callout`

Pulldown Menu: `DM Electrical->Equipment->Modify Equipment Callout`

You will be prompted to select the equipment callout to be modified.

Select equipment or callout:

Select Equipment Callout Style Dialog Box
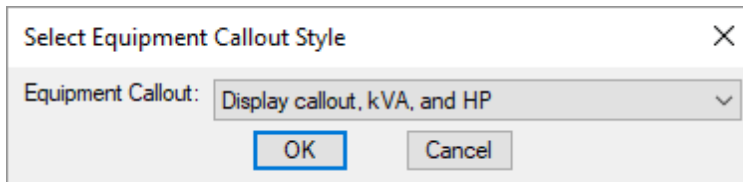
Equipment Callout: Select a callout style from the list.
- **Display callout, kVA, and HP:** The equipment callout, kVA, and horsepower will be displayed.
- **Display callout, amps, and HP:** The equipment callout, amps, and horsepower will be displayed.
- **Display callout and amps only:** The equipment callout and amps will be displayed.
- **Display callout only:** Only the equipment callout will be displayed.
- **No callout:** No equipment callout is displayed. If an equipment callout is currently inserted on the drawing, it will be removed.
- **Select custom block:** Select this option to specify a drawing for the equipment callout block.

Related Options

Equipment callout: Sets the default information displayed in the equipment callout.

Remove Equipment Callout

To remove an equipment callout on the drawing, go to

Ribbon: `DM Electrical->Devices->` `Remove Equipment Callout`

Pulldown Menu: `DM Electrical->Equipment->Remove Equipment Callout`

You will be prompted to select the equipment for which the callout will be removed.

Select equipment or callout:

The selected callout will be removed from the drawing. The callout can be reinserted with the **Insert or Move Equipment Callout** command.

Find Equipment on Drawing

To find a piece of equipment located on the current drawing, go to

Ribbon: `DM Electrical->Devices->` **EQU FIND** `Find Equipment on Drawing`

Pulldown Menu: `DM Electrical->Equipment->Find Equipment on Drawing`

Find Equipment Dialog Box
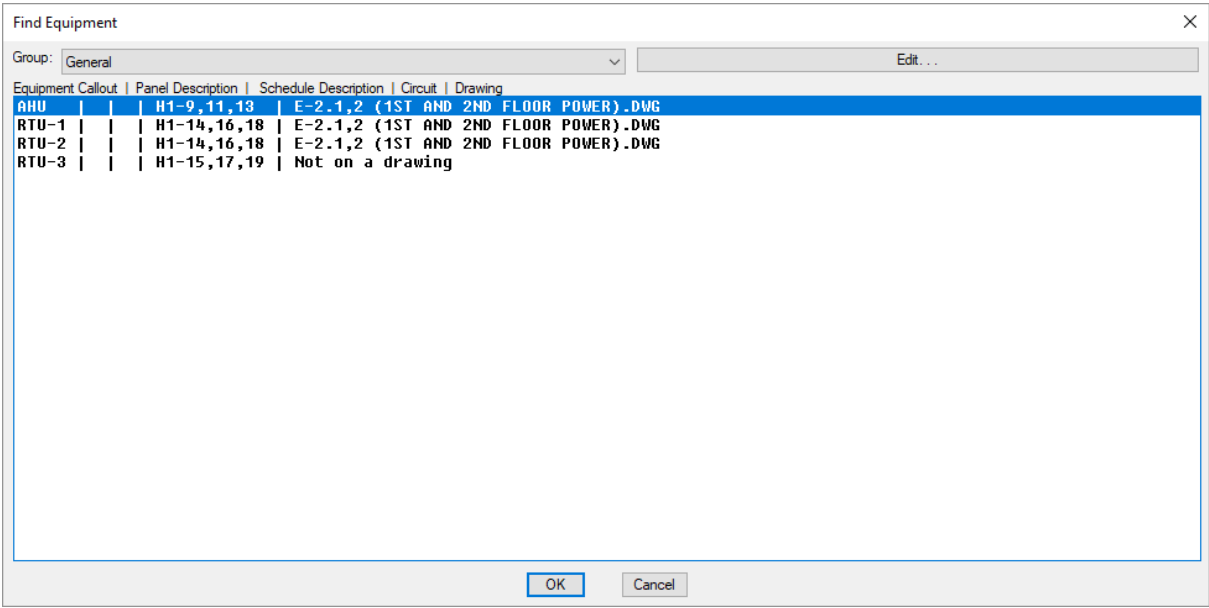
Group: A list of the equipment groups in the project. The equipment listed in the *Equipment* list are all part of the selected group. See the **Equipment Project Groups** section for more information.

Equipment: Select the equipment you want to find. The equipment callout, panel and schedule description, the circuit it is connected to, and the drawing are all displayed in this list. This additional information is provided to help you determine which equipment you want to find.

Press the **OK** button to find the selected equipment.

Import Equipment from Another Project

The `Import Equipment from Another Project` command is used to copy equipment created in another project to the current project. This command is useful when you want to reuse an equipment schedule from a previous project.

To import equipment from another project, go to

Ribbon: `DM Electrical->Devices->` **IMP EQU** `Import Equipment from Another Project`

Select Database to Import Equipment From Dialog Box
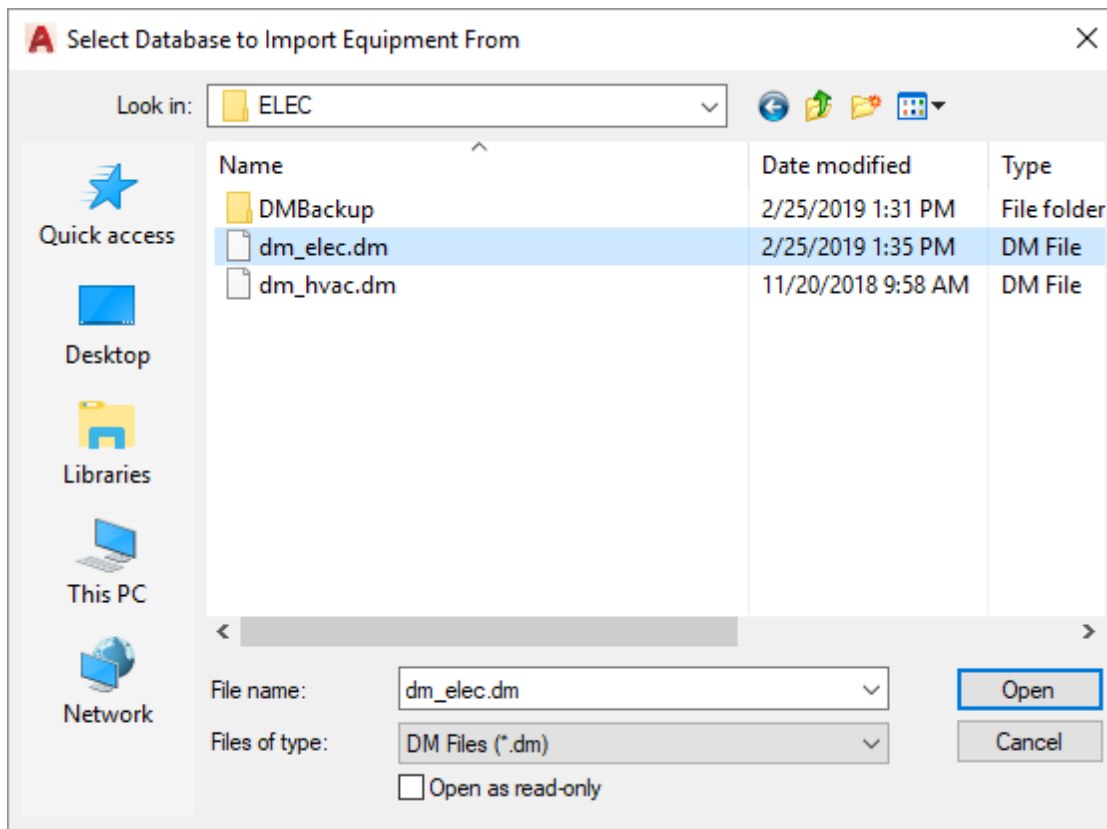
The dialog box will open with the folder for the current project displayed. Browse to the folder of the project you want to import from and select the "dm_elec.dm" file.

Press the **Open** button and the equipment from the selected project will be imported to the current project.

Insert Equipment List

The `Insert Equipment List` command is used to insert a list of equipment on the drawing. The list will include all of the equipment from a single group in the project. Separate lists can be inserted for each group.

To insert an equipment list, go to

Ribbon: `DM Electrical->Devices->` ⊞ `Insert Equipment List`

Pulldown Menu: `DM Electrical->Equipment->Insert Equipment List`

Edit Equipment Group Settings Dialog Box

Sort Column: How the list of equipment on the drawing will be sorted. If multiple pieces of equipment are being sorted on the same value, they are then sorted alphabetically by callout.

- **Callout:** The equipment is sorted alphabetically by callout.
- **Circuit:** The equipment is sorted alphabetically by panel name, then by circuit number. Equipment with no location is sorted to the top of the list.
- **Location:** The equipment is sorted by the value in the *Location* field.
- **Room:** The equipment is sorted by the name of the room in which the equipment is located. Equipment that is not inserted in a room is sorted to the top of the list.
- **Sheet:** The equipment is sorted by the value in the *Sheet* field.
- **Volts:** The equipment is sorted by the value in the *Volts* field.
- **kVA:** The equipment is sorted by the total connected load in kVA. The load is the sum of the four loads on the equipment connection.

Callout: Sets the callout style for the equipment group when equipment is inserted on the drawing. The default *Equipment callout* setting can be changed in the **Equipment** options section. See the **Modify Equipment Callout** section for more information.

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line

straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Show Equipment Not Inserting on Drawing: Whether equipment that is not inserted on a drawing will be displayed in the schedule.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the equipment schedule on the drawing. See the **Edit Equipment Project Schedule List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the list title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the list title for continued sections.

Equipment Project Groups

Equipment groups are used to categorize different types of equipment. Each group has a separate list that can be inserted on the drawing. These lists can have different formats to display different information for the different types of equipment.

To create and modify the equipment groups in your project, go to

Ribbon: `DM Electrical->Devices->` EQU P `Equipment Project Groups`

Pulldown Menu: `DM Electrical->Equipment->Equipment Project Groups`

Equipment Project Groups Dialog Box

See the **Schedule Groups** section for more information about creating groups of equipment in the project database.
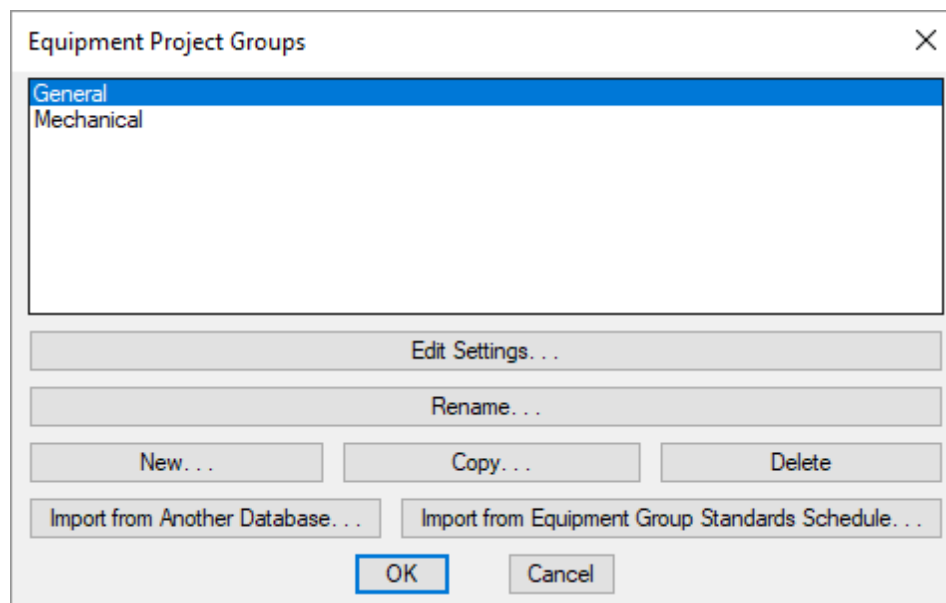
Edit Settings: Press this button to open the **Edit Equipment Group Settings** dialog box. See the **Insert Equipment List** section for more information.

Equipment Standards Groups

Settings for equipment groups can be created and stored in the standards database. These settings can be copied to new projects when they are created.

The equipment groups in the standards do not have any equipment in them, only the settings for the equipment list that is inserted on the drawing.

To create and modify equipment group settings in the standards database, go to

Ribbon: `DM Electrical->Devices->` EQU S `Equipment Standards Groups`

Pulldown Menu: `DM Electrical->Equipment->Equipment Standards Groups`

Equipment Standards Groups Dialog Box

See the **Schedule Groups** section for more information about creating equipment groups in the standards database.
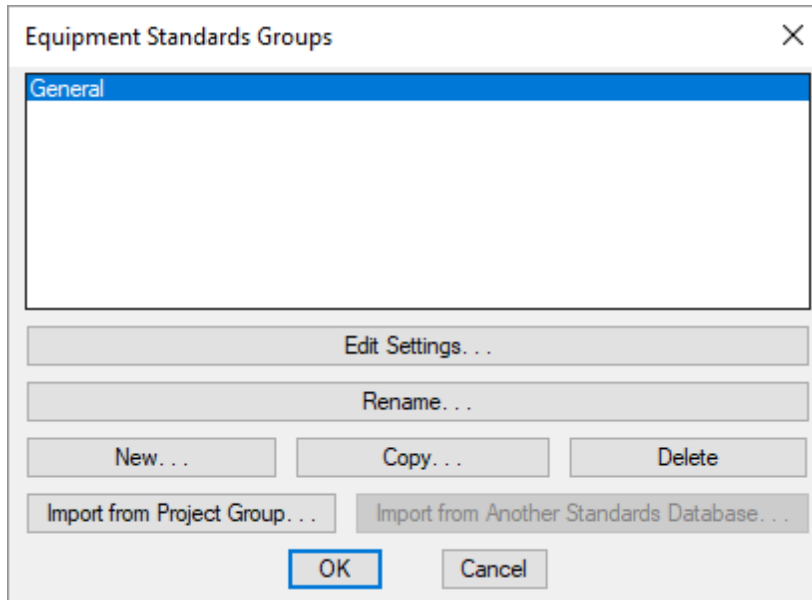
Edit Settings: Press this button to open the **Edit Equipment Group Settings** dialog box. See the **Insert Equipment List** section for more information.

Junction Boxes

This section describes how to use junction boxes. These junction boxes are used to connect device loops. They are connection points for wires only and do not have any associated loads.

If you need to create a junction box with a load, model it using either a **receptacle** or an **equipment connection**.

Junction boxes are never connected directly to a circuit. Any circuit they are connected to is based upon the devices that are connected to the junction box itself.

Common Junction Box Information

The values that can be specified for a junction box are listed below.

Include in Schedule: Whether the junction box is displayed in the junction box schedule that is inserted on the drawing. Most junction boxes will have this box checked. It is commonly unchecked for existing junction box types and junction boxes that are being used to model other types of devices in your project.

Schedule Note 1, Schedule Note 2, Schedule Note 3: These notes will appear on the junction box schedule on the drawing. See the **Schedule Notes** section for more information.

Insert Junction Box

To insert a junction box on a drawing, go to

Ribbon: `DM Electrical->Devices->` `Insert Junction Box`

Pulldown Menu: `DM Electrical->Junction Boxes->Insert Junction Box`

Insert Junction Box Dialog Box
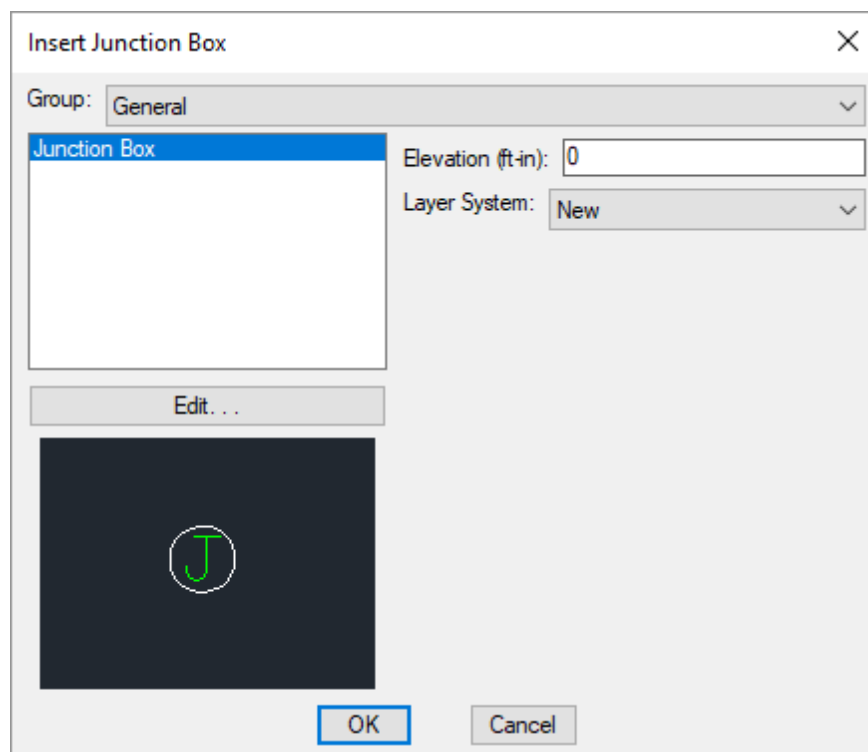

Junction Box Type: The type of junction box to be inserted on the drawing. Press the **Edit** button to access the **Junction Box Project Schedule** dialog box, where the list of available junction boxes is defined.

Elevation: The elevation of the junction box. See the **Elevation** section for more information.

Layer System: See the **Layer System** section for more information.

Inserting the Junction Box on the Drawing

To insert a junction box on the drawing, press the **OK** button. The settings specified in the dialog box will be used for the inserted junction box. You will be prompted to identify where on the drawing the junction box is to be inserted.

Specify insertion point:

You will then be prompted for the rotation angle. If the junction box block is not able to be rotated, this prompt will be skipped.

Specify rotation angle <0>:

If there are any notes included with the junction box, you will then be prompted to insert the notes on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the notes.

You will then be prompted to insert another junction box of the same type. Continue to insert junction boxes, or press **ENTER** to finish the command.

Related Options

Insert 3D blocks on drawing: Sets whether a 3D block is inserted on the drawing along with the 2D block.

Querying Junction Boxes

To modify a junction box that is inserted on the drawing, go to

Ribbon: `DM Electrical->Devices->` `Query Device`

Pulldown Menu: `DM Electrical->Modify Devices->Query Device`

Select a junction box on the drawing. The information in the database regarding the selected junction box will be displayed in the **Query Junction Box** dialog box.

Query Junction Box Dialog Box
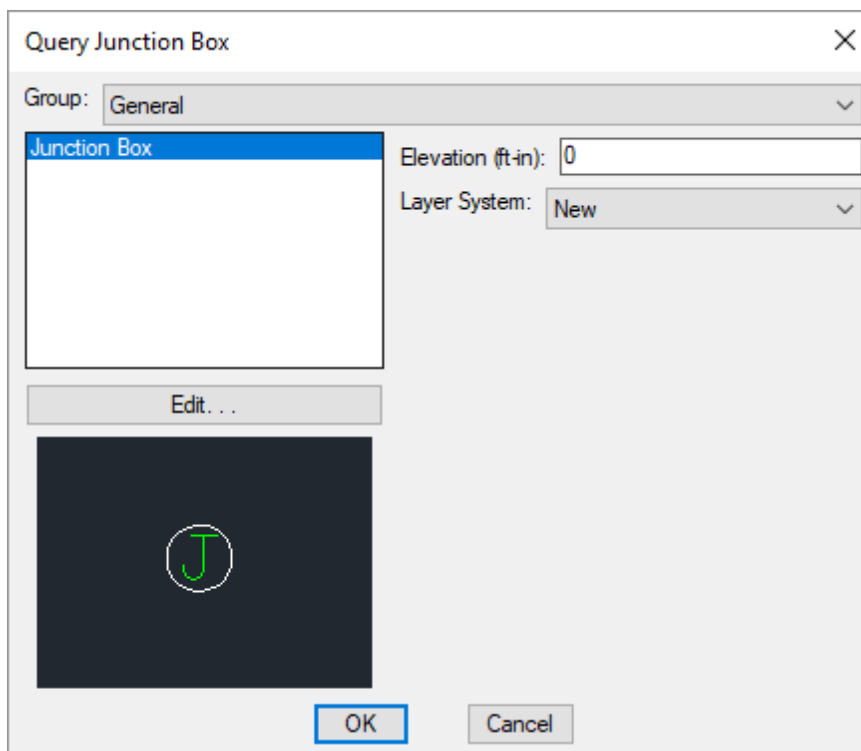
Changes can be made to the junction box in the same way as when it was inserted. See the **Insert Junction Boxes** command for more information.

Edit Multiple Junction Boxes

To edit multiple junction boxes, go to

Ribbon: `DM Electrical->Devices->` `Edit Multiple Junction Boxes`
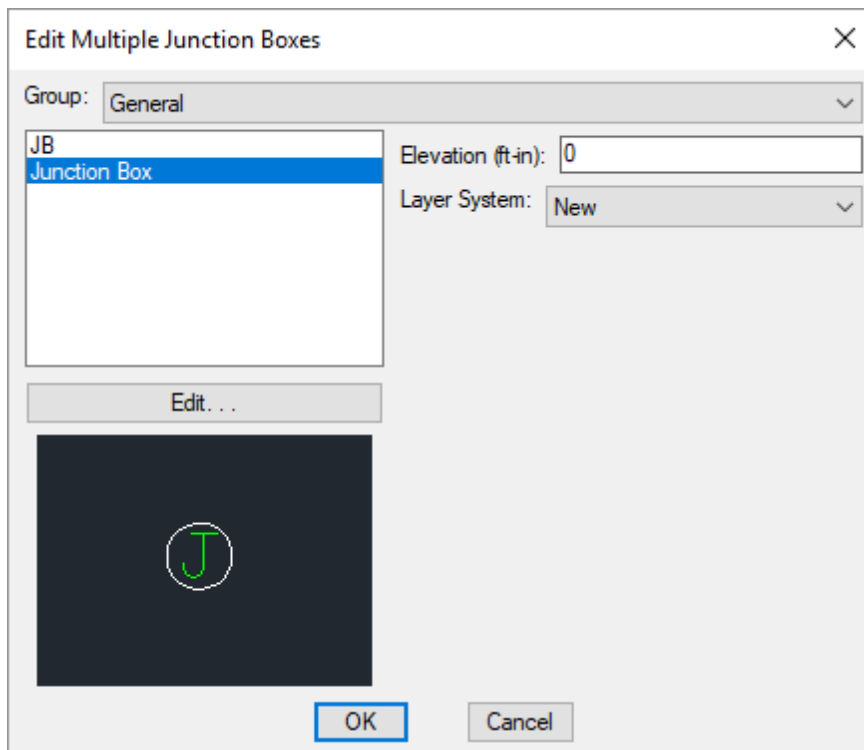
Pulldown Menu: `DM Electrical->Junction Boxes->Edit Multiple Junction Boxes`

You will be prompted to select the junction boxes to be edited.

Select junction boxes to edit:

The **Edit Multiple Junction Boxes** dialog box will appear.

Edit Multiple Junction Boxes Dialog Box

Changes can be made to the junction boxes in the same way as when they were inserted. See the **Insert Junction Boxes** section for more information.

Add Device to Junction Box

To connect devices to a junction box, go to

Ribbon: `DM Electrical->Devices->` `Add Device to Junction Box`

Pulldown Menu: `DM Electrical->Junction Boxes->Add Device to Junction Box`

You will be prompted to select the junction box.

Select junction box:

You will then be prompted to select the devices you want to associate with the junction box.

Select objects:

The selected junction box and devices are now associated with each other. When loops are drawn using the **Draw Homerun and Loops** command, these devices will be directly connected to the junction box. The junction box will be connected to other devices on the circuit. Nothing else will be connected to the devices.

If the device was previously connected to a different junction box, it will now be connected to this junction box. You do not need to disconnect the device from the old junction box before connecting it to a new junction box.

Although the drawing indicates the junction box and devices are associated by the loops drawn, the devices have not been circuited. You still need to connect the devices to the appropriate circuit.

Remove Device from Junction Box

To remove a device from a junction box, go to

Ribbon: `DM Electrical->Devices->` `Remove Device from Junction Box`

Pulldown Menu: `DM Electrical->Junction Boxes->Remove Device from Junction Box`

You will be prompted to select the devices you want to remove.

Select objects:

All of the devices selected will be disconnected from the junction box to which they are connected. The loop between the device and the junction box will be removed.

If you want to connect a device to a different junction box, use the **Add Device to Junction Box** command. You do not need to use the `Remove Device from Junction Box` command to move a device from one junction box to another.

Insert Junction Box Project Schedule

A junction box schedule that lists all of the junction boxes used in the current project can be inserted on the drawing.

To insert this schedule onto the drawing, go to

Ribbon: `DM Electrical->Devices->` `Insert Junction Box Project Schedule`

Pulldown Menu: `DM Electrical->Junction Boxes->Insert Junction Box Project Schedule`

Edit Junction Box Group Settings Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Show Unused Junction Boxes: Whether junction boxes in the schedule that have not been inserted on a drawing will be displayed in the schedule.

Edit Title, Column Labels, and Column Order: Use this button to modify the layout of the junction box schedule on the drawing. See the **Edit Junction Box Project Schedule List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Junction Box Project Schedule

The junction box project schedule contains the junction box types used in the current project.

To create and modify junction box types, go to

Ribbon: DM Electrical->Devices-> P Junction Box Project Schedule

Pulldown Menu: DM Electrical->Junction Boxes->Junction Box Project Schedule

Junction Box Project Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Common Junction Box Information** section for more information about the values that can be specified for junction boxes using this dialog box.

See the **Import from Another Database** and **Import from Master Schedule** sections for more information about importing junction boxes to the current project from other locations.

Current Group: The active junction box group is listed here. All of the junction box types that are created and modified will be associated with this group.

Select Group: Press this button to open the **Junction Box Project Schedule Groups** dialog box. The group selected will become the current group. See the **Schedule Groups** section for more information.

Select Block: Press this button to open the **Select Junction Box Block** dialog box. This dialog box allows you to choose the block used to represent the junction box on the drawing.
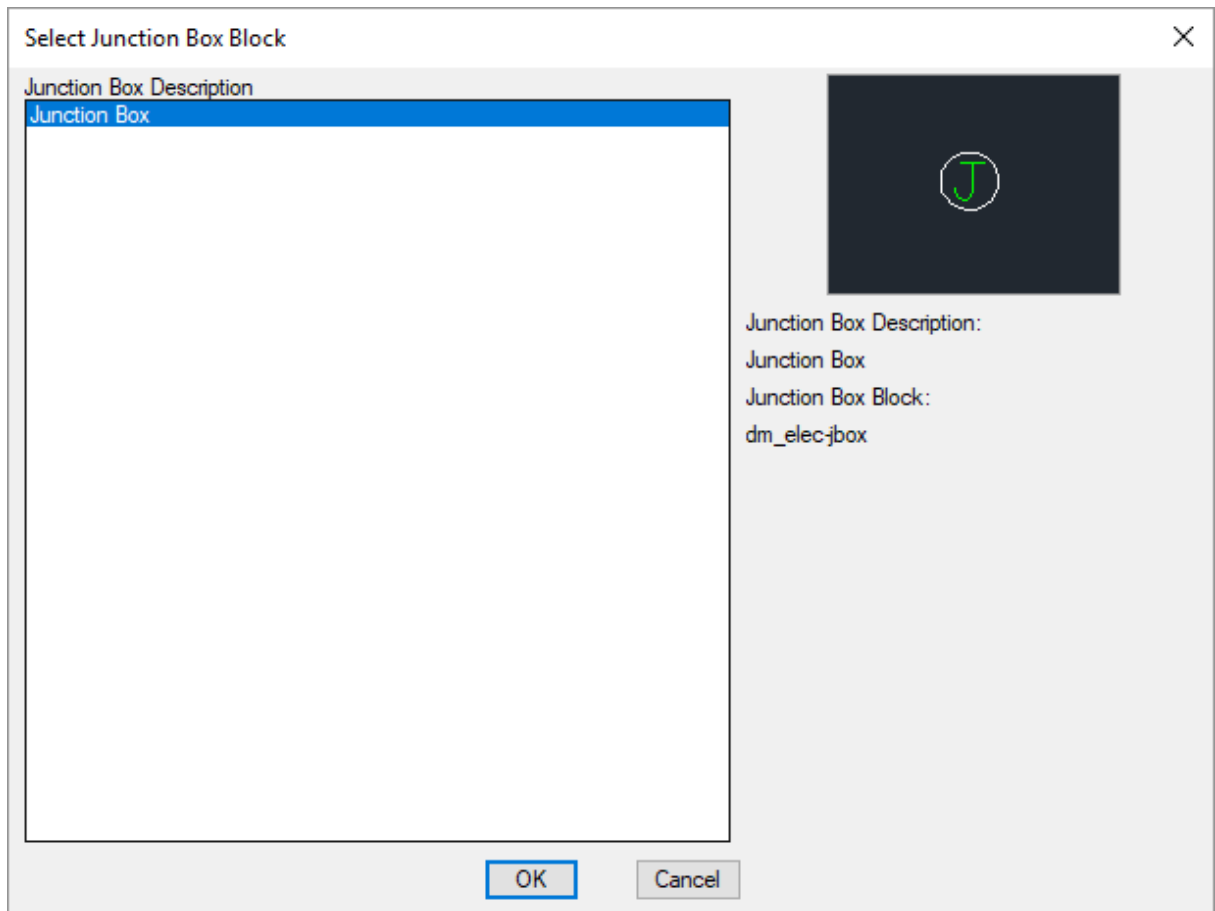
You can add more blocks to this list using the **Junction Box Blocks->Edit Project List** command.

Junction Box Master Schedule

The junction box master schedule contains the junction box types that could be used by a company on a project. See the **Master and Standards Databases** section for more information about using master databases.

To create and modify junction box types in the master schedule, go to

Ribbon: DM Electrical->Devices-> ![JBOX M icon] Junction Box Master Schedule

Pulldown Menu: DM Electrical->Junction Boxes->Junction Box Master Schedule

Junction Box Master Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next**, **New**, **Copy**, **Save**, and **Delete** buttons.

See the **Common Junction Box Information** section for more information about the values that can be specified for junction boxes using this dialog box.

See the **Import from Project Schedule** section for more information about importing junction boxes from a project to the master database.

Current Group: The active junction box group is listed here. All of the junction box types that are created and modified will be associated with this group.

Select Group: Press this button to open the **Junction Box Master Schedule Groups** dialog box. The group selected will become the current group. See the **Schedule Groups** section for more information.

Select Block: Press this button to open the **Select Junction Box Block** dialog box. This dialog allows you to choose the block used to represent the receptacle on the drawing.
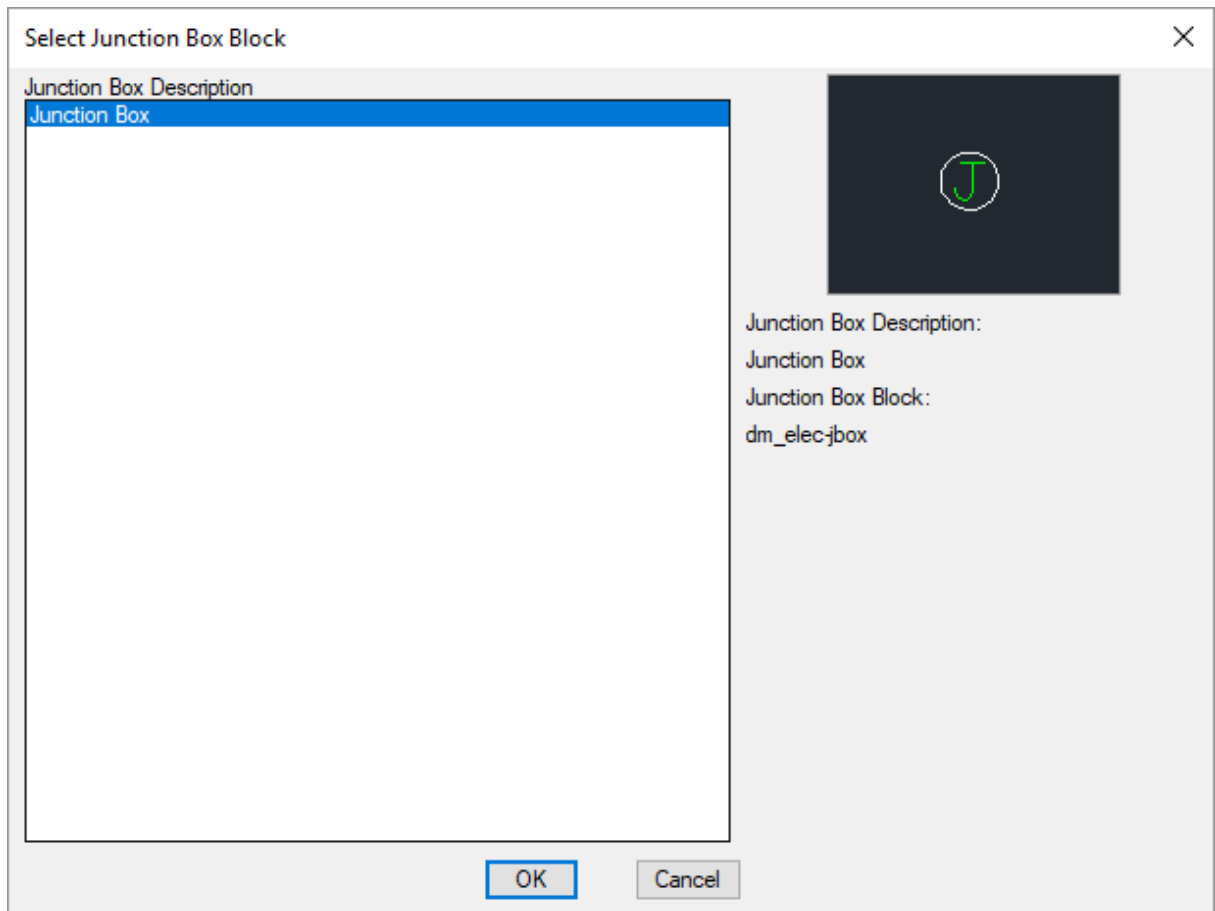
You can add more blocks to this list using the **Junction Box Blocks->Edit Master List** command.

Switches

This section describes how to use switches.

Common Switch Information

The values that can be specified for a switch are listed below. Some values can be set in the schedule only. Others can have a default set in the schedule that can be changed on individual switches.

Schedule-Only Settings

This section describes switch values that can only be set in the schedule. These cannot be changed on individual switches. If you need to insert a switch that has a different setting for one of these values, you must create a new record in the schedule.

Include in Schedule: Whether the switch type is displayed in the switch schedule that is inserted on the drawing. Most switch types will have this box checked. It is commonly unchecked for existing switch types and switches that are being used to model other types of devices in your project.

Default Subscript: The default value for the subscript inserted with the switch. Subscripts are different from

other notes in that they rotate with the switch, rather than always being displayed at angle 0.

Variable Subscript: Whether the subscript can be changed for individual switches. If this is unchecked, all switches of this type will use the *Default Subscript* for their *Subscript*. If this is checked, the *Subscript* can be changed for individual switches.

Schedule Note 1, Schedule Note 2, Schedule Note 3: These notes will appear on the switch schedule on the drawing. See the **Schedule Notes** section for more information.

Individual Switch Settings

This section describes switch values that be changed for individual switches. The value in the switch schedule is the default value.

Circuit: The circuit to which the switch is connected.

Drawing Note 1, Drawing Note 2, Drawing Note 3: These notes will appear on the drawing for each switch.

Switch Groups

A switch group consists of switches, devices, a group label, and a circuit.

Switch groups contain one or more switches. If they contain multiple switches, the group represents an n-way switch. You can have any number of switches in a group.

Switch groups can contain zero, one, or more devices. All of the devices in the group are controlled by the switches in the group.

The switch group has a common *Group* label that is set when inserting or editing a switch. Changing the *Group* label on one switch in the group changes it for all of the switches. The *Group* label is inserted on the drawing next to all of the switches and devices in the group.

The switch group can be connected to a circuit on a distribution equipment. All of the switches and devices will be connected to a single circuit. If you change the circuit connected to a switch, it will change the circuit for the whole group. If you change the circuit for a device in the group, you will be given the option to change the circuit for the whole group or to remove the selected device from the group.
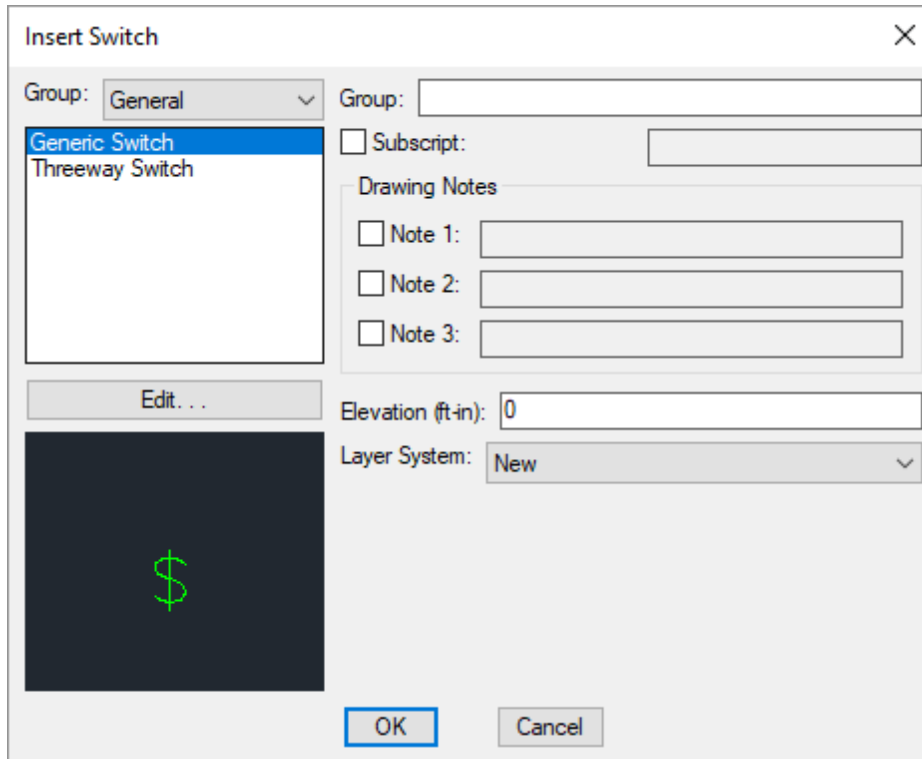
Insert Switch

To insert a switch on a drawing, go to

Ribbon: `DM Electrical->Devices->` $ `Insert Switch`

Pulldown Menu: `DM Electrical->Switches->Insert Switch`

Insert Switch Dialog Box

Switch Type: Select the type of switch to be inserted. The list of switches available is defined in the **Switch Project Schedule** command.

The default values for the switch are specified based upon the chosen switch type.

Group: This value is used to indicate the group to which a switch belongs, typically a lower case letter. This value is displayed next to the switch and the devices connected to the switch, indicating which switch controls which fixtures in a room. If the group label on a switch is modified, all connected devices will be updated automatically.

Subscript: The subscript for the switch. Subscripts are inserted like notes, but are rotated to match the angle of the switch. If *Variable Subscript* is not checked in the switch project schedule, the subscript value will be fixed for the individual switch and cannot be changed.

Elevation: The elevation of the switch. See the **Elevation** section for more information.

Layer System: See the **Layer System** section for more information.

See the **Common Switch Information** section for more information about the *Drawing Notes* in this dialog box.

Inserting the Switch on the Drawing

To insert a switch on the drawing, press the **OK** button. The settings specified in the dialog box will be used tor the inserted switches. You will be prompted to insert them on the drawing.

Specify insertion point:

You will then be prompted for the rotation angle. If the switch block is not able to be rotated, this prompt will be skipped.

Specify rotation angle <0>:

If there are any notes included with the switch, you will then be prompted to insert the notes on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the notes.

You will then be prompted to insert another switch of the same type. Continue to insert switches, or press **ENTER** to finish the command.

Related Options

Switch rotation: Sets how the subscript is rotated relative to the switch block.

Allow upside down switch subscripts: Sets whether the subscript is displayed upside down if the switch is inserted upside down on the drawing.

Provide neutral wire to switches: Sets whether a neutral is provided to switches, in compliance with *2011 NEC 404.2(C)*.

Insert 3D blocks on drawing: Sets whether a 3D block is inserted on the drawing along with the 2D block.
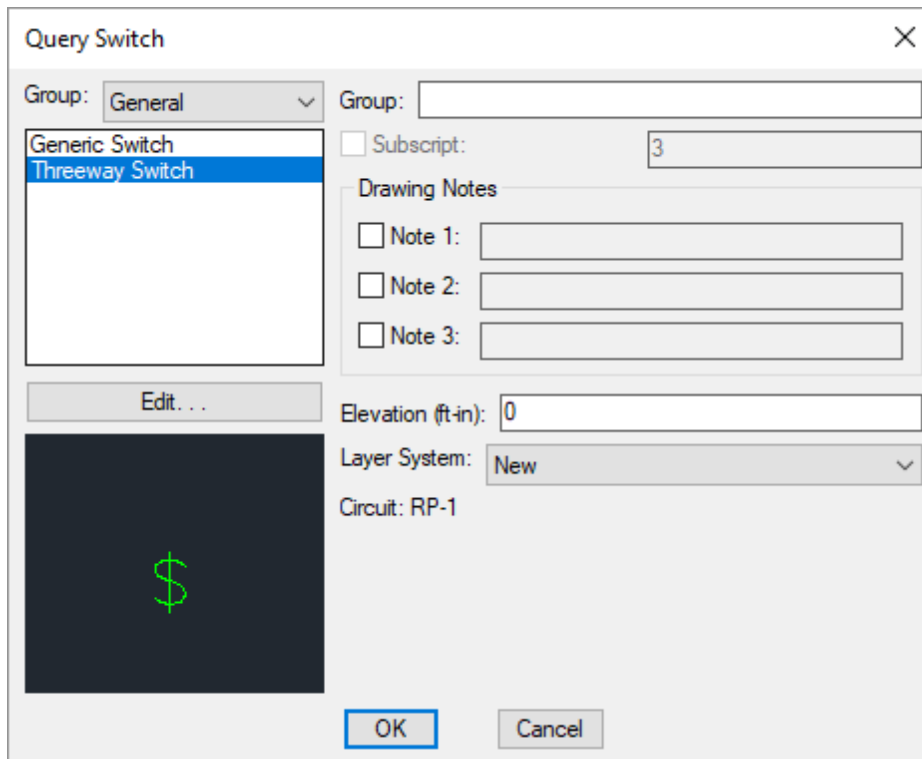
Querying Switches

To modify a switch box that is inserted on the drawing, go to

Ribbon: `DM Electrical->Devices->` `Query Device`

Pulldown Menu: `DM Electrical->Modify Devices->`Query Device

Select a switch on the drawing. The information in the database regarding the selected switch will be displayed in the **Query Switch** dialog box.

Query Switch Dialog Box

Changes can be made to the switch in the same way as when it was inserted. See the **Insert Switch** section for more information.

Edit Multiple Switches

To edit multiple switches, go to

Ribbon: `DM Electrical->Devices->` `Edit Multiple Switches`
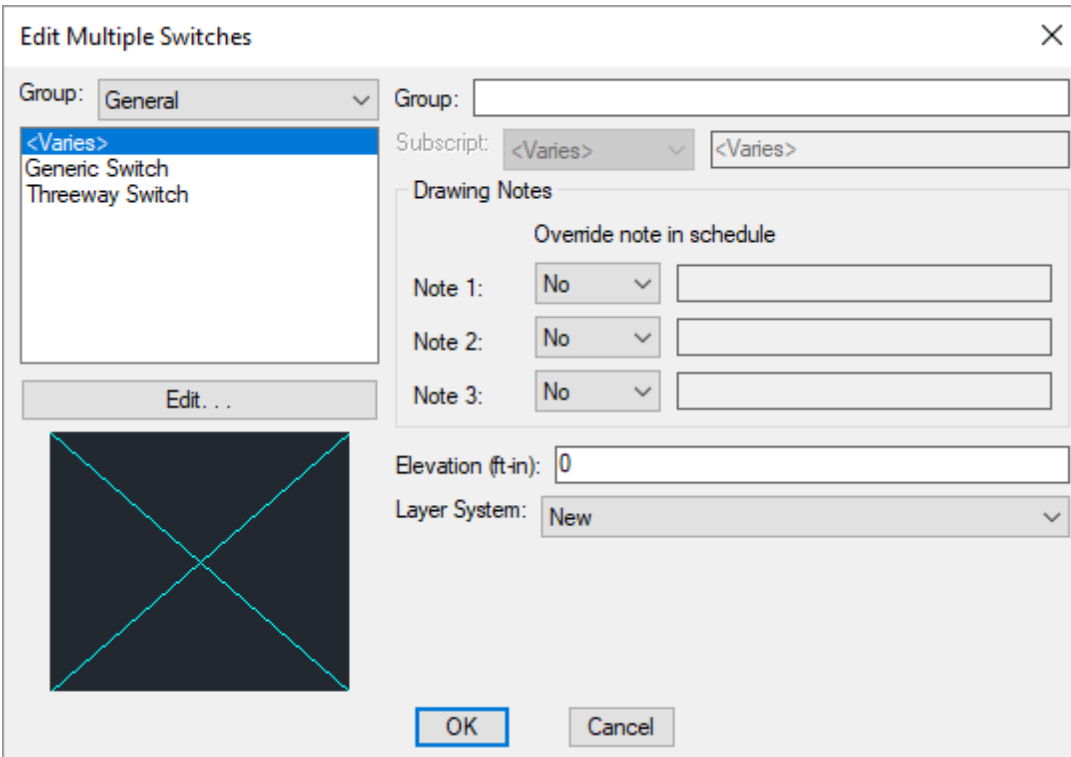
Pulldown Menu: `DM Electrical->Switches->Edit Multiple Switches`

You will be prompted to select the switches to be edited.

Select switches to edit:

The **Edit Multiple Switches** dialog box will appear.

Edit Multiple Switches Dialog Box

Changes can be made to the switches in the same way as when they were inserted. See the **Insert Switch** section for more information.

Trace Architect's Switches

The `Trace Architect's Switches` command is used to trace a switch layout from another drawing that does not have Design Master entities.

Before using this command, create switches using the **Switch Project Schedule** command.

This command should be run on the drawing from which you are copying the switch layout. Do not run this command on the drawing you are using for your electrical design.
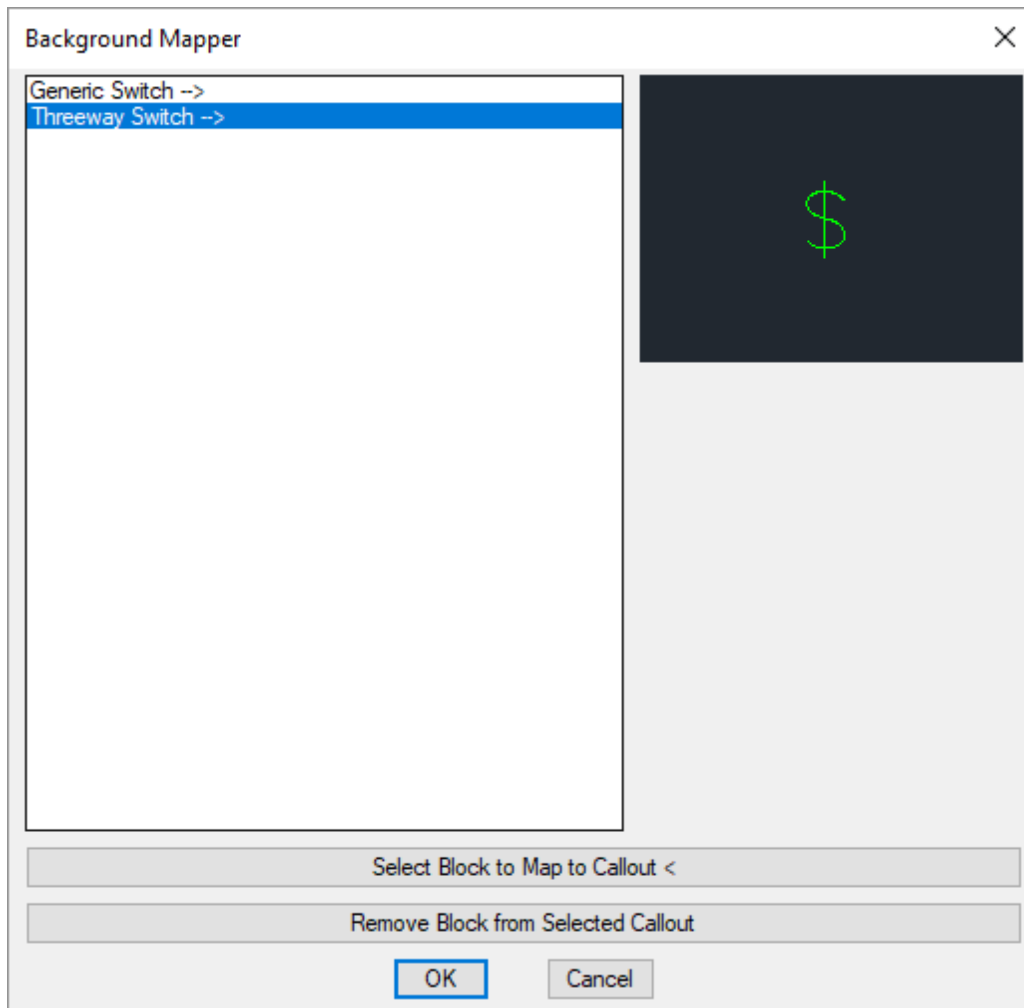
To trace a switch layout, go to

Ribbon: `DM Electrical->Devices->` `SW TRACE` `Trace Architect's Switches`

Pulldown Menu: `DM Electrical->Receptacles->Trace Architect's Switches`

The **Background Mapper** dialog box will appear.

Background Mapper Dialog Box

See the **Trace Architect's Light Fixtures** section for more information about how this command works. The **Background Mapper** dialog box works the same in these two commands.

Connect Device to Switch Group

The `Connect Device to Switch Group` command is used to connect devices to a switch group. See the **Switch Groups** section for more information.

To connect one or more devices to a switch, go to

Ribbon: `DM Electrical->Devices->` `Connect Device to Switch Group`

Pulldown Menu: `DM Electrical->Switches->Connect Device to Switch Group`

You will be prompted to select a switch.

Select switch:

You will then be prompted to select the devices you want to connect.

Select objects:

The devices will be added to the switch group associated with the switch you selected. A label indicating the switch group will be added to each device.

Remove Device from Switch Group

The `Remove Device to Switch Group` command is used to remove devices from a switch group. See the **Switch Groups** section for more information.

To remove a device from a switch group, go to

Ribbon: `DM Electrical->Devices->` 🧩 `Remove Device to Switch Group`

Pulldown Menu: `DM Electrical->Switches->Remove Device from Switch Group`

You will be prompted to select one or more devices.

Select devices to remove from their switch groups:

All of the selected devices will be removed from the switch group they are in. Any switch group label associated with the device will be removed from the drawing.

If the switch group is connected to a circuit, you will be asked whether the device should be removed from the circuit.

Add Switch to Switch Group

The `Add Switch to Switch Group` command is used to add a switch to another switch group. See the **Switch Groups** section for more information.

To add a switch to a switch group, go to

Ribbon: `DM Electrical->Devices->` 🧩 `Add Switch to Switch Group`

Pulldown Menu: `DM Electrical->Switches->Add Switch to Switch Group`

You will be prompted to select a switch that is already in the group to which you would like to add another switch.

Select a switch in the group:

You will then be prompted to select the switch to add to that group.

Select objects:

The second switch you select will be added to the group associated with the first switch. If the second switch was associated with another group that contains other switches or devices, the **Switch is Already Grouped** dialog box will appear.
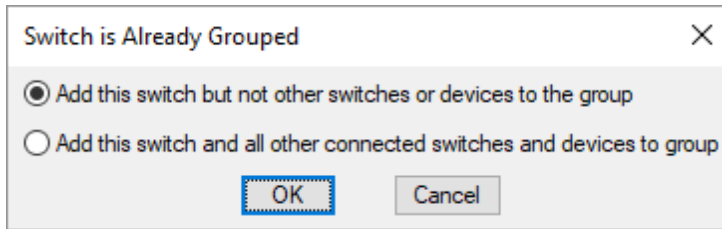
Add this switch but not other switches or devices to the group: The selected switch will be moved to the new switch group. All of the other switches and devices will remain in the other switch group. The switch will be connected to the circuit associated with the new switch group.

Add this switch and all other connected switches and devices to the group: Everything in the switch group associated with the second switch will be added to the new switch group. The switches and devices will be connected to the circuit associated with the new switch group.

Remove Switch from Switch Group

The `Remove Switch from Switch Group` command is used to remove a switch from a switch group. See the **Switch Groups** section for more information.

To remove a switch from a switch group, go to

Ribbon: `DM Electrical->Devices->` `Remove Switch from Switch Group`

Pulldown Menu: `DM Electrical->Switches->Remove Switch from Switch Group`

You will be prompted to select the switch to be removed from a switch group.

Select switch to remove from group:

The selected switch will be removed from the switch group it is in and placed in a group by itself. All of the devices in the group are left in the group and will still be connected to any other switches that existed in the group.

Highlight Switch Group

The `Highlight Switch Group` command is used to highlight all of the switches and devices in a switch group. See the **Switch Groups** section for more information.
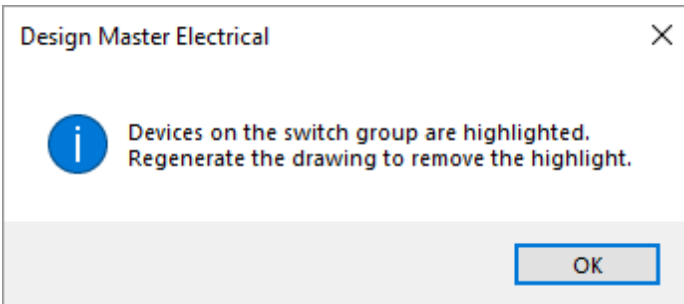
To highlight a switch group, go to

Ribbon: `DM Electrical->Devices->` `Highlight Switch Group`

Pulldown Menu: `DM Electrical->Switches->Highlight Switch Group`

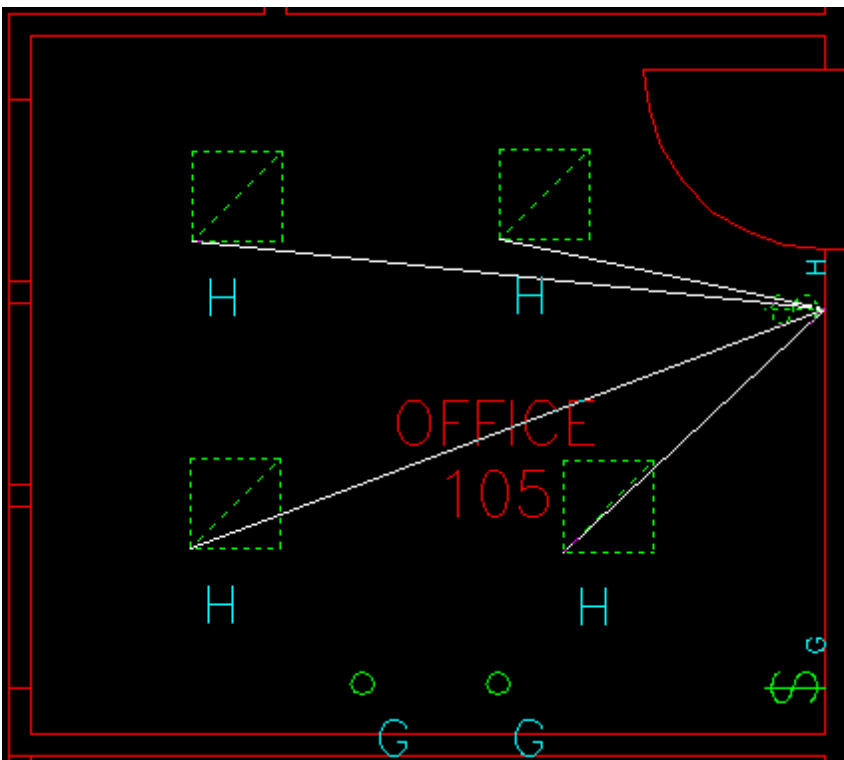You will be prompted to select a switch in the switch group you want to highlight.

Select switch:

A **Design Master Electrical** dialog box will appear as shown below.

Press the **OK** button.

All of the switches and devices in the switch group will be highlighted. Lines will be drawn from the selected switch to all of the other items in the group.

To return the devices to their normal appearance, use the standard CAD REGEN command.

View Switch Load

The `View Switch Load` command is used to calculate the total load and number of devices for a switch or switch group.
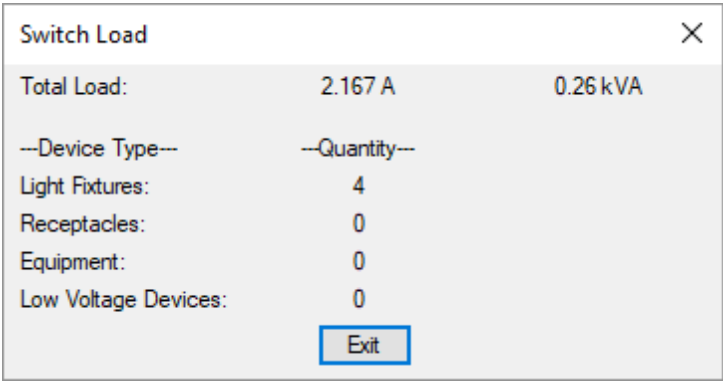
To view the load for a switch or switch group, go to

Ribbon: `DM Electrical->Devices->` Σ `View Switch Load`

Pulldown Menu: `DM Electrical->Switches->View Switch Load`

You will be prompted to select a switch.

Select switch:

Switch Load Dialog Box

The **Switch Load** dialog box will display the total load on the selected switch. If the switch is in a switch group, the total load for the switch group will be displayed.

The number of light fixtures, receptacles, equipment, and low voltage devices connected to the switch will also be displayed. If the switch is in a switch group, the quantities for the switch group will be displayed.
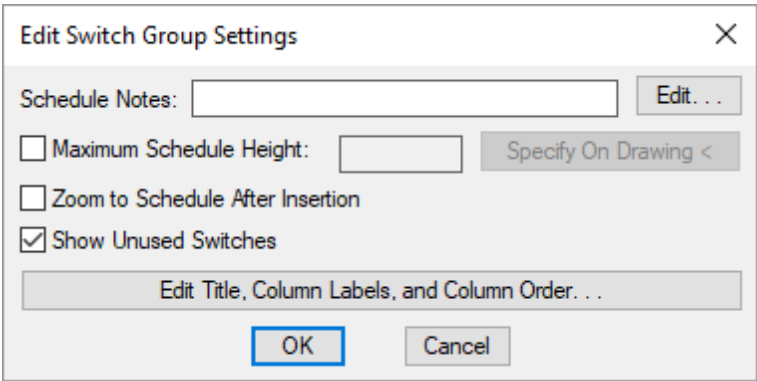
Insert Switch Project Schedule

A switch schedule that lists all of the switches used in the current project can be inserted on the drawing.

To insert this schedule on the drawing, go to

Ribbon: DM Electrical->Devices-> Insert Switch Project Schedule

Pulldown Menu: DM Electrical->Switches->Insert Switch Project Schedule

Insert Switch Project Schedule Dialog Box

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Show Unused Switches: Whether switches in the schedule that have not been inserted on a drawing will be displayed in the schedule.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the switch schedule on the drawing. See the **Edit Switch Project Schedule List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Switch rotation: Sets how the subscript is rotated relative to the switch block.

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Switch Project Schedule

The switch project schedule contains the switch types used in the current project.

To create and modify switch types, go to

Ribbon: `DM Electrical->Devices->` ^{SW}`P` `Switch Project Schedule`

Pulldown Menu: `DM Electrical->Switches->Switch Project Schedule`

Switch Project Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Common Switch Information** section for more information about the values that can be specified for switches using this dialog box.

See the **Import from Another Database** and **Import from Master Schedule** sections for more information about importing switches to the current project from other locations.

Current Group: The active switch group is listed here. All of the switch types that are created and modified will be associated with this group.

Select Group: Press this button to open the **Switch Project Schedule Groups** dialog box. The group selected will become the current group. See the **Schedule Groups** section for more information.

Select Block: Press this button to open the **Select Switch Block** dialog box. This dialog box allows you to choose the block used to represent the switch on the drawing.
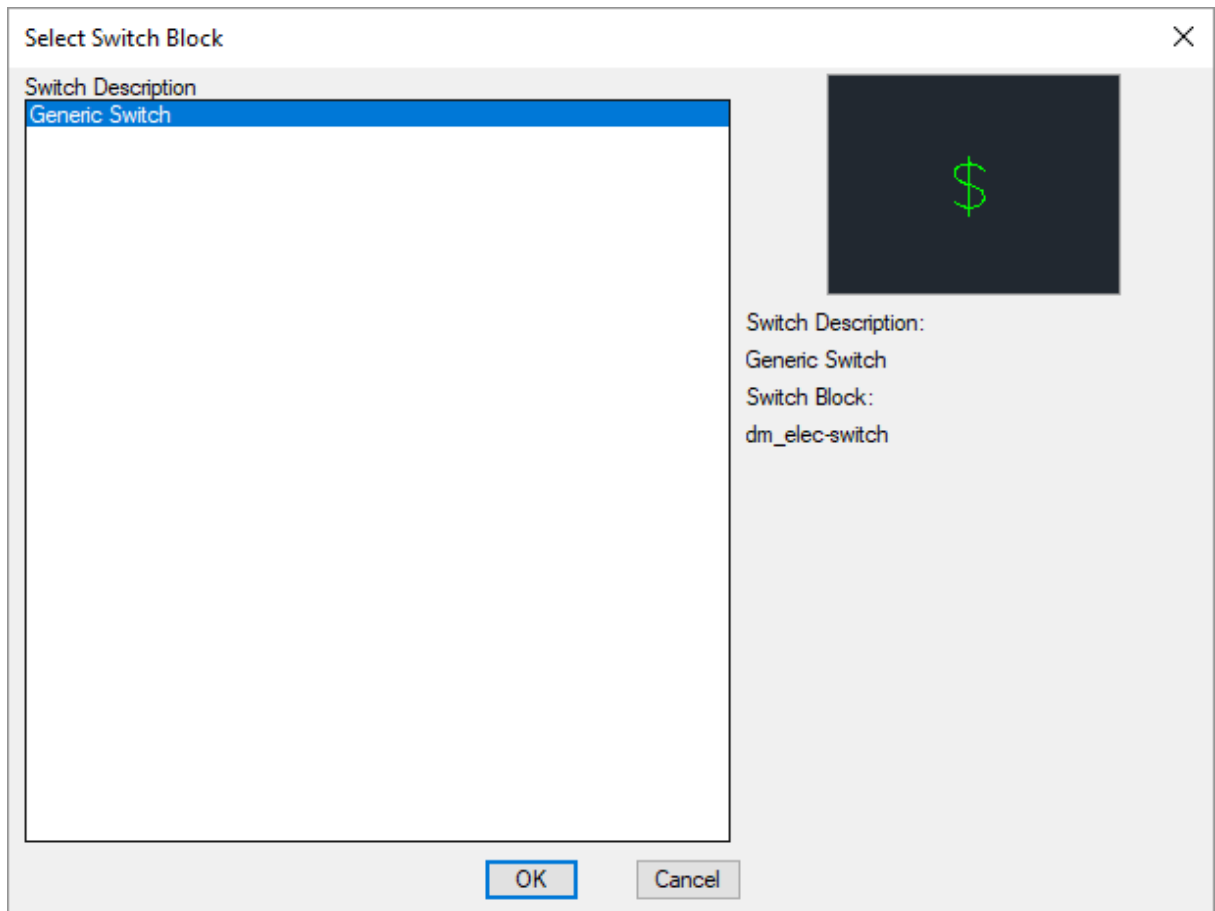
You can add more blocks to this list using the **Switch Block->Edit Project List** command.

Related Options

Switch rotation: Sets how the subscript is rotated relative to the switch block.

Provide neutral wire to switches: Sets whether a neutral is provided to switches, in compliance with *2011 NEC 404.2(C)*.

Switch Master Schedule

The switch master schedule contains the switches that could be used by a company on a project. See the **Master and Standards Databases** section for more information about using master databases.

To create and modify switches in the master schedule, go to

Ribbon: DM Electrical->Devices-> Switch Master Schedule

Pulldown Menu: DM Electrical->Switches->Switch Master Schedule
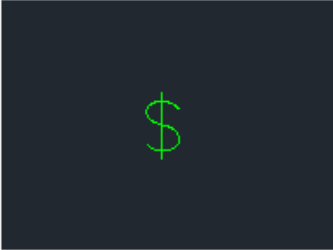
Switch Master Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Common Switch Information** section for more information about the values that can be specified for switches using this dialog box.

See the **Import from Project Schedule** section for more information about importing switches from a project to the master database.

Current Group: The active switch group is listed here. All of the switch types that are created and modified will be associated with this group.

Select Group: Press this button to open the **Switch Master Schedule Groups** dialog box. The group selected will become the current group. See the **Schedule Groups** section for more information.

Select Block: Press this button to open the **Select Switch Block** dialog box. This dialog box allows you to choose the block used to represent the switch on the drawing.
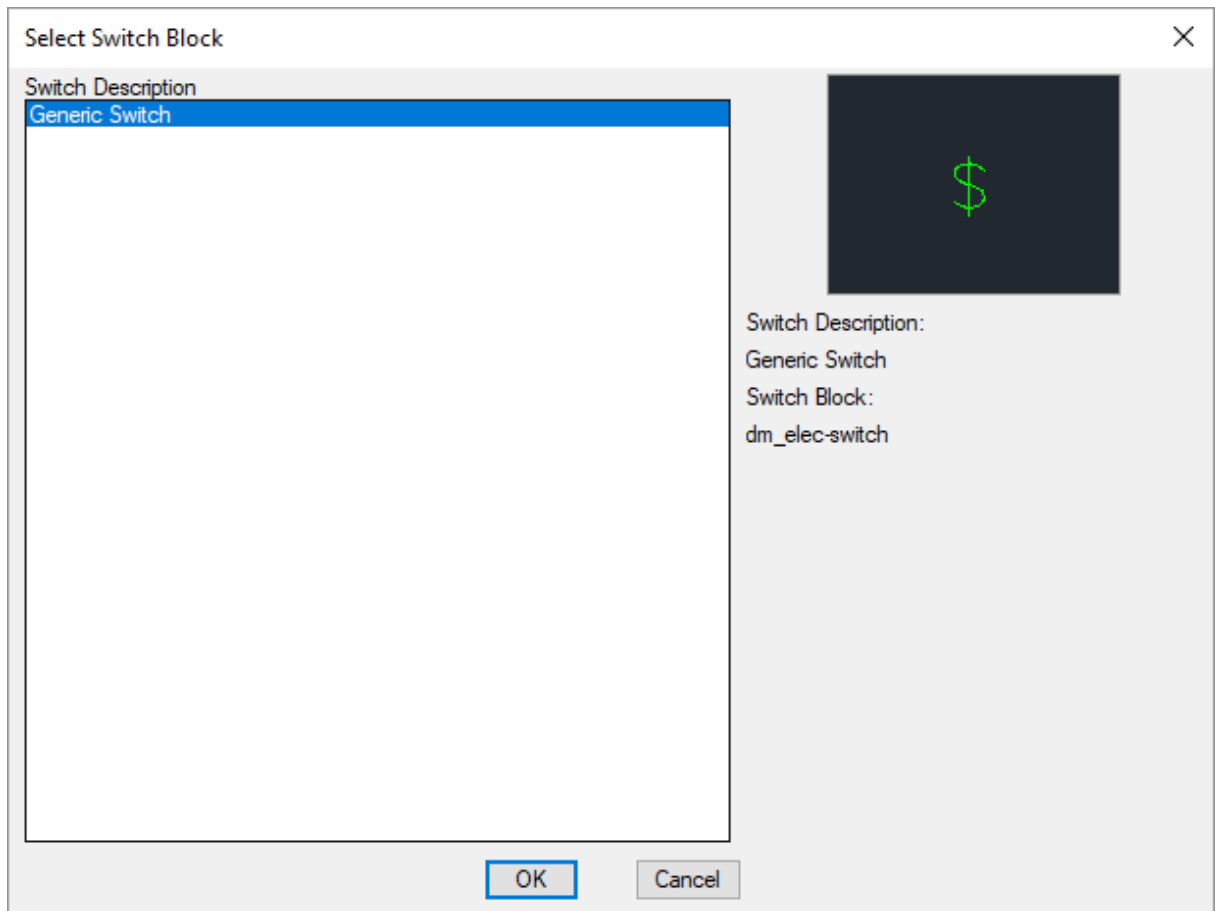
You can add more blocks to this list using the **Switch Block->Edit Master List** command.

Low Voltage Devices

This section describes how to use low voltage devices. Low voltage devices refer to devices you want to insert on your electrical drawing that are not a light fixture, receptacle, equipment, junction box, or switch.

Multiple types of low voltage devices can be created using separate groups. Each group is inserted on separate layers and has a separate schedule.

Low voltage devices can be connected to low voltage panels. Total loads can be calculated on the panels. No other calculations are available for low voltage devices.

Common Low Voltage Device Information

The values that can be specified for a low voltage device are listed below. Some values can be set in the schedule only. Others can have a default set in the schedule that can be changed on individual low voltage devices.

Schedule Only-Settings

This section describes low voltage device values that can only be set in the schedule. These cannot be changed on individual low voltage devices. If you need to insert a low voltage device that has a different setting for one of

these values, you must create a new record in the schedule.

Include in Schedule: Whether the low voltage device type is displayed in the low voltage device schedule that is inserted on the drawing. Most low voltage types will have this box checked. It is commonly unchecked for existing low voltage device types and low voltage devices that are being used to model other types of devices in your project.

Individual Low Voltage Device Settings

Load 1, Load 2: Low voltage devices can have up to two separate loads on them. Each load can be assigned a different load type and connected to a different circuit. See the **Load Information** section for more information.

Subscript Note: This note is inserted with the low voltage device on the drawing. This note can also be displayed in the low voltage device schedule if the *Show Subscript on Symbols* box is checked when inserting the schedule. See the **Insert Low Voltage Device Project Schedule** section for more information.

Drawing Note 1, Drawing Note 2, Drawing Note 3: These notes will appear on the drawing for each low voltage device.

Schedule Note 1, Schedule Note 2, Schedule Note 3: These notes will appear on the low voltage device schedule on the drawing. See the **Schedule Notes** section for more information.

Insert Low Voltage Device

To insert a low voltage device on a drawing, go to

Ribbon: `DM Electrical->Devices->` LVD `Insert Low Voltage Device`

Pulldown Menu: `DM Electrical->Low Voltage Devices->Insert Low Voltage Device`

Insert Low Voltage Device Dialog Box

Group: The active low voltage device group. The items listed in the *Callout* list are based upon the active low voltage device group. See the **Low Voltage Devices Project Schedule Groups** section for more information.

Callout: The low voltage device type to be inserted. The low voltage device callouts in this list are based upon the *Group* selection. The list of low voltage devices available within a specific low voltage device group is defined in the **Low Voltage Device Project Schedule** command.

The default values for the low voltage device are specified based upon the low voltage device type you choose.

Some of these values cannot be changed in the individual low voltage device. To use a different value, you must choose a different low voltage device type.

Other values can be overridden in the individual low voltage device. To do this, check the box next to the value you want you change. You can the specify the value that will apply to the low voltage devices you will insert on the drawing.

Override Schedule Load: Check these boxes to override the load defined in the schedule. If this box is checked, the load fields below it will be enabled and can be changed.

Elevation: The elevation of the low voltage device. See the **Elevation** section for more information.

Layer System: See the **Layer System** section for more information.

256

See the **Common Low Voltage Device Information** section for more information about the other values in this dialog box.

Inserting the Low Voltage Device on the Drawing

To insert a low voltage device on the drawing, press the **OK** button. The settings you specified in the dialog box will be used tor the low voltage devices that you insert. You will be prompted to insert them on the drawing.

Specify insertion point:

You will then be prompted for the rotation angle. If the low voltage device block is not able to be rotated, this prompt will be skipped.

Specify rotation angle <0>:

If there are any notes included with the low voltage device, you will then be prompted to insert the notes on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the notes.

You will then be prompted to insert another low voltage device of the same type. Continue to insert low voltage devices, or press **ENTER** to finish the command.

Related Options

Insert 3D blocks on drawing: Sets whether a 3D block is inserted on the drawing along with the 2D block.

Querying Low Voltage Devices

To modify a low voltage device that is inserted on the drawing, go to

Ribbon: `DM Electrical->Devices->` 📟 `Query Device`

Pulldown Menu: `DM Electrical->Modify Devices->Query Device`

Select a low voltage device on the drawing. The information in the database regarding the selected low voltage device will be displayed in the **Query Low Voltage Device** dialog box.

Query Low Voltage Device Dialog Box

Changes can be made to the low voltage device in the same way as when it was inserted. See the **Insert Low Voltage Device** section for more information.

Edit Multiple Low Voltage Devices

To edit multiple low voltage devices at one time, go to

Ribbon: `DM Electrical->Devices->` `Edit Multiple Low Voltage Devices`

Pulldown Menu: `DM Electrical->Low Voltage Devices->Edit Multiple Low Voltage Devices`

You will be prompted to select the low voltage devices to be edited.

Select low voltage devices to edit:

The **Edit Multiple Low Voltage Devices** dialog box will appear.

Edit Multiple Low Voltage Devices Dialog Box

Changes can be made to the low voltage devices in the same way as when they were inserted. See the **Insert Low Voltage Device** section for more information.

Trace Architect's Low Voltage Devices

The `Trace Architect's Low Voltage Devices` command is used to trace a low voltage device layout from another drawing that does not have Design Master entities.

Before using this command, create low voltage devices using the **Low Voltage Device Project Schedule** command.

This command should be run on the drawing from which you are copying the low voltage device layout. Do not run this command on the drawing you are using for your electrical design.

To trace a low voltage device layout, go to

Ribbon: `DM Electrical->Devices->` LVD TRACE `Trace Architect's Low Voltage Devices`
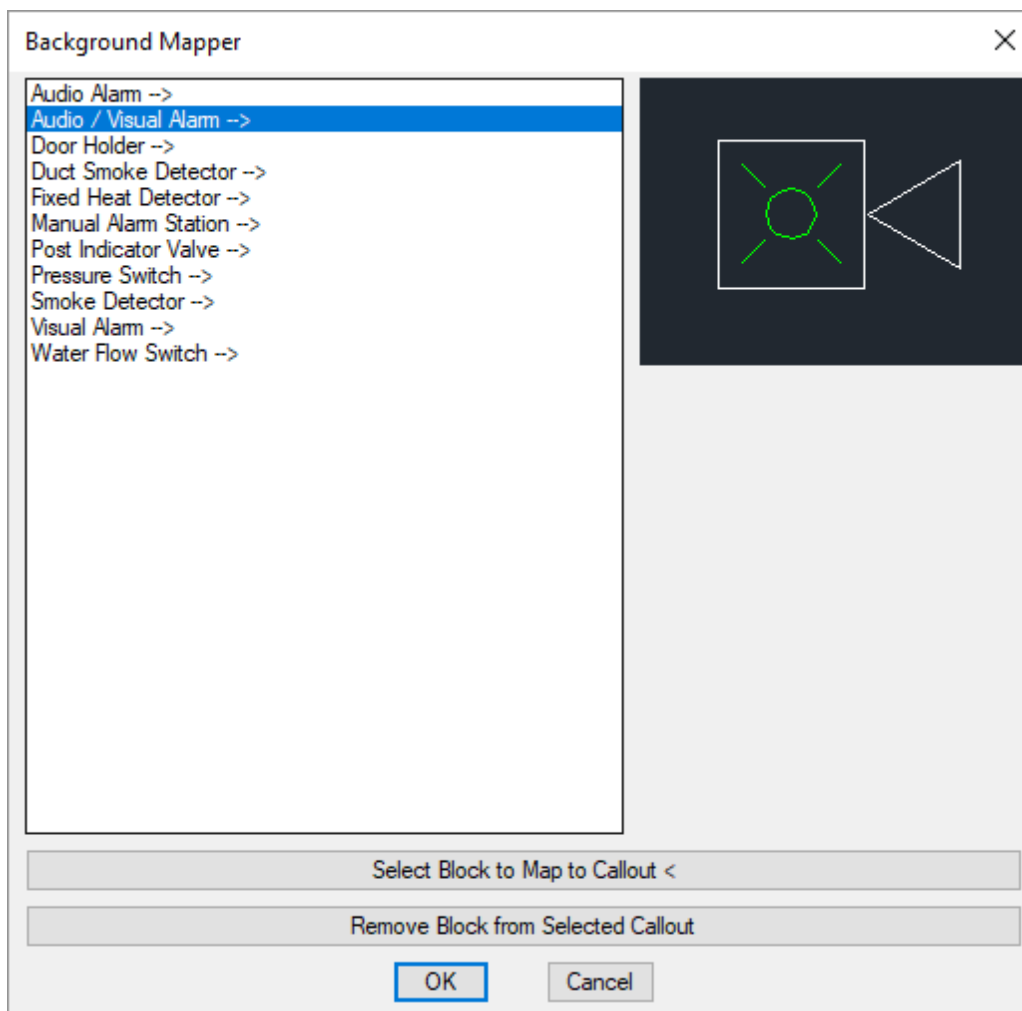
Pulldown Menu: `DM Electrical->Low Voltage Devices->Trace Architect's Low Voltage Devices`

The **Select Low Voltage Device Group** dialog box will appear.
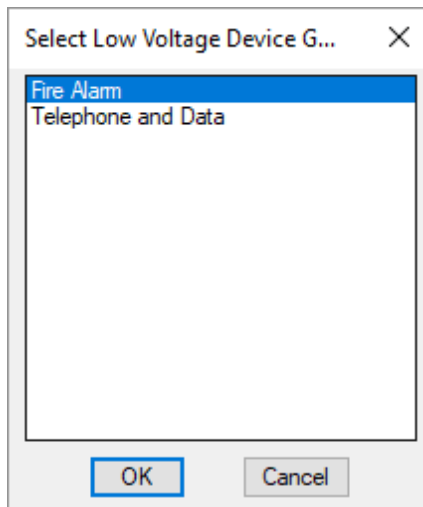
Select a low voltage device group and press the **OK** button.

The **Background Mapper** dialog box will appear.

See the **Trace Architect's Light Fixtures** section for more information about how this command works. The **Background Mapper** dialog box works the same in these two commands.
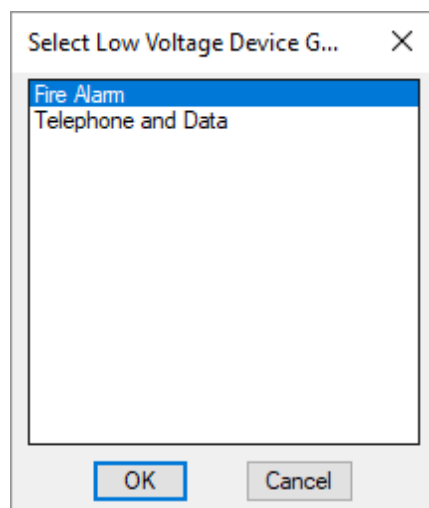
Insert Low Voltage Device Project Schedule

A low voltage device schedule that lists all of the low voltage devices used in the current project can be inserted on the drawing.

To insert this schedule on the drawing, go to

Ribbon: `DM Electrical->Devices->` `Insert Low Voltage Device Project Schedule`

Pulldown Menu: `DM Electrical->Low Voltage Devices->Insert Low Voltage Device Project Schedule`

Select Low Voltage Device Group Dialog Box

Select the low voltage device group to be listed on the schedule and press the **OK** button. The **Edit Low Voltage Device Group Settings** dialog box will appear.

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Specify on Drawing: Press this button to specify the *Maximum Schedule Height* on the drawing. The dialog box will be closed and you will be prompted to specify the height.

Specify maximum schedule height:

The distance that you enter on the drawing will be used as the maximum height. It is simplest to draw a line straight down to where the bottom of the schedule should be located.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule location after it has been inserted or updated. Check this box if you are having difficulty locating your schedule on a busy drawing.

Show Unused Low Voltage Devices: Whether low voltage devices in the schedule that have not been inserted on a drawing will be displayed in the schedule.

Show Subscript on Symbols: Whether subscript notes are displayed in the symbol column of the schedule. The subscript note is defined in the project schedule.

If this box is unchecked, the symbol column will display the low voltage device symbol without the subscript note.

Edit Title, Column Labels, and Column Order: Use this button to modify the layout of the low voltage device schedule on the drawing. See the **Edit Low Voltage Device Project Schedule List** section for more information.

Inserting or Updating the Schedule on the Drawing

To insert or update the schedule, press the **OK** button.

Inserting the Schedule the First Time

If the schedule is not currently inserted on the drawing, you will be prompted for the insertion location of the schedule.

Specify insertion point for schedule:

The location you specify will be used as the top-left corner for the schedule.

Updating the Schedule Already on the Drawing

If the schedule is already inserted on the drawing, it will be updated in its current location.

You must update the schedule on the drawing when changes are made to the project schedule. The schedule will not update automatically.

The schedule will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If additional graphics, such as revision clouds, have been inserted over the schedule graphic, be sure to check their location after updating the schedule.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Low Voltage Device Project Schedule

The low voltage device project schedule contains the low voltage device types that can be used in the current project.

To create and modify low voltage device types, go to

Ribbon: DM Electrical->Devices-> LVD Ⓟ Low Voltage Device Project Schedule

Pulldown Menu: DM Electrical->Low Voltage Devices->Low Voltage Device Project Schedule

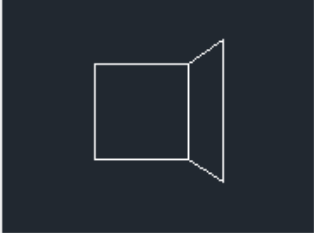## Low Voltage Device Project Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Common Low Voltage Device Information** section for more information about the values that can be specified for low voltage devices using this dialog box.

See the **Import from Another Database** and **Import from Master Schedule** sections for more information about importing low voltage devices to the current project from other locations.

Current Group: The active low voltage device group is listed here. All of the low voltage device types that are created and modified will be associated with this group.

Select Group: Press this button to open the **Low Voltage Device Project Schedule Groups** dialog box. See the **Low Voltage Devices Project Schedule Groups** section below for more information.

Select Block: Press this button to open the **Select Low Voltage Device Block** dialog box. This dialog box allows you to choose the block used to represent the low voltage device on the drawing.
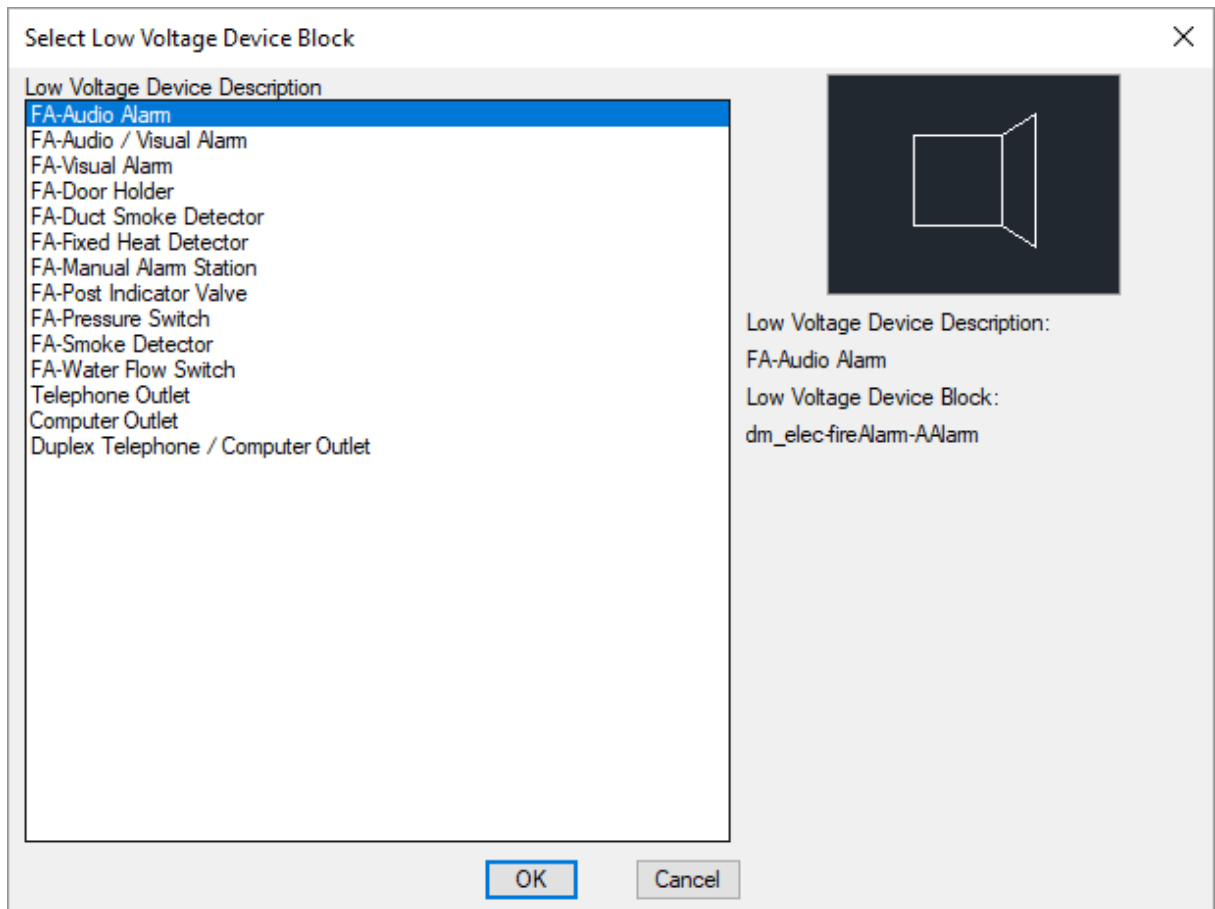
You can add more blocks to this list using the **Low Voltage Device Block->Edit Project List** command.

Low Voltage Device Project Schedule Groups Dialog Box

This section describes how to create and modify low voltage device groups in your project. Multiple groups for different types of low voltage devices can be created in your project.
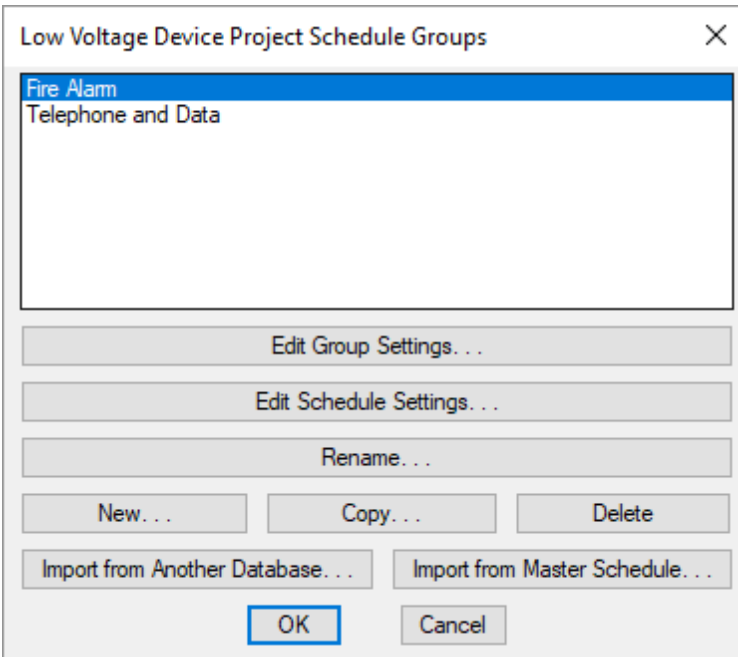
See the **Schedule Groups** section for more information about the **New**, **Copy**, **Delete**, **Rename**, **Delete All Low Voltage Devices in Group**, and **Import Group from Master Schedule** buttons on this dialog box.

Edit Group Settings: Press this button to edit settings for the selected low voltage device group. The **Edit Group Settings** dialog box will appear.
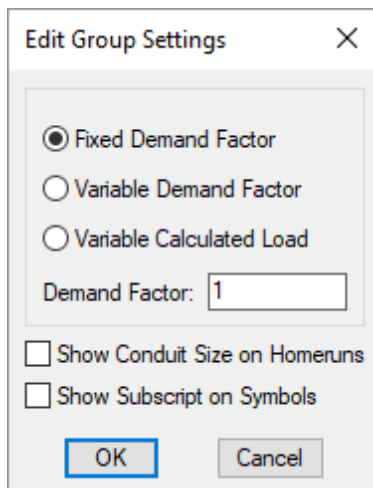
Fixed Demand Factor: The demand factor is set for all devices in the group and cannot be changed.

Variable Demand Factor: The default demand factor is set, but can be changed for each device in the group. The demand factor is input directly as a calculated load.

Variable Calculated Load: The demand factor can be changed for each device in the group. The demand factor is input as a multiplier of the connected load.

Demand Factor: The demand factor for devices in the group. This field is disabled if *Variable Calculated*

267

Load is selected.

Show Conduit Sizes on Homeruns: Whether conduit sizes are shown on homeruns on the drawing for devices in the group.

Show Subscripts on Symbols: Whether subscript notes are displayed in the symbol column of the schedule when it is inserted on the drawing. See the **Insert Low Voltage Device Project Schedule** section for more information.

Edit Schedule Settings: Press this button to open the **Edit Low Voltage Device Group Settings** dialog box.

Low Voltage Device Master Schedule

The low voltage device master schedule contains the low voltage device types that could be used by a company in a project. See the **Master and Standards Databases** section for more information about using master databases.

To create and modify low voltage device types in the master schedule, go to

Ribbon: `DM Electrical->Devices->` LVD **M** `Low Voltage Device Master Schedule`

Pulldown Menu: `DM Electrical->Low Voltage Devices->Low Voltage Device Master Schedule`

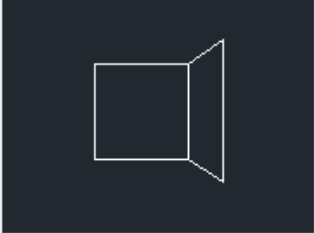## Low Voltage Device Master Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Common Low Voltage Device Information** section for more information about the values that can be specified for low voltage devices using this dialog box.

See the **Common Groups Dialog Box Features** section for more information about creating groups of low voltages devices in the master database.

See the **Import from Project Schedule** section for more information about importing low voltage devices from a project to the master database.

Current Group: The active low voltage device group is listed here. All of the low voltage device types that are created and modified will be associated with this group.

Select Group: Press this button to open the **Low Voltage Device Master Schedule Groups** dialog box. See the **Low Voltage Devices Project Schedule Groups** section for more information about how this dialog box works.

Select Block: Press this button to open the **Select Low Voltage Device Block** dialog box. This dialog box allows you to choose the block used to represent the low voltage device on the drawing.
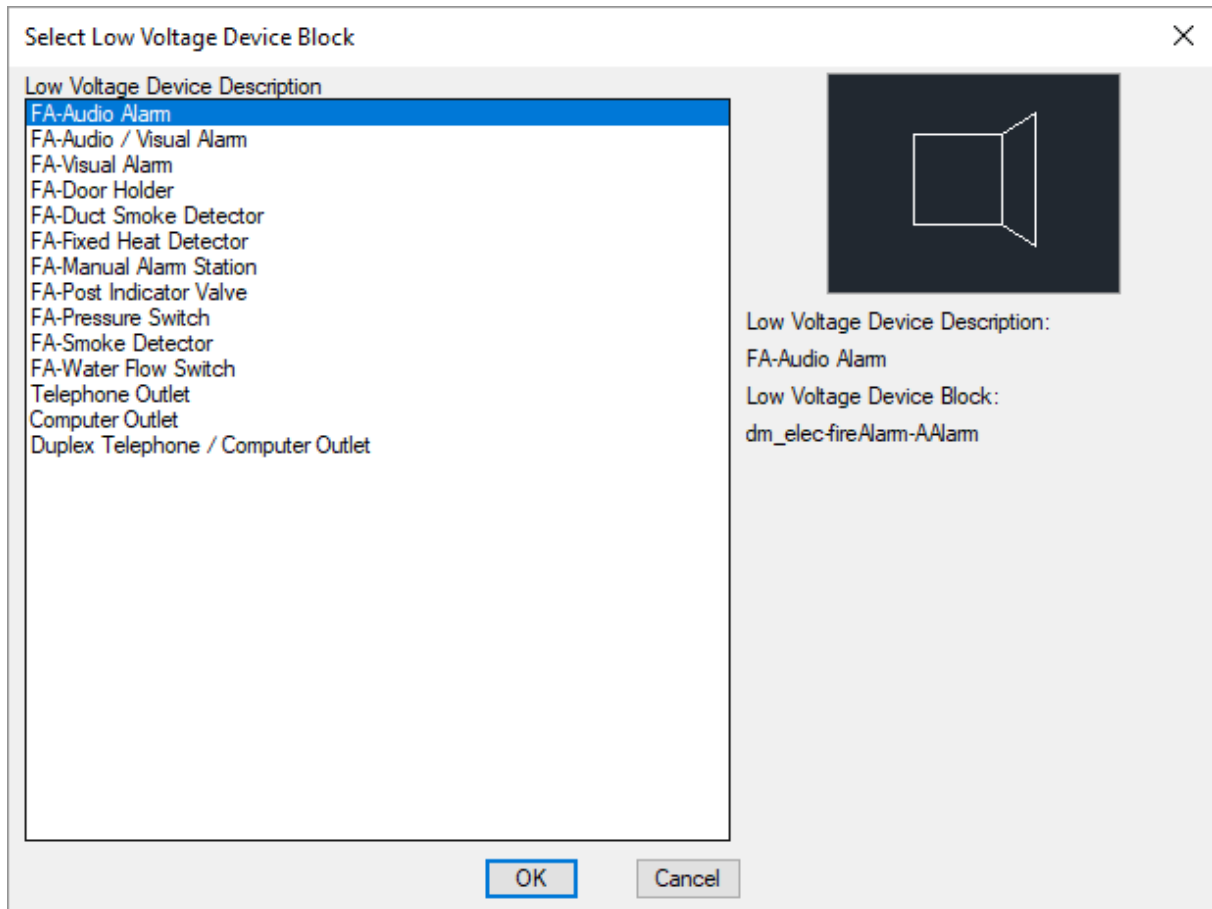
You can add more blocks to this list using the **Low Voltage Device Block->Edit Master List** command.

Conduits

This section describes how to use conduits.

Common Conduit Information

The values that can be specified for a conduit are listed below.

Group: The group to which the conduit is assigned. The group controls the layer on which the conduit is inserted. See the **Conduit Groups** section for more information.

Starting Elevation: The starting elevation of the conduit. The reference point for the starting elevation of the conduit is set by the *Vertical Alignment*. See the **Elevation** section for more information.

Ending Elevation: The ending elevation of the conduit. The reference point for the ending elevation of the conduit is set by the *Vertical Alignment*. See the **Elevation** section for more information.

Vertical Alignment: Specifies the reference point used for the *Starting Elevation* and *Ending Elevation*.
- **Center of Conduit:** The elevations refer to the centerline of the conduit. The conduit will extend both above and below these elevations.
- **Top of Conduit:** The elevations refer to the top of the conduit. The conduit will extend below these elevations.
- **Bottom of Conduit:** The elevations refer to the bottom of the conduit. The conduit will extend above these elevations. If the conduit has hangers, they will extend below this elevation.
- **Bottom of Hanger:** The elevations refer to the bottom of the hanger for the conduit. The hangers and the conduit will extend above these elevations. If there is no hanger, the bottom of the conduit will be at these elevations.

Horizontal Alignment: Specifies the reference point for the conduit construction line.
- **Centered:** The construction line is the centerline of the conduit. The conduit extends to both sides of the construction line. Any transitions in the conduit are concentric.
- **Flat on Left:** The construction line is the left edge of the conduit. The conduit extends to the right of the construction line. Any transitions in the conduit will be eccentric, with the flat edge on the left.
- **Flat on Right:** The construction line is the right edge of the conduit. The conduit extends to the left of the construction line. Any transitions in the conduit will be eccentric, with the flat edge on the right.

Shape: The shape of the conduit.
- **Round:** The conduit will be round. Most conduits will be modeled using this shape.
- **Rectangular:** The conduit will be rectangular. This shape can be used to model bus ducts.

Multiple parallel conduits can be modeled using the rectangular shape. The space that the conduits occupy can be blocked out with a single rectangular conduit. For collision detection, this limited detail is often all that is necessary to verify conflicts in the model.

Diameter: The diameter of the conduit, in inches. This size should be the actual exterior dimensions of the conduit, not the trade size. The exact value is used to draw the conduit for collision detection purposes. This field is enabled if the *Shape* is set to **Round**.

Width: The width of the conduit, in inches. This size should be the actual exterior dimensions of the conduit. The exact value is used to draw the conduit for collision detection purposes. This field is enabled if the *Shape* is set to **Rectangular**.

Depth: The depth of the conduit, in inches. This size should be the actual exterior dimensions of the conduit. The exact value is used to draw the conduit for collision detection purposes. This field is enabled if the *Shape* is set to **Rectangular**.

Hanger Type: The type of hanger. Controls whether hangers and their graphics are included.
- **None:** No hangers are included. The other hanger fields will be disabled.
- **Wrap:** The hanger will wrap around the item. A single support rod will be inserted in the middle of the item.
- **Trapeze:** The hanger will be a support underneath the item with two support rods on either side.

Size: The size of the hanger, in inches. The size is used to control most of the dimensions of the hanger graphic. The support rod diameter will be equal to this size. Other dimensions will be scaled based upon this size.

Hanger Depth: The distance the hanger extends below the item, in inches.

Starting Offset: The distance from the start of the centerline to the location of the first hanger, in inches.

Ending Offset: The distance from the end of the centerline to the location of the last hanger, in inches.

Spacing: The distance between each hanger, in inches.

Left Width: The distance the hanger extends to the left side of the centerline, in inches.

If this value is 0, the hanger will be wide enough to support the item.

If it is greater than 0, the hanger will extend exactly that distance to the left of the centerline. It is possible this distance will be less than the width of the item the hanger is supporting. In that case, you need to increase the width. A width greater than necessary to support the specific item can be used to support a rack of items or to allow for future expansion.

Right Width: The distance the hanger extends to the right side of the centerline, in inches.

If this value is 0, the hanger will be wide enough to support the item.

If it is greater than 0, the hanger will extend exactly that distance to the right of the centerline. It is possible this distance will be less than the width of the item the hanger is supporting. In that case, you need to increase the width. A width greater than necessary to support the specific item can be used to support a rack of items or to allow for future expansion.

Ending Elevation: The elevation where the support rods end. The support rods will be drawn from the item up to this elevation. See the **Elevation** section for more information.

Radius Criteria: How the conduit bend radius will be sized. The options available are dependent upon the shape of the conduit:

    Round Conduit
- **Based Upon Conduit Size (NEC Table 2):** The conduit bend radius will be based upon the size of the conduit. *NEC Table 2* is used to choose the radius of the conduit bend.
- **Custom Size:** Specify the conduit bend radius in the *Custom Radius* field.

    Rectangular Conduit
- **Based Upon <1/2" to 6"> Conduit (NEC Table 2):** The conduit bend radius is based upon the size selected and *NEC Table 2*. A selection for each size in the table is included.
- **Square Elbow:** The conduit bend is a square elbow instead of a bend.
- **Custom Size:** Specify the conduit bend radius in the *Custom Radius* field.

Custom Radius: The radius of the conduit bend, in inches. This field is enabled if **Custom Size** is selected from *Radius Criteria*.

Layer System: See the **Layer System** section for more information.

Insert Conduit

To insert a conduit on a drawing, go to

Ribbon: `DM Electrical->3D-BIM->` ↗ `Insert Conduit`

Pulldown Menu: `DM Electrical->Conduits->Insert Conduit`

You will be prompted to specify the first point of the conduit. The prompt that is displayed depends upon whether you last inserted a forward conduit or a backward conduit.

Specify first point for forward conduit or [Set conduit information/Backward]:
Specify first point for backward conduit or [Set conduit information/Forward]:

Specify first point: Specify the location of the first point for the conduit. You will be prompted to specify the next point of the conduit. The prompt that is displayed depends upon whether you are inserting a forward conduit or a backward conduit.

Specify next point for forward conduit or [Set conduit information/Vertical/Backward/start New run]:
Specify next point for backward conduit or [Set conduit information/Vertical/Forward/start New run]:

Specify next point: Specify the location for the next point of the conduit.

If the conduit is not connected to another conduit, the **Insert Conduit** dialog box will appear, described below.

If the conduit is connected to another conduit, the conduit information will be based upon the connected conduit.

You will be prompted again to specify the next point, allowing you to continue inserting conduits.

Set conduit information: Type **s** to change the settings for the next conduit that will be inserted. The **Insert Conduit** dialog box will appear, described below.

Vertical: Type **v** to insert a vertical conduit at the location of the last point specified.

If the conduit is not connected to another conduit, the **Insert Conduit** dialog box will appear, described below.

If the conduit is connected to another conduit, you will be prompted to enter an elevation for the conduit. Forward conduits will prompt for the *Ending Elevation*. Backward conduits will prompt for the *Starting Elevation*.
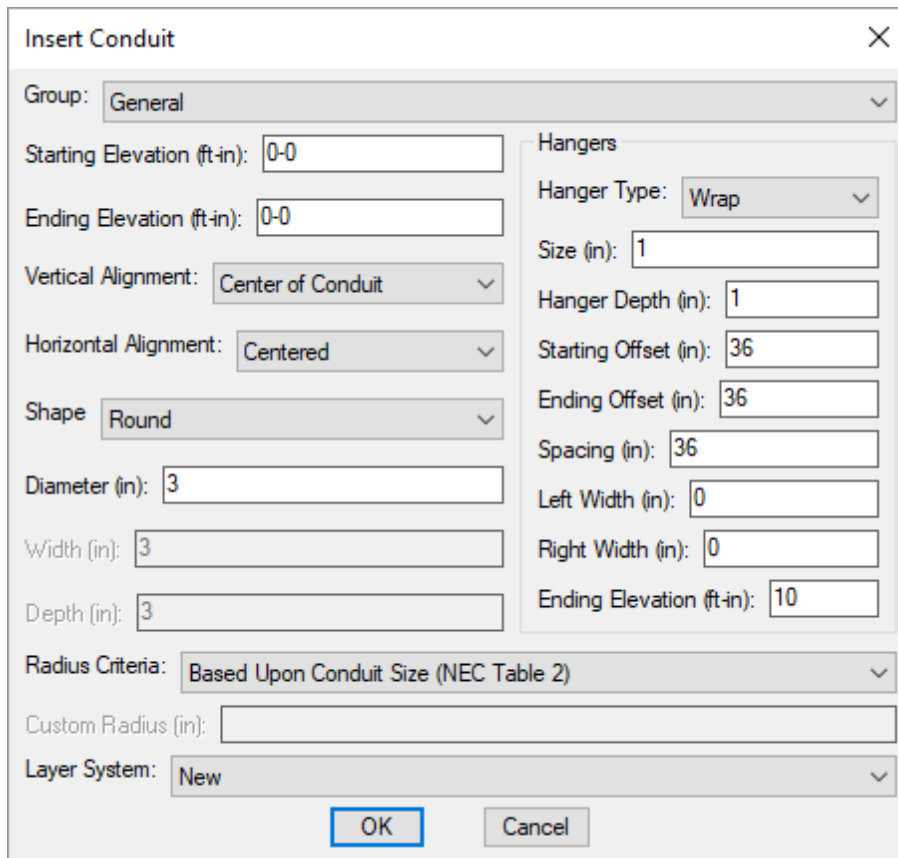
Backward: Type **B** to change the current direction of drawing conduits from forward to backward. The first point of a backward conduit is the location of the arrowhead. The next point of a backward conduit is the location of the tail of the arrow.

Forward: Type **F** to change the current direction of drawing conduits from backward to forward. The first point of a forward conduit is the location of the tail of the arrow. The next point of a forward conduit is the location of the arrowhead.

start New run: Type **N** to finish the current run of conduit. You will be returned to the first prompt of the command and asked to specify the first point for a conduit.

Insert Conduit Dialog Box

The **Insert Conduit** dialog box is displayed when you need to specify information for the conduit that you are about to insert.

See the **Common Conduit Information** section for more information about the values that may be specified for conduits.

Querying Conduits

To modify a conduit that is inserted on the drawing, go to

Ribbon: `DM Electrical->Devices->` `Query Device`

Pulldown Menu: `DM Electrical->Modify Devices->`Query Device

Select a conduit centerline on the drawing. The information in the database regarding the selected conduit will be displayed in the **Edit Conduit** dialog box.

Edit Conduit Dialog Box

Changes can be made to the conduit in the same way as when it was inserted. See the **Insert Conduit** section for more information.

Edit Multiple Conduit

The `Edit Multiple Conduit` command is used to edit multiple individual conduit sections. The selected conduits do not need to be connected to each other. You do not need to select all of the conduits in a run.

To edit multiple conduits, go to

Ribbon: `DM Electrical->3D-BIM->` `Edit Multiple Conduit`

Pulldown Menu: `DM Electrical->Conduits->Edit Multiple Conduit`

You will be prompted to select the conduits to be edited.

Select conduits to edit:

The **Edit Multiple Conduits** dialog box will appear.

Edit Multiple Conduits Dialog Box

Changes can be made to the conduits in the same way as when they were inserted. See the **Insert Conduit** section for more information.

Edit Conduit Branch

The `Edit Conduit Branch` command is used to edit a run of connected conduits. All of the sections of conduit that are connected to the section you select will be modified.

To edit all of the conduits in a connected branch, go to

Ribbon: `DM Electrical->3D-BIM->` ![COND EB] `Edit Conduit Branch`

Pulldown Menu: `DM Electrical->Conduits->Edit Conduit Branch`

You will be prompted to select a single conduit.

Select conduit in branch to edit:

The **Edit Multiple Conduits** dialog box will appear.

Edit Multiple Conduits Dialog Box

Changes can be made to the conduits in the same way as when they were inserted. See the **Insert Conduit** section for more information.

Erase Conduit Run

The `Erase Conduit Run` command is used to erase a conduit and all of the pieces of conduit connected to it. Use this command when you need to easily erase a whole run of conduit from the drawing.

To erase a run of conduit, go to

Ribbon: `DM Electrical->3D-BIM->` `Erase Conduit Run`

Pulldown Menu: `DM Electrical->Conduits->Erase Conduit Run`

You will be prompted to identify which conduit run is to be erased.

Select conduit run to erase:

The selected conduit and all of the conduits connected to it will be erased.

Offset Conduit

The `Offset Conduit` command is used to create parallel runs of conduit. After inserting a single conduit run between two panels, use this command to create the additional conduit runs. The parallel runs can be either

vertically or horizontally offset to the first conduit you inserted.

To offset a conduit run, go to

Ribbon: `DM Electrical->3D-BIM->` `Offset Conduit`

Pulldown Menu: `DM Electrical->Conduits->Offset Conduit`

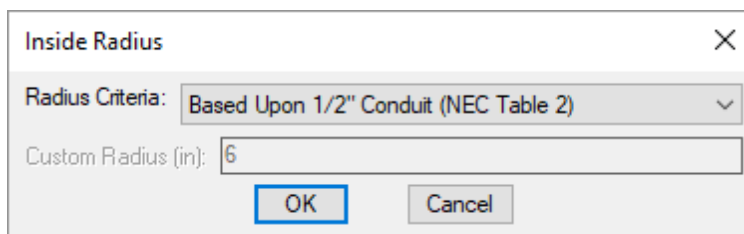You will be prompted to enter the distance of the offset.

Specify offset distance:

You will then be prompted to identify which conduit is to be offset.

Select conduit to offset:

All of the conduits connected to the one you selected will be duplicated and offset.

The **Inside Radius** dialog box will appear.

See the **Adjust Conduit Radius** section for more information about setting the inside radius.

Select the *Radius Criteria* and press the **OK** button. You will be prompted to specify on which side the offset conduit is to be located.

Specify point on side to offset or [Up/Down/Multiple]:

> *Specify point:* The conduit run will be offset to the side you select on the drawing.
>
> *Up:* The conduit will be offset up. Use this setting to create vertical parallel runs of conduit.
>
> *Down:* The conduit will be offset down. Use this setting to create vertical parallel runs of conduit.
>
> *Multiple:* You will be able to offset the conduit multiple times. The **Inside Radius** will be used to set the radius value on all of the conduits that are offset.
>
> After selecting *Multiple*, you will then be prompted to specify the directions to offset the conduit run, using the same options as before.
>
> *Specify point on side to offset or [Up/Down/<select Next object>]:*
>
> > *select Next object:* Finishing offsetting the selected conduit and select a new conduit to offset.

Adjust Conduit Radius

The `Adjust Conduit Radius` command is used to adjust the bend radii of one or more conduits. This can be used on offset conduits to make the bends concentric rather than staggered.
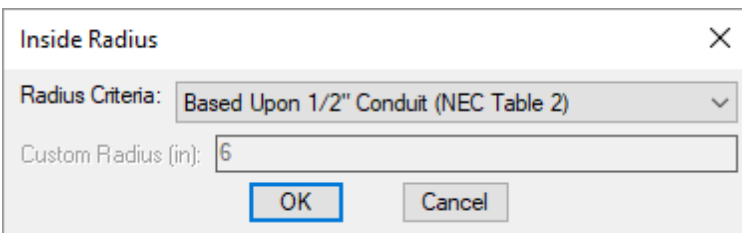
To adjust the bend radius of a conduit, go to

Ribbon: `DM Electrical->3D-BIM->` Adjust Conduit Radius

Pulldown Menu: `DM Electrical->Conduit->Adjust Conduit Radius`

You will be prompted to identify which conduits have radii to be adjusted.

Select conduits to adjust radius on:

The **Inside Radius** dialog box will appear.

The radius specified in this dialog box will be used for the inside radius of bends in the conduit. With multiple parallel conduits, this value is specified for the conduit that is on the inside with the smallest radius. The bends in the other conduits will be adjusted so that they have the same center point as the inside bend.

Select the desired radius criteria and press the **OK** button.

The conduit bend radii will be adjusted.

Calculate Conduit Information

To calculate the total length, number of bends, and total angle of bends in a conduit run, go to

Ribbon: `DM Electrical->3D-BIM->` Calculate Conduit Information

Pulldown Menu: `DM Electrical->Conduits->Calculate Conduit Information`

You will be prompted to specify the conduit run you want calculated.

Select conduit to calculate information about:

A **Design Master Electrical** dialog box will appear with the information, as shown below. The information is also printed on the command line.
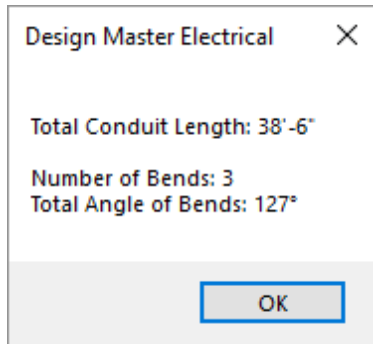
Press the **OK** button to close the dialog box.

Cable Tray

This section describes how to use cable trays.

Common Cable Tray Information

The values that can be specified for cable trays are listed below.

Group: The group to which the cable tray is assigned. The group controls the layer on which the cable tray is inserted. See the **Cable Tray Groups** section for more information.

Starting Elevation: The starting elevation of the cable tray. The reference point for the starting elevation of the cable tray is set by the *Vertical Alignment*. See the **Elevation** section for more information.

Ending Elevation: The ending elevation of the cable tray. The reference point for the ending elevation of the cable tray is set by the *Vertical Alignment*. See the **Elevation** section for more information.

Vertical Alignment: Specifies the reference point used for the *Starting Elevation* and *Ending Elevation*.
- **Center of Cable Tray:** The elevations refer to the centerline of the cable tray. The cable tray will extend both above and below these elevations.
- **Top of Cable Tray:** The elevations refer to the top of the cable tray. The cable tray will extend below these elevations.
- **Bottom of Cable Tray:** The elevations refer to the bottom of the cable tray. The cable tray will extend above these elevations. If the cable tray has hangers, they will extend below this elevation.
- **Bottom of Hanger:** The elevations refer to the bottom of the hanger for the cable tray. The hangers and the cable tray will extend above these elevations. If there is no hanger, the bottom of the cable tray will be at these elevations.

Horizontal Alignment: Specifies the reference point for the cable tray construction line.
- **Centered:** The construction line is the centerline of the cable tray. The cable tray extends to both sides of the construction line. Any transitions in the cable tray are concentric.
- **Flat on Left:** The construction line is the left edge of the cable tray. The cable tray extends to the right of the construction line. Any transitions in the cable tray will be eccentric, with the flat edge on the left.
- **Flat on Right:** The construction line is the right edge of the cable tray. The cable tray extends to the left of the construction line. Any transitions in the cable tray will be eccentric, with the flat edge on the right.

Width: The width of the cable tray, in inches. This size should be the actual exterior dimensions of the cable tray. The exact value is used to draw the cable tray for collision detection purposes.

Depth: The depth of the cable tray, in inches. This size should be the actual exterior dimensions of the cable tray. The exact value is used to draw the cable tray for collision detection purposes.

Radius: The inside radius of the fittings in the cable tray.

Elbow Type: The graphic that is used for elbows in the cable tray. Tees and crosses look the same for both selections.

- **Square-Radius:** The inside edge of the elbow is curved. The outside edge of the elbow is square as shown below.
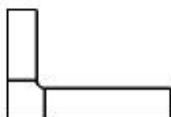
- **Radius:** The edges of the elbow are curved.

Hanger Type: The type of hanger. Controls whether hangers and their graphics are included.

- **None:** No hangers are included. The other hanger fields will be disabled.
- **Wrap:** The hanger will wrap around the item. A single support rod will be inserted in the middle of the item.
- **Trapeze:** The hanger will be a support underneath the item with two support rods on either side.

Size: The size of the hanger, in inches. The size is used to control most of the dimensions of the hanger graphic. The support rod diameter will be equal to this size. Other dimensions will be scaled based upon this size.

Hanger Depth: The distance the hanger extends below the item, in inches.

Starting Offset: The distance from the start of the centerline to the location of the first hanger, in inches.

Ending Offset: The distance from the end of the centerline to the location of the last hanger, in inches.

Spacing: The distance between each hanger, in inches.

Left Width: The distance the hanger extends to the left side of the centerline, in inches.

If this value is 0, the hanger will be wide enough to support the item.

If it is greater than 0, the hanger will extend exactly that distance to the left of the centerline. It is possible this distance will be less than the width of the item the hanger is supporting. In that case, you need to increase the width. A width greater than necessary to support the specific item can be used to support a rack of items or to allow for future expansion.

Right Width: The distance the hanger extends to the right side of the centerline, in inches.

If this value is 0, the hanger will be wide enough to support the item.

If it is greater than 0, the hanger will extend exactly that distance to the right of the centerline. It is possible this distance will be less than the width of the item the hanger is supporting. In that case, you need to increase the width. A width greater than necessary to support the specific item can be used to support a rack of items or to allow for future expansion.

Ending Elevation: The elevation where the support rods end. The support rods will be drawn from the item up to this elevation. See the **Elevation** section for more information.

Layer System: See the **Layer System** section for more information.

Insert Cable Tray

To insert a cable tray on a drawing, go to

Ribbon: `DM Electrical->3D-BIM->` ↗ `Insert Cable Tray`

Pulldown Menu: `DM Electrical->Cable Tray->Insert Cable Tray`

You will be prompted to specify the first point of the cable tray. The prompt that is displayed depends upon whether you last inserted a forward cable tray or a backward cable tray.

Specify first point for forward cable tray or [Set conduit information/Backward]:
Specify first point for backward cable tray or [Set conduit information/Forward]:

> **Specify first point:** Specify the location of the first point for the cable tray. You will be prompted to specify the next point of the cable tray. The prompt that is displayed depends upon whether you are inserting a forward cable tray or a backward cable tray.
>
> *Specify next point for forward cable tray or [Set cable tray information/Vertical/Backward/start New run]:*
> *Specify next point for backward cable tray or [Set cable tray information/Vertical/Forward/start New run]:*
>
>> **Specify next point:** Specify the location for the next point of the cable tray.
>>
>> If the cable tray is not connected to another cable tray, the **Insert Cable Tray** dialog box will appear, described below.
>>
>> If the cable tray is connected to another cable tray, the cable tray information will be based upon the connected cable tray.
>>
>> You will be prompted again to specify the next point, allowing you to continue inserting cable trays.
>>
>> **Set cable tray information:** Type **s** to change the settings for the next cable tray that will be inserted. The **Insert Cable Tray** dialog box will appear, described below.
>>
>> **Vertical:** Type **v** to insert a vertical cable tray at the location of the last point specified.
>>
>> If the cable tray is not connected to another cable tray, the **Insert Cable Tray** dialog box will appear, described below.
>>
>> If the cable tray is connected to another cable tray, you will be prompted to enter an elevation for the cable tray. Forward cable trays will prompt for the *Ending Elevation*. Backward cable trays will prompt for the *Starting Elevation*.
>>
>> **Backward:** Type **b** to change the current direction of drawing cable trays from forward to backward. The first point of a backward cable tray is the location of the arrowhead. The next point of a backward

cable tray is the location of the tail of the arrow.

Forward: Type **F** to change the current direction of drawing cable trays from backward to forward. The first point of a forward cable tray is the location of the tail of the arrow. The next point of a forward cable tray is the location of the arrowhead.

start New run: Type **N** to finish the current run of cable tray. You will be returned to the first prompt of the command and asked to specify the first point for a cable tray.

Insert Cable Tray Dialog Box

The **Insert Cable Tray** dialog box is displayed when you need to specify information for the cable tray that you are about to insert.

See the **Common Cable Tray Information** section for more information about the values that may be specified for cable trays.

Querying Cable Trays

To modify a cable tray that is inserted on the drawing, go to

Ribbon: `DM Electrical->Devices->` `Query Device`

Pulldown Menu: `DM Electrical->Modify Devices->Query Device`

Select a cable tray centerline on the drawing. The information in the database regarding the selected cable tray will be displayed in the **Edit Cable Tray** dialog box.

Edit Cable Tray Dialog Box

Changes can be made to the cable tray in the same way as when it was inserted. See the **Insert Cable Tray** section for more information.

Edit Multiple Cable Tray

The `Edit Multiple Cable Trays` command is used to edit multiple individual cable trays sections. The selected cable trays do not need to be connected to each other. You do not need to select all of the cable trays in a branch.

To edit multiple cable trays, go to

Ribbon: `DM Electrical->3D-BIM->` TRAY EM `Edit Multiple Cable Trays`
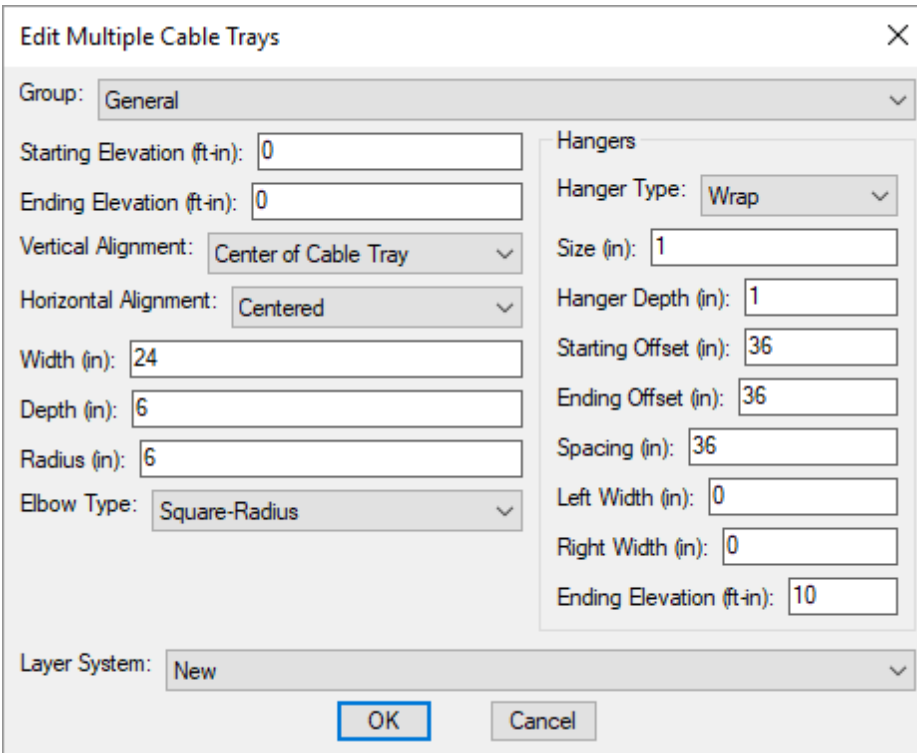
Pulldown Menu: `DM Electrical->Cable Tray->Edit Multiple Cable Trays`

You will be prompted to select the cable trays to be edited.

Select cable trays to edit:

The **Edit Multiple Cable Trays** dialog box will appear.

Edit Multiple Cable Trays Dialog Box

Changes can be made to the cable trays in the same way as when they were inserted. See the **Insert Cable Tray** section for more information.

Edit Cable Tray Branch

The `Edit Cable Tray Branch` command is used to edit a run of connected cable trays. All of the sections of cable tray that are connected to the section you select will be modified.

To edit all of the cable trays in a connected branch, go to

Ribbon: `DM Electrical->3D-BIM->` `Edit Cable Tray Branch`

Pulldown Menu: `DM Electrical->Cable Tray->Edit Cable Tray Branch`

You will be prompted to select a single cable tray.

Select cable tray in branch to edit:

The **Edit Multiple Cable Trays** dialog box will appear.

Edit Multiple Cable Trays Dialog Box

Changes can be made to the cable trays in the same way as when they were inserted. See the **Insert Cable Tray** section for more information.

3D-BIM

This section describes the commands available to perform 3D-BIM modeling and analysis.

Training Videos

- **3D Modeling**

Draw 2D Conduit or Cable Tray

To draw conduits or cable trays in 2D, go to

Ribbon: `DM Electrical->3D-BIM->` 2D `Draw 2D Conduit or Cable Tray`

Pulldown Menu: `DM Electrical->3D-BIM->Draw 2D Conduit or Cable Tray`

You will be prompted to identify which conduit or cable tray is to be drawn in 2D.

Select conduit or cable tray in system to draw in 2D:

The entire run of the selected conduit or cable tray will be drawn in 2D.

Erase 2D Conduit or Cable Tray

To erase the 2D graphics for a conduit or cable tray, go to

Ribbon: `DM Electrical->3D-BIM->` `Erase 2D Conduit or Cable Tray`

Pulldown Menu: `DM Electrical->3D-BIM->Erase 2D Conduit or Cable Tray`

You will be prompted to identify from which conduit or cable tray the 2D graphics are to be erased.

Select conduit or cable tray in system to erase 2D from:

The 2D graphics for the entire run of the selected conduit or cable tray will be erased.

Draw 3D Conduit or Cable Tray

To draw conduits or cable trays in 3D, go to

Ribbon: `DM Electrical->3D-BIM->` `Draw 3D Conduit or Cable Tray`

Pulldown Menu: `DM Electrical->3D-BIM->Draw 3D Conduit or Cable Tray`

You will be prompted to select which conduit or cable tray is to be drawn in 3D.

Select conduit or cable tray in system to draw in 3D:

The entire run of the selected conduit or cable tray will be drawn in 3D.

Erase 3D Conduit or Cable Tray

To erase the 3D graphics for a conduit or cable tray, go to

Ribbon: `DM Electrical->3D-BIM->` `Erase 3D Conduit or Cable Tray`

Pulldown Menu: `DM Electrical->3D-BIM->Erase 3D Conduit or Cable Tray`

You will be prompted to identify from which conduit or cable tray the 3D graphics are to be erased.

Select conduit or cable tray in system to erase 3D from:

The 3D graphics for the entire run of the selected conduit or cable tray will be erased.

Fillet Conduit or Cable Tray

The `Fillet Conduit or Cable Tray` command is used to join two conduits or cable trays together. The commands works similar to the standard CAD **FILLET** command.

To fillet two conduits or cable trays, go to

Ribbon: `DM Electrical->3D-BIM->` `Fillet Conduit or Cable Tray`

Pulldown Menu: `DM Electrical->3D-BIM->Fillet Conduit or Cable Tray`

You will be prompted to select the first conduit or cable tray to fillet.

Select first conduit or cable tray to fillet:

You will then be prompted to select the second conduit or cable tray.

Select second <conduit/cable tray>:

The two conduits or cable trays will be extended to each other and connected. If the conduits or cable trays are at different elevations, a vertical piece will be inserted at the elbow to complete the connection.

Break Conduit or Cable Tray

The `Break Conduit or Cable Tray` command will break a single conduit or cable tray centerline into two pieces.

To break a conduit or cable tray, go to

Ribbon: `DM Electrical->3D-BIM->` `Break Conduit or Cable Tray`

Pulldown Menu: `DM Electrical->3D-BIM->Break Conduit or Cable Tray`

You will be prompted to specify the location of the break on the conduit or cable tray centerline.

Select conduit or cable tray at break point:

The centerline will be broken into two pieces. One piece will go from the original starting point to the break point. The other piece will go from the break point to the original ending point.

Remove Conduit or Cable Tray Node

The `Remove Conduit or Cable Tray Node` command is used to convert two conduit or cable tray centerlines into a single centerline. The two centerlines you select must be connected. The new centerline that is created will run from the starting point of the first centerline to the ending point of the second centerline. This command

cannot be used on intersections with more than two centerlines.

To remove a conduit or cable tray node, go to

Ribbon: `DM Electrical->3D-BIM->` 🪁 `Remove Conduit or Cable Tray Node`

Pulldown Menu: `DM Electrical->3D-BIM->Remove Conduit or Cable Tray Node`

You will be prompted to specify a point on the conduit or cable tray near the node to be removed.

Select conduit or cable tray near node to remove:

The two centerlines will be removed and replaced with a single centerline.

Straighten Next Conduit or Cable Tray

The `Straighten Next Conduit or Cable Tray` command is used to make conduit or cable tray centerlines straight relative to other connected centerlines. The centerline you select will be treated as straight by the command. All of the centerlines that the selected centerline points to will be made parallel, perpendicular, or at a 45-degree angle to the selected centerline, based upon their previous angles.

Any centerlines connected to the centerlines that are straightened will have their starting points moved to stay connected. Their ending points will not be moved, which can result in centerlines that are no longer straight. Continue using the `Straighten Next Conduit or Cable Tray` command until all of the centerlines are straight.

To straighten conduits or cable trays, go to

Ribbon: `DM Electrical->3D-BIM->` 🔲 `Straighten Next Conduit or Cable Tray`

Pulldown Menu: `DM Electrical->3D-BIM->Straighten Next Conduit or Cable Tray`

You will be prompted to select the conduit or cable tray centerline that is straight.

Select the conduit or cable tray that is currently straight:

The centerlines that the selected centerline points to will be made straight.

Straighten Previous Conduit or Cable Tray

The `Straighten Previous Conduit or Cable Tray` command is used to make conduit or cable tray centerlines straight relative to other connected centerlines. The centerline you select will be treated as straight by the command. All of the centerlines that point to the selected centerline will be made parallel, perpendicular, or at a 45-degree angle to the selected centerline, based upon their previous angles.

Any centerlines connected to the centerlines that are straightened will have their ending points moved to stay connected. Their starting points will not be moved, which can result in centerlines that are no longer straight. Continue using the `Straighten Previous Conduit or Cable Tray` command until all of the centerlines are straight.

To straighten conduits or cable trays, go to

Ribbon: `DM Electrical->3D-BIM->` `Straighten Previous Conduit or Cable Tray`

Pulldown Menu: `DM Electrical->3D-BIM->Straighten Previous Conduit or Cable Tray`

You will be prompted to select the conduit or cable tray centerline that is straight.

Select the conduit or cable tray that is currently straight:

The centerlines that point to the selected centerline will be made straight.

Insert Hanger

To insert a hanger on a conduit or cable tray, go to

Ribbon: `DM Electrical->3D-BIM->` `Insert Hanger`

Pulldown Menu: `DM Electrical->3D-BIM->Insert Hanger`

You will be prompted to select a point on the conduit or cable tray where a hanger is to be inserted.

Select conduit or cable tray at location to insert hanger:

A hanger will be inserted at the specified point. The location of other hangers will be adjusted based upon the newly inserted hanger.

Move Hanger

To move a hanger to a new location on a conduit or cable tray, go to

Ribbon: `DM Electrical->3D-BIM->` `Move Hanger`

Pulldown Menu: `DM Electrical->3D-BIM->Move Hanger`

You will be prompted to identify which hanger is to be moved.

Select hanger on conduit or cable tray to move:

You will then be prompted to identify where the hanger is to be placed.

Select the new location

The hanger will be moved to the new location. The location of other hangers will be adjusted based upon the new location of the hanger.

Remove Hanger

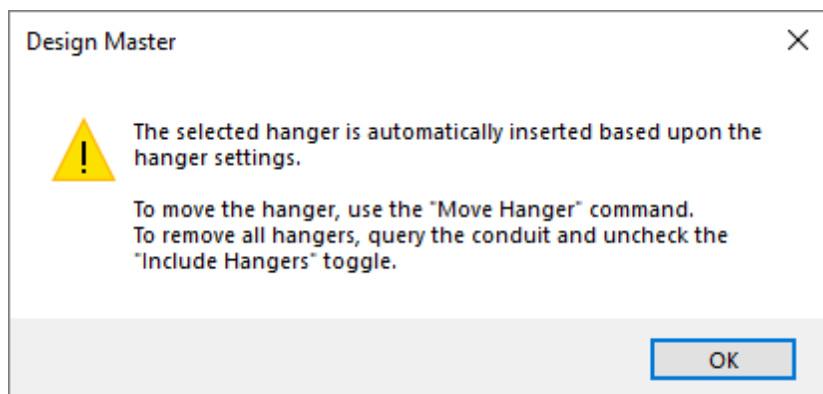To remove a hanger from a conduit or cable tray, go to

Ribbon: `DM Electrical->3D-BIM->` `Remove Hanger`

Pulldown Menu: `DM Electrical->3D-BIM->Remove Hanger`

You will be prompted to identify which hanger is to be removed.

Select hanger on conduit or cable tray to remove:

Not all hangers can be removed. If the selected hanger is necessary based upon the spacing requirements set for the conduit or cable tray, you will not be able to remove it and the dialog box shown below will appear.

Insert Top Elevation Label

The `Insert Top Elevation Label` command is used to insert an elevation label on a conduit or cable tray that displays the elevation of the top of the conduit or cable tray.

To insert a top elevation label on a conduit or cable tray, go to

Ribbon: `DM Electrical->3D-BIM->` ELEV `Insert Top Elevation Label`

Pulldown Menu: `DM Electrical->3D-BIM->Insert Top Elevation Label`

You will be prompted to identify which section of conduit or cable tray is to be labeled.

Select conduit or cable tray to add elevation label to:

You will then be prompted to specify the location of the top elevation label on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the label.

Related Options

Leader graphic: Sets the block used for elevation label leaders.

Leader location: Sets whether leaders snap to the edge or center of conduits and cable trays.

Top block: Sets the block used for the top elevation label.

Top label: Sets how the top elevation is displayed on the label.

Insert Bottom Elevation Label

The `Insert Bottom Elevation Label` command is used to insert an elevation label on a conduit or cable tray that displays the elevation of the bottom of the conduit or cable tray.

To insert a bottom elevation label on a conduit or cable tray, go to

Ribbon: `DM Electrical->3D-BIM->` ELEV `Insert Bottom Elevation Label`

Pulldown Menu: `DM Electrical->3D-BIM->Insert Bottom Elevation Label`

You will be prompted to identify which section of conduit or cable tray is to be labeled.

Select conduit or cable tray to add elevation label to:

You will then be prompted to specify the location of the bottom elevation label on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the label.

Related Options

Leader graphic: Sets the block used for elevation label leaders.

Leader location: Sets whether leaders snap to the edge or center of conduits and cable trays.

Bottom block: Sets the block used for the bottom elevation label.

Bottom label: Sets how the bottom elevation is displayed on the label.

Insert Top and Bottom Elevation Label

The `Insert Top and Bottom Elevation Label` command is used to insert an elevation label on a conduit or cable tray that displays the elevation of the top and the bottom of the conduit or cable tray.

To insert a top and bottom elevation label on a conduit or cable tray, go to

Ribbon: `DM Electrical->3D-BIM->` ELEV `Insert Top and Bottom Elevation Label`

Pulldown Menu: `DM Electrical->3D-BIM->Insert Top and Bottom Elevation Label`

You will be prompted to identify which section of conduit or cable tray is to be labeled.

Select conduit or cable tray to add elevation label to:

You will then be prompted to specify the location of the top and bottom elevation label on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the label.

Related Options

Leader graphic: Sets the block used for elevation label leaders.

Insert Centerline Elevation Label

The `Insert Centerline Label` command is used to insert an elevation label on a conduit or cable tray that displays the elevation of the center of the conduit or cable tray.

To insert a centerline elevation label on a conduit or cable tray, go to

Ribbon: `DM Electrical->3D-BIM->` `Insert Centerline Elevation Label`

Pulldown Menu: `DM Electrical->3D-BIM->Insert Centerline Elevation Label`

You will be prompted to identify which section of conduit or cable tray is to be labeled.

Select conduit or cable tray to add elevation label to:

You will then be prompted to specify the location of the centerline elevation label on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about inserting the label.

Related Options

Leader graphic: Sets the block used for elevation label leaders.

Leader location: Sets whether leaders snap to the edge or center of conduits and cable trays.

Centerline block: Sets the block used for the centerline elevation label.

Centerline label: Sets how the centerline elevation is displayed on the label.

Move Elevation Label

The `Move Elevation Label` command is used to move an elevation label on a conduit or cable tray.

To move a conduit or cable tray elevation label, go to

Ribbon: `DM Electrical->3D-BIM->` `Move Elevation Label`

Pulldown Menu: `DM Electrical->3D-BIM->Move Elevation Label`

You will be prompted to identify which elevation label is to be moved.

Select elevation label to move:

You will then be prompted to specify the new location of the elevation label on the drawing. See the **Inserting Notes and Leaders on the Drawing** section for more information about moving the label.

Offset 3D Device Block

The `Offset 3D Device Block` command is used to position the 3D representation of a device to a different location from the 2D representation of the device. In 2D drafting, the location of a device does not always match its location in reality. For 3D collision detection, devices need to be inserted exactly where they belong in the building.

Insert devices at the location where the 2D representation goes. Then, if the project is using 3D-BIM, you can use this command to move the 3D representation.

To offset a 3D device block, go to

Ribbon: `DM Electrical->3D-BIM->` `Offset 3D Device Block`

Pulldown Menu: `DM Electrical->3D-BIM->Offset 3D Device Block`

You will be prompted to identify which device is to be offset.

Select device to offset 3D block:

You will then be prompted to specify the location for the 3D block.

Specify location for 3D block:

The 3D block will be moved to the specified location.

Reset 3D Device Block

The `Reset 3D Device Block` command is used to move the 3D representation of a device back to the same location as the 2D representation. This command undoes the `Offset 3D Device Block` command.

To reset the location of a 3D device block, go to

Ribbon: `DM Electrical->3D-BIM->` `Reset 3D Device Block`

Pulldown Menu: `DM Electrical->3D-BIM->Reset 3D Device Block`

You will be prompted to identify which device is to be reset.

Select device to reset 3D block offset:

The 3D block will be moved to the same location as the 2D block.

Export Entire Project to IFC File

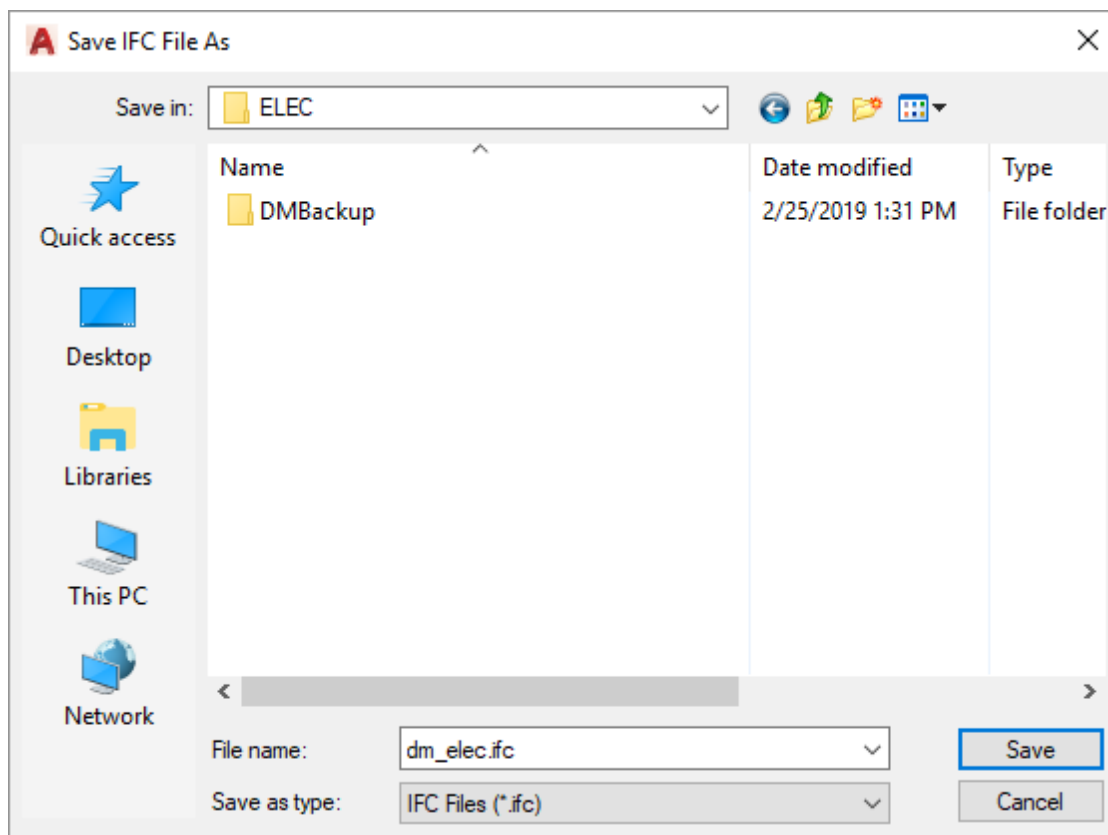The `Export Entire Project to IFC File` command is used to export all of the 3D-BIM elements from a

project to a single IFC file. The IFC file format is recommended for 3D collision detection in Revit.

To export the entire project to an IFC file, go to

Ribbon: `DM Electrical->3D-BIM->` ᴾᴿᴼᴶ `IFC` `Export Entire Project to IFC File`

Pulldown Menu: `DM Electrical->3D-BIM->Export Entire Project to IFC File`

Save IFC File As Dialog Box

Enter a name and location for the IFC file, then press the **Save** button. The entire project will be saved to the IFC file.

Related Options

3D circle approximation: Sets the number of sides on polygons used to approximate circles in the 3D model.

Export Drawing to IFC File

The `Export Drawing to IFC File` command is used to export all of the 3D-BIM elements from the current drawing to an IFC file. The IFC file format is recommended for 3D collision detection in Revit.

If you export to an IFC file that does not exist, it will be created with the 3D-BIM elements from the current

295

drawing.

If you export to an IFC file that exists but does not contain any 3D-BIM elements from the current drawing, the 3D-BIM elements from the current drawing will be appended to the end of the IFC file.

If you export to an IFC file that exists and contains 3D-BIM elements from the current drawing, the existing 3D-BIM elements in the IFC file will be removed and replaced with the current 3D-BIM elements on the drawing.

To export the current drawing to an IFC file, go to

Ribbon: `DM Electrical->3D-BIM->` DWG IFC `Export Drawing to IFC File`

Pulldown Menu: `DM Electrical->3D-BIM->Export Drawing to IFC File`

Save IFC File As Dialog Box
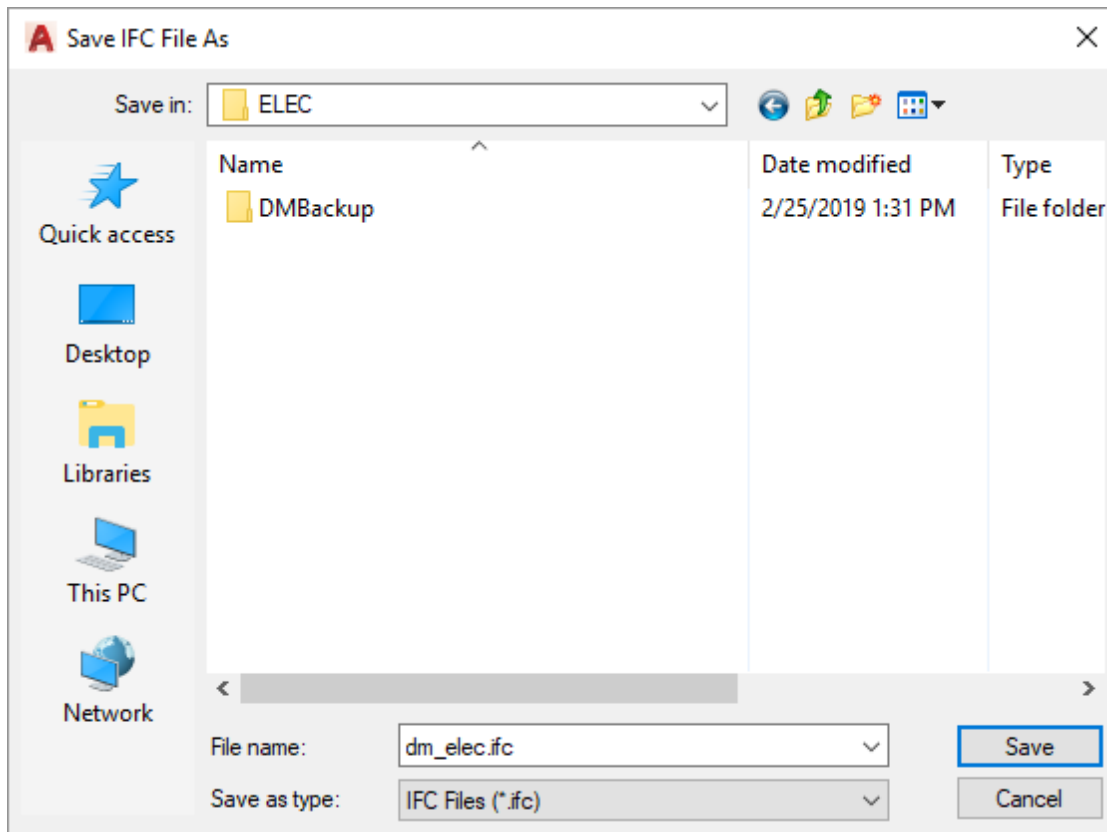
Enter a name and location for the IFC file, then press the **Save** button. The current drawing will be saved to the IFC file.

Related Options

3D circle approximation: Sets the number of sides on polygons used to approximate circles in the 3D model.

Export One Area to IFC File

The `Export One Area to IFC File` command is used to export all of the 3D-BIM elements from a single alignment point area to an IFC file. The IFC file format is recommended for 3D collision detection in Revit.

If you export to an IFC file that does not exist, it will be created with the 3D-BIM elements from the current drawing.

If you export to an IFC file that exists, but does not contain any 3D-BIM elements from the selected alignment point area, the 3D-BIM elements from the selected alignment point area will be appended to the end of the IFC file.

If you export to an IFC file that exists, and contains 3D-BIM elements from the selected alignment point area, the existing 3D-BIM elements in the IFC file will be removed and replaced with the current 3D-BIM elements in the selected alignment point area.

To export a single alignment point area to an IFC file, go to

Ribbon: `DM Electrical->3D-BIM->` **AREA IFC** `Export One Area to IFC File`

Pulldown Menu: `DM Electrical->3D-BIM->Export One Area to IFC File`

You will be prompted to identify the alignment point area to be exported.

Select point inside area to export:

The **Save IFC File As** dialog box will appear.

Save IFC File As Dialog Box
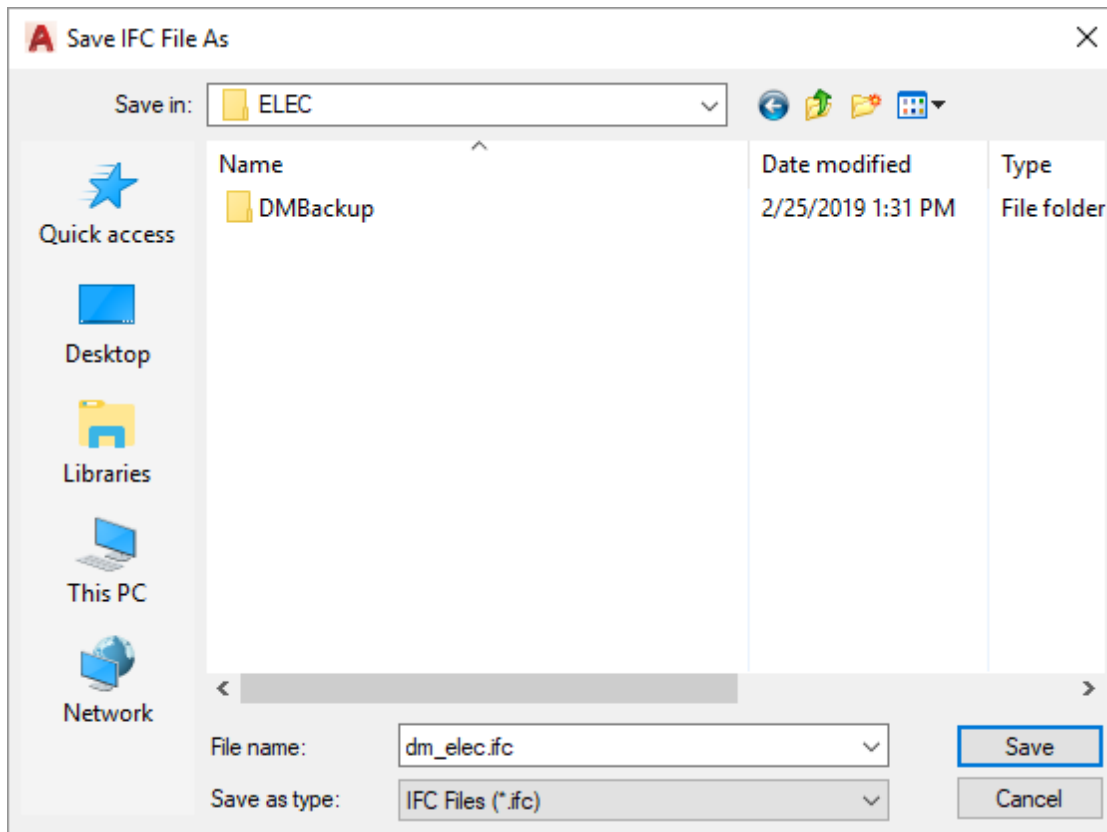
Enter a name and location for the IFC file, then press the **Save** button. The selected alignment point area will be saved to the IFC file.

Related Options

3D circle approximation: Sets the number of sides on polygons used to approximate circles in the 3D model.
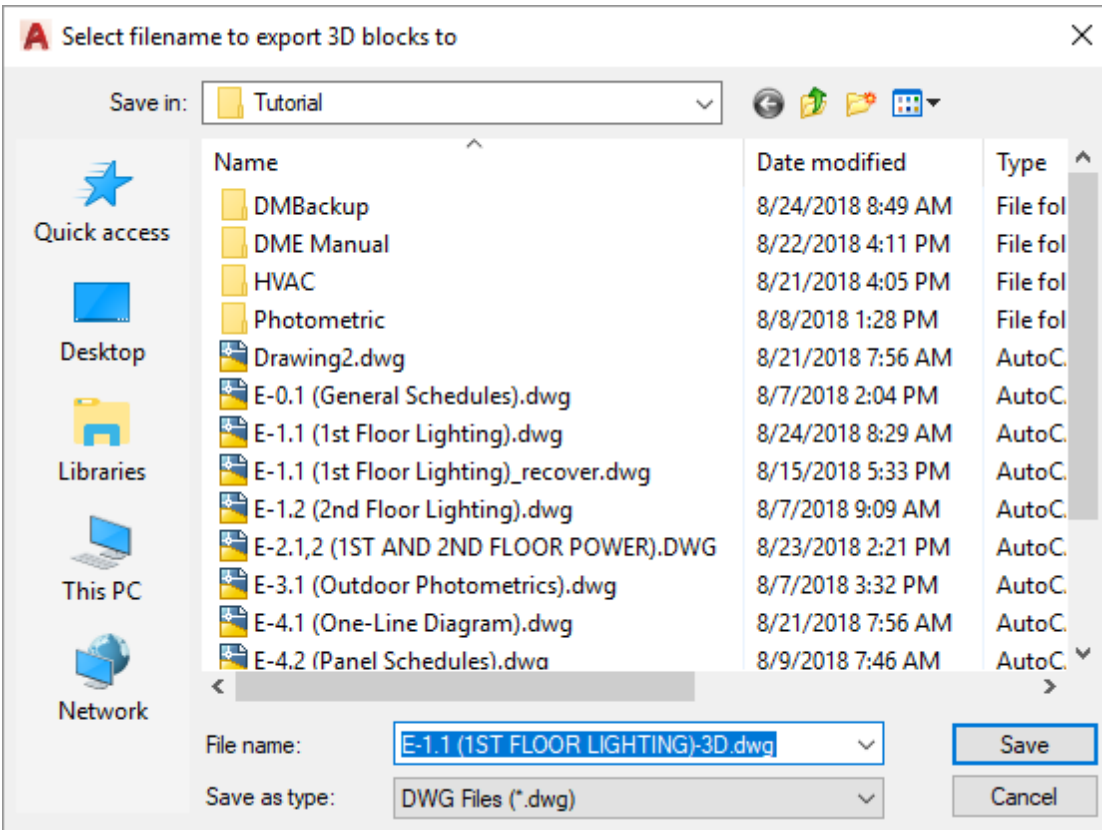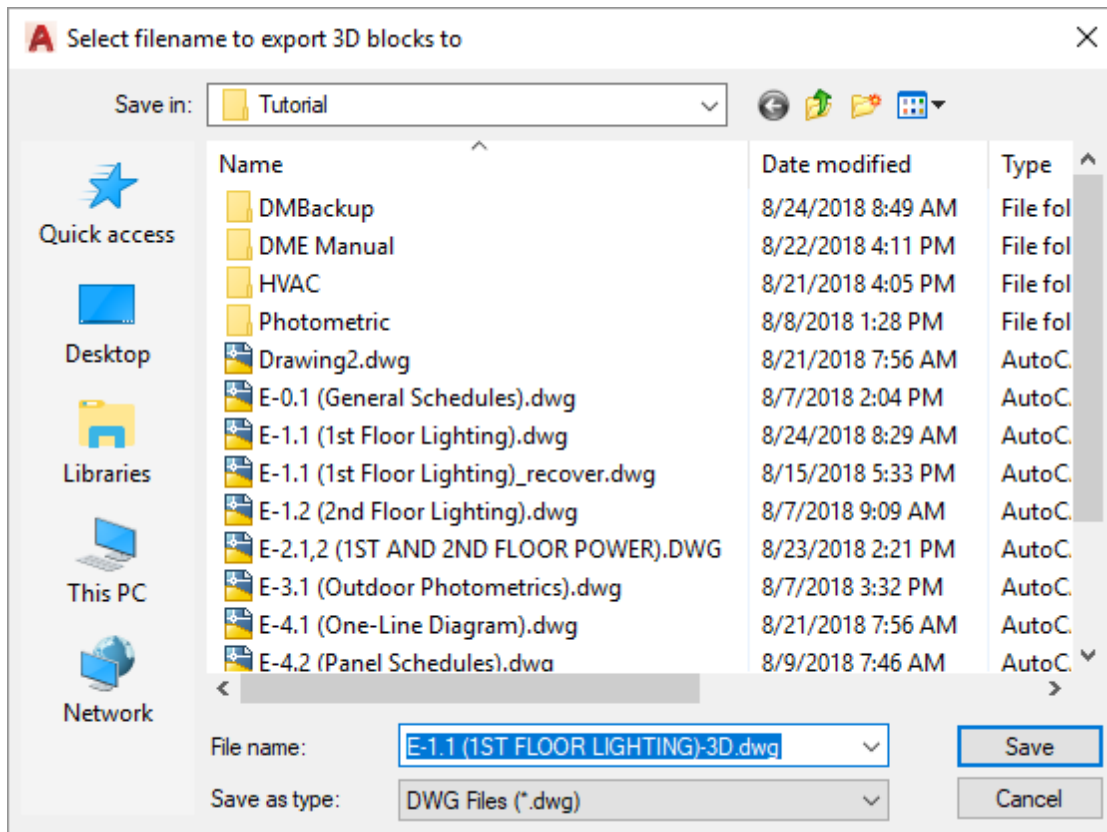
Export Entire Project to DWG File

The `Export Entire Project to DWG File` command is used to export all of the 3D-BIM elements from a project to a single DWG file. The DWG file format is recommended for 3D collision detection in Navisworks.

To export the entire project to a DWG file, go to

Ribbon: DM Electrical->3D-BIM-> PROJ DWG Export Entire Project to DWG File

Pulldown Menu: DM Electrical->3D-BIM->Export Entire Project to DWG File

Select Filename to Export 3D Blocks to Dialog Box

Enter a name and location for the DWG file, then press the **Save** button. The entire project will be saved to the DWG file.

Related Options

3D circle approximation: Sets the number of sides on polygons used to approximate circles in the 3D model.

Export Drawing to DWG File

The `Export Drawing to DWG File` command is used to export all of the 3D-BIM elements from the current drawing to a DWG file. The DWG file format is recommended for 3D collision detection in Navisworks.

To export the current drawing to a DWG file, go to

Ribbon: `DM Electrical->3D-BIM->` `DWG` `Export Drawing to DWG File`

Pulldown Menu: `DM Electrical->3D-BIM->Export Drawing to DWG File`

Select Filename to Export 3D Blocks to Dialog Box

Enter a name and location for the DWG file, then press the **Save** button. The current drawing will be saved to the DWG file.

Related Options

3D circle approximation: Sets the number of sides on polygons used to approximate circles in the 3D model.

Export One Area to DWG File

The `Export One Area to DWG File` command is used to export all of the 3D-BIM elements from a single alignment point area to a DWG file. The DWG file format is recommended for 3D collision detection in Navisworks.

To export a single alignment point area to a DWG file, go to

Ribbon: DM Electrical->3D-BIM-> ⬛AREA DWG Export One Area to DWG File

Pulldown Menu: DM Electrical->3D-BIM->Export One Area to DWG File

You will be prompted to identify the alignment point area to be exported.

Select point inside area to export:

The **Save DWG File As** dialog box will appear.

Select Filename to Export 3D Blocks to Dialog Box
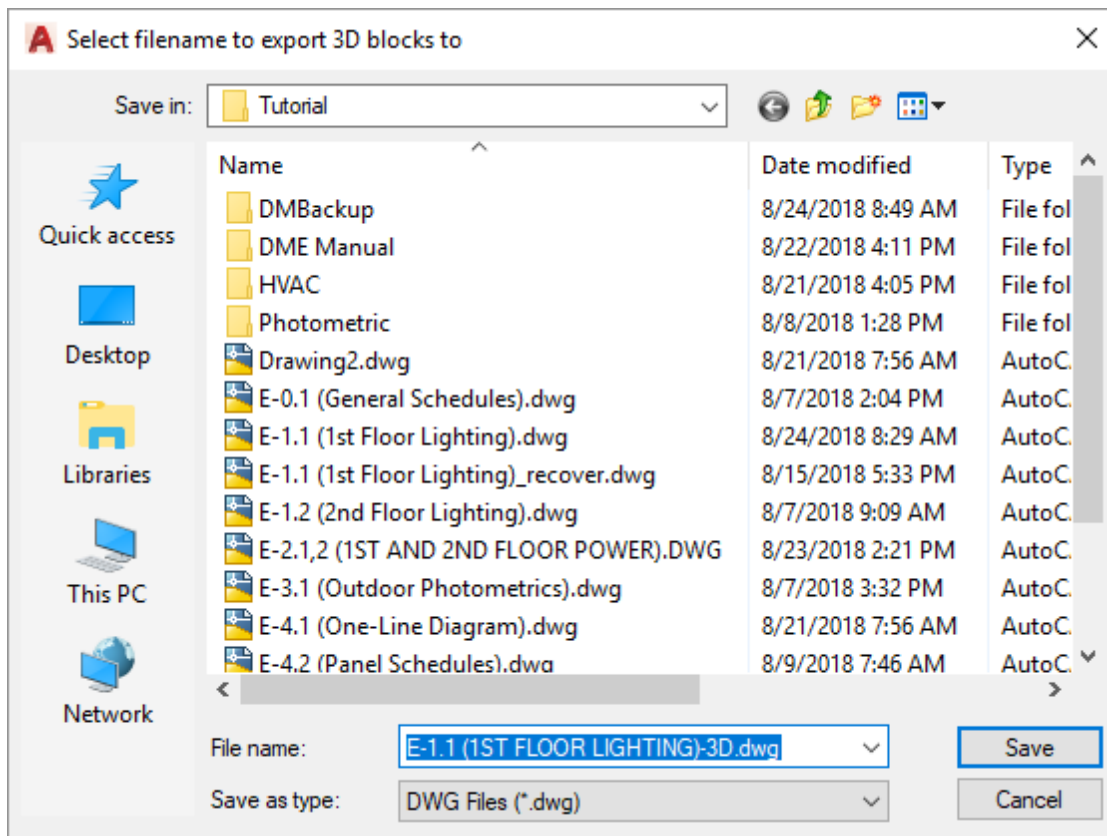
Enter a name and location for the DWG file, then press the **Save** button. The selected alignment point area will be saved to the DWG file.

Related Options

3D circle approximation: Sets the number of sides on polygons used to approximate circles in the 3D model.

Layer Management

This section describes the commands available to help you manage your layers. All of the layers created and used are standard CAD layers and can be managed using standard CAD commands. These commands are included for your convenience, but are not necessary. Everything you can do with these commands can be done with standard CAD commands.

These commands specifically change the *On* setting for the layers, toggling them between **On** and **Off**. The *Freeze* setting for the layers is not changed.

If you run these commands while in a paper space viewport, the *VP Freeze* setting will be changed as well. This can be helpful when setting up drawings to display only the 2D or 3D layers in a specific viewport. You can use the **Turn Double Line Layers On (and 3D Layers Off)** command in the viewport that should display the 2D

301

conduits and cable trays and the **Turn 3D Layers on (and Double Line Layers Off)** command in the viewport that should display the 3D conduits and cable trays. You can then return to model space and use the **Turn Double Line and 3D Layers On** command to display both the 2D and 3D conduits and cable trays. You can continue to update both the 2D and 3D drafting while you design, but only have the desired view shown in each viewport.

Turn Centerlines On

The `Turn Centerlines On` command turns conduit and cable tray centerline layers on. These layers are typically on a non-plotting layer that will not appear on your printed drawings. It is often helpful during the design to turn these centerlines on and off to see what your final drawing will look like.

To turn the centerlines on, go to

Ribbon: `DM Electrical->3D-BIM->` `Turn Centerlines On`

Pulldown Menu: `DM Electrical->Layer Management->Turn Centerlines On`

The centerline layers will be turned on and will be visible on the drawing.

Turn Centerlines Off

The `Turn Centerlines Off` command turns conduit and cable tray centerline layers off. These layers are typically on a non-plotting layer that will not appear on your printed drawings. It is often helpful during the design to turn these centerlines on and off to see what your final drawing will look like.

To turn the centerlines off, go to

Ribbon: `DM Electrical->3D-BIM->` `Turn Centerlines Off`

Pulldown Menu: `DM Electrical->Layer Management->Turn Centerlines Off`

The centerline layers will be turned off and will not be visible on the drawing.

Turn Double Line and 3D Layers On

The `Turn Double Line and 3D Layers On` command turns on the double line and 3D representations of conduits and cable trays.

To turn on the double line and 3D conduit and cable tray layers, go to

Ribbon: `DM Electrical->3D-BIM->` `Turn Double Line and 3D Layers On`

Pulldown Menu: `DM Electrical->Layer Management->Turn Double Line and 3D Layers On`

The double line and 3D conduit and cable tray layers will be turned on and visible on the drawing.

If this command is run while in a paper space viewport, the *VP Freeze* setting for the layers will be changed to **Thawed**.

Turn Double Line and 3D Layers Off

The `Turn Double Line and 3D Layers Off` command turns off the double line and 3D representations of conduits and cable trays.

To turn off the double line and 3D conduit and cable tray layers, go to

Ribbon: `DM Electrical->3D-BIM->` ⬛ `Turn Double Line and 3D Layers Off`

Pulldown Menu: `DM Electrical->Layer Management->Turn Double Line and 3D Layers Off`

The double line and 3D conduit and cable tray layers will be turned off and not visible on the drawing.

If this command is run while in a paper space viewport, the *VP Freeze* setting for the layers will be changed to **Frozen**.

Turn Double Line Layers On (and 3D Layers Off)

The `Turn Double Line Layers On (and 3D Layers Off)` command turns on the double line representations and turns off the 3D representations of conduits and cable trays.

The double line and 3D representations of conduits and cable trays are drawn slightly different. Viewing both of them at the same time can make it difficult to see what is drawn. Turning one on and the other off makes it easier to see the design.

Use this command to view the conduits and cable trays while in plan view.

To turn on the double line and turn off the 3D conduits and cable trays, go to

Ribbon: `DM Electrical->3D-BIM->` ⬛ `Turn Double Line Layers On (and 3D Layers Off)`

Pulldown Menu: `DM Electrical->Layer Management->Turn Double Line Layers On (and 3D Layers Off)`

The double line conduit and cable tray layers will be turned on and visible on the drawing.

If this command is run while in a paper space viewport, the *VP Freeze* setting for the double line layers will be changed to **Thawed**.

The 3D conduit and cable tray layers will be turned off and not visible on the drawing.

If this command is run while in a paper space viewport, the *VP Freeze* setting for the 3D layers will be changed to **Frozen**.

Turn 3D Layers On (and Double Line Layers Off)

The `Turn 3D Layers On (and Double Line Layers Off)` command turns off the double line representations and turns on the 3D representations of conduits and cable trays.

The double line and 3D representations of conduits and cable trays are drawn slightly different. Viewing both of them at the same time can make it difficult to see what is drawn. Turning one on and the other off makes it easier to see the design.

Use this command to view the conduits and cable trays while in an isometric or other 3D view.

To turn on the 3D layers and turn off the double line conduit and cable tray layers, go to

Ribbon: `DM Electrical->3D-BIM->` 3 `Turn 3D Layers On (and Double Line Layers Off)`

Pulldown Menu: `DM Electrical->Layer Management->Turn 3D Layers On (and Double Line Layers Off)`

The 3D conduit and cable tray layers will be turned on and visible on the drawing.

If this command is run while in a paper space viewport, the *VP Freeze* setting for the 3D layers will be changed to **Thawed**.

The double line conduit and cable tray layers will be turned off and not visible on the drawing.

If this command is run while in a paper space viewport, the *VP Freeze* setting for the double line layers will be changed to **Frozen**.

Point-by-Point Photometrics

This section describes how to calculate and display indoor, outdoor, and egress point-by-point photometrics.

Training Videos

- **Photometrics Overview**

Common Photometric Boundary Information

The values that can be specified for a photometric boundary are listed below.

Override Schedule Title: Whether the schedule title for the current calculation area is to be changed from the default calculation schedule title.

Schedule Title: The title to be used in the calculation schedule for the calculation area. This field is enabled if the *Override Schedule Title* box is checked.

Grid/Illuminance Level Spacing: The distance between points where illuminance levels will be calculated. See the **Distance Input Format** section for more information about specifying this distance.

Calculation Elevation: The elevation of the floor within the calculation area. See the **Distance Input Format** section for more information about specifying this elevation.

Boundary Obstructions: Whether the area has a ceiling and walls.
- **None:** The area has no ceiling or walls.
- **Ceiling:** The area has a ceiling.

- **Ceiling and Walls:** The area has a ceiling and walls. This setting prevents light generated from light fixtures outside the boundary from affecting the photometric calculations.

Ceiling Height: The height of the boundary ceiling. This setting is only available if *Boundary Obstructions* is set to **Ceiling** or **Ceiling and Walls**. See the **Distance Input Format** section for more information about specifying this height.

Surface Reflectance: The percentage of light reflected by the boundary. This setting is only available if *Boundary Obstructions* is set to **Ceiling** or **Ceiling and Walls**, and is only used if reflections are enabled in the calculation.

Top Elevation: This is the elevation at the top of the slope. See the **Distance Input Format** section for more information about specifying this elevation.

Bottom Elevation: This is the elevation at the bottom of the slope. See the **Distance Input Format** section for more information about specifying this elevation.

Calculate Vertical Illuminance Levels (At Area Boundary): Whether vertical illuminance levels are to be calculated along a calculation line or at the edges of a calculation area.

Vertical Calculation Spacing: The distance between points where vertical illuminance levels will be calculated. See the **Distance Input Format** section for more information about specifying this distance.

Vertical Display Spacing: The distance between vertical calculation points as they are displayed on the drawing. See the **Distance Input Format** section for more information about specifying this distance.

Ending Vertical Calculation Elevation (Above Slope): The maximum height at which vertical illuminance levels will be calculated. See the **Distance Input Format** section for more information about specifying this distance.

If the calculation area is sloped, the *Ending Vertical Calculation Elevation Above Slope* specifies the maximum vertical distance above each point along the slope. For example, an area with a *Top Elevation* of 30 and an *Ending Vertical Calculation Elevation Above Slope* of 10 will calculate illuminance levels from 30' to 40' at the top of the slope.

Common Photometric Solid Information

The values that can be specified for a photometric solid are listed below.

Starting Elevation: The elevation at which the photometric solid starts. See the **Distance Input Format** section for more information about specifying this elevation.

If this is 0 and the object is a closed area, no values inside the object will be calculated or displayed.

If this is greater than 0, the light levels underneath the object will be calculated. This setting can be used to model overhangs.

Ending Elevation: The elevation at which the photometric solid ends.

If this elevation is lower than the elevation of a light fixture, some light will pass over the object. See the **Distance Input Format** section for more information about specifying this elevation.

Reflectance: The percentage of light reflected by the photometric solid. It is only used if reflections are enabled in the calculation.

Point-by-Point Photometrics Calculations

There are two types of point-by-point photometric calculations available: general and egress. Each calculation uses separate fixtures, separate fixture lumens, and separate layers.

Separate Light Fixtures: Each calculation uses a separate set of light fixtures. Each light fixture on the drawing can be marked as being included in the general lighting calculation, egress lighting calculation, both calculations, or neither calculation. This is important for egress calculations, as only certain light fixtures will be on during an emergency situation. See the **Insert Light Fixtures** section for more information about specifying whether a light fixture is used for general or egress calculations.

Separate Light Fixture Lumens: In the light fixture schedule, the total lumens for the fixture can be set differently for the general calculation and egress calculation. This is important for egress calculations for fixtures, as only certain lamps will be on in an emergency situation. See the **Light Fixtures** section for more information about lumens settings.

Separate Layers: Light fixtures included in the egress calculation can be inserted on a separate layer from other light fixtures. The separate layer allows you to create a view of the drawing with only the egress light fixtures included. See the **Layers** section for more information.

Multiple Areas

Multiple **general calculation boundaries** and **egress calculation boundaries** can be defined on a single drawing. The general areas include the light from all general lights on the drawing. The egress areas include the light from all egress lights on the drawing.

Reflections

Reflections from buildings and walls can be included in the calculation. The walls for each boundary have a reflectance value that is used to calculate the amount of light that they reflect. For outdoor photometrics, this reflection results in slightly higher values near buildings. For indoor photometrics, wall reflections are an important component of the calculation.

Reflections from the ceiling can also be included in the calculation. The ceiling height and reflectance values are set when an area is first inserted and can be modified using the **Query Photometric Information** command.

IES Files

The illuminance level at each point within a defined boundary is calculated using the IES file for each light fixture. See the **Light Fixture Information** section for more information about light fixture IES files.

General Versus Egress Photometrics

There are two separate point-by-point photometric calculations: general and egress. They are calculated the same way using similar commands. There are two differences between the types of calculations.

Lumens / Lamps

Light fixtures have two settings for light output: *Lumens / Lamp* and *Egress Lumens / Lamp*. The general calculation uses *Lumens / Lamp*. The egress calculation uses *Egress Lumens / Lamp*.

The egress lumens is separate from the regular lumens to model situations where the light uses fewer lamps in emergency mode.

General Light vs Egress Light

Light fixtures have a *General Light* checkbox and an *Egress Light* checkbox. These checkboxes control whether the light fixture is included in the corresponding calculation. Light fixtures can have one, both, or neither of these boxes checked.

Insert General Photometric Calculation Area
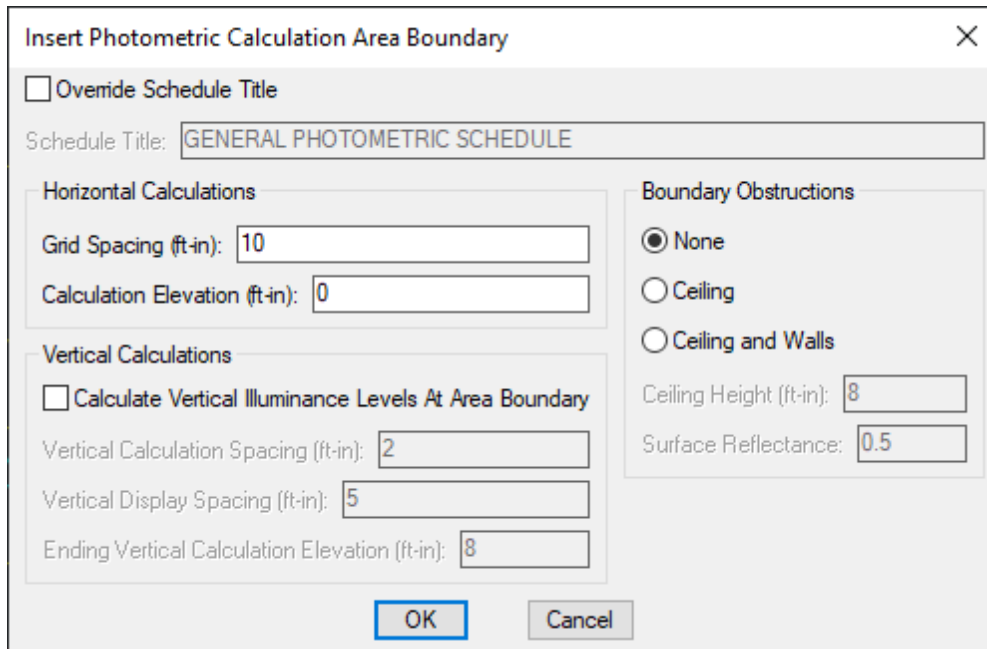
The `Insert General Photometric Calculation Area` command is used to insert a flat area on the drawing where general illuminance levels are to be calculated.

To insert a general area boundary on the drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` ▦ `Insert General Photometric Calculation Area`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Insert General Photometric Calculation Area`

Insert Photometric Calculation Area Boundary Dialog Box

See the **Common Photometric Boundary Information** section for information about the values in this dialog box.

Inserting the Calculation Area on the Drawing

To insert the calculation area on the drawing, press the **OK** button. The settings you specified in the dialog box will be used for the area.

You will be prompted to draw the calculation area on the drawing.

Specify first point of calculation area or [<Select existing boundary>]:

> ***First point of calculation area:*** Specify a point on the drawing.Continue entering points on the drawing until the calculation area is complete. Press **ENTER** to finish drawing the calculation area.

> ***Select existing boundary:*** Press **ENTER** to draw the calculation area by tracing a polyline on the drawing. Specify the polyline on the drawing to be traced. You will be asked whether to erase the polyline after the calculation area is inserted.

You will then be prompted to insert the calculation schedule, group the calculation area with another area, or press **ENTER** to finish the command.

Specify insertion point for calculation schedule or point inside boundary to group with:

> ***Insert a Calculation Schedule:*** To insert a calculation schedule, select a point on the drawing that is not inside of another calculation area. A calculation schedule will be inserted at the point specified.

> ***Group the Area with Another Group:*** To group the area with another group, select a point on the drawing that is inside of another calculation area. The two calculation areas will be grouped. If the selected calculation area already has a calculation schedule on the drawing, it will be used as the calculation schedule for the group. If it does not have a calculation schedule, no calculation schedule will be used for the group.

> ***Finish the Command:*** Press **ENTER** to finish the command without inserting a calculation schedule or grouping the calculation area with another group. No calculation schedule will be inserted and the calculation area will be in its own separate group.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Default grid spacing: Sets the default value for the *Grid Spacing* setting.

Insert General Sloped Photometric Calculation Area

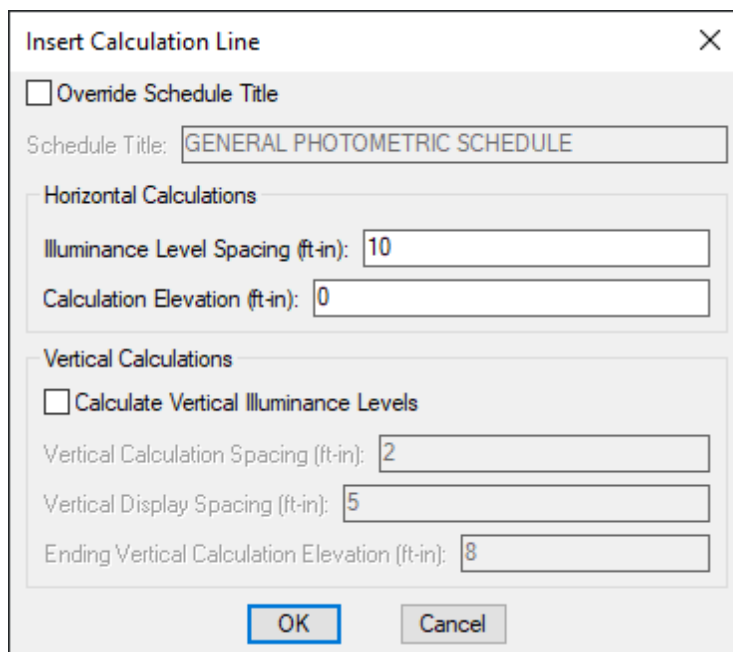The `Insert General Sloped Photometric Calculation Area` command is used to insert a sloped area on the drawing, such as stairs, where general illuminance levels are to be calculated.

To insert a sloped general area boundary on the drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Insert General Sloped Photometric Calculation Area`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Insert General Sloped Photometric Calculation Area`

Insert Sloped Photometric Calculation Area Boundary Dialog Box

See the **Common Photometric Boundary Information** section for information about the values in this dialog box.

Inserting the Sloped Calculation Area on the Drawing

To insert the calculation area on the drawing, press the **OK** button. The settings you specified in the dialog box will be used for the area.

You will be prompted to draw the calculation area on the drawing. The area will be inserted as a rectangle. The top elevation will be on one side of the rectangle and the bottom elevation will be on the opposite side of the rectangle. The two connecting sides will slope from the top side to the bottom side.

You will draw two lines to define the area. The first line should start at the top elevation of the slope and end at the bottom elevation of the slope. The top and bottom edges of the boundary will be drawn perpendicular to this line. The sides of the boundary will be drawn parallel to this line.

The second line you draw should start at one side of the slope and end at the opposite side. The locations of the sides of the rectangle will be determined by the start and end points of this line.

You will first be prompted to specify a point on the slope that is at the top elevation.

Specify top of calculation area:

You will then be prompted to specify a point on the slope that is at the bottom elevation.

Specify bottom of calculation area:

You will then be prompted to specify a point on one side of the slope.

Specify point on edge of calculation area:

You will then be prompted to specify a point on the opposite side of the slope.

Specify point on opposite edge of calculation area:

A rectangular boundary will be inserted on the drawing enclosing the specified points. The outline of the boundary will be thicker at the higher elevation to help you visualize how it slopes.

You will then be prompted to insert the calculation schedule, group the calculation area with another area, or press **ENTER** to finish the command.

Specify insertion point for calculation schedule or point inside boundary to group with:

> ***Insert a Calculation Schedule:*** To insert a calculation schedule, select a point on the drawing that is not inside of another calculation area. A calculation schedule will be inserted at the point specified.

> ***Group the Area with Another Group:*** To group the area with another group, select a point on the drawing that is inside of another calculation area. The two calculation areas will be grouped. If the selected calculation area already has a calculation schedule on the drawing, it will be used as the calculation schedule for the group. If it does not have a calculation schedule, no calculation schedule will be used for the group.

> ***Finish the Command:*** Press **ENTER** to finish the command without inserting a calculation schedule or grouping the calculation area with another group. No calculation schedule will be inserted and the calculation area will be in its own separate group.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Default grid spacing: Sets the default value for the *Grid Spacing* setting.

Insert General Photometric Calculation Line

The `Insert General Photometric Calculation Line` command is used to insert a line on the drawing where general illuminance levels are to be calculated.

To insert a general calculation line on the drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` 🟰 `Insert General Photometric Calculation Line`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Insert General Photometric Calculation Line`

Insert Calculation Line Dialog Box

See the **Common Photometric Boundary Information** section for information about the values in this dialog box.

Inserting the Calculation Line on the Drawing

To insert the calculation line on the drawing, press the **OK** button. The settings you specified in the dialog box will be used for the line.

You will be prompted to draw the calculation line on the drawing.

Specify first point of calculation line or [<Select existing line>]:

> **First point of calculation line:** Specify a point on the drawing.Continue entering points on the drawing until the calculation line is complete. Press **ENTER** to finish drawing the calculation line.

> **Select existing line:** Press **ENTER** to draw the calculation line by tracing a polyline on the drawing. Specify the polyline on the drawing to be traced. You will be asked whether to erase the polyline after the calculation line is inserted.

If you checked *Calculate Vertical Illuminance Levels*, you will then be prompted to specify on which side of the

line the vertical illuminance levels will be displayed.

Specify side to place vertical illuminance values:

You will then be prompted to insert the calculation schedule, group the calculation line with an area, or press **ENTER** to finish the command.

Specify insertion point for calculation schedule or point inside boundary to group with:

Insert a Calculation Schedule: To insert a calculation schedule, select a point on the drawing that is not inside of a calculation area. A calculation schedule will be inserted at the point specified.

Group with an Area: To group the line with an area, select a point on the drawing that is inside of a calculation area. The two will be grouped. If the selected calculation area already has a calculation schedule on the drawing, it will be used as the calculation schedule for the group. If it does not have a calculation schedule, no calculation schedule will be used for the group.

Finish the Command: Press **ENTER** to finish the command without inserting a calculation schedule or grouping the calculation line with an area. No calculation schedule will be inserted and the calculation line will be in its own separate group.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Default grid spacing: Sets the default value for the *Illuminance Level Spacing* setting.

Insert General Sloped Photometric Calculation Line

The `Insert General Sloped Photometric Calculation Line` command is used to insert a sloped line on the drawing where general illuminance levels are to be calculated.

To insert a general sloped calculation line on the drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Insert General Sloped Photometric Calculation Line`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Insert General Sloped Photometric Calculation Line`

Insert Sloped Calculation Line Dialog Box

See the **Common Photometric Boundary Information** section for information about the values in this dialog box.

Inserting the Calculation Line on the Drawing

To insert the calculation line on the drawing, press the **OK** button. The settings you specified in the dialog box will be used for the line.

You will be prompted to specify a point that is at the top elevation.

Specify top of calculation line:

You will then be prompted to specify a point that is at the bottom elevation.

Specify bottom of calculation line:

The line will be thicker at the higher elevation to help you visualize how it slopes.

If you checked *Calculate Vertical Illuminance Levels*, you will then be prompted to specify on which side of the line the vertical illuminance levels will be displayed.

Specify side to place vertical illuminance values:

You will then be prompted to insert the calculation schedule, group the calculation line with an area, or press **ENTER** to finish the command.

Specify insertion point for calculation schedule or point inside boundary to group with:

> **Insert a Calculation Schedule:** To insert a calculation schedule, select a point on the drawing that is not

inside of a calculation area. A calculation schedule will be inserted at the point specified.

Group with an Area: To group the line with an area, select a point on the drawing that is inside of a calculation area. The two will be grouped. If the selected calculation area already has a calculation schedule on the drawing, it will be used as the calculation schedule for the group. If it does not have a calculation schedule, no calculation schedule will be used for the group.

Finish the Command: Press **ENTER** to finish the command without inserting a calculation schedule or grouping the calculation line with an area. No calculation schedule will be inserted and the calculation line will be in its own separate group.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Default grid spacing: Sets the default value for the *Illuminance Level Spacing* setting.

Calculate General Photometrics

The `Calculation General Photometrics` command is used to calculate point-by-point photometrics inside of general calculation areas. All of the settings available in the command are the same as the `Calculate Egress Photometrics` command. This sections describes the options available for these two commands.

The two commands use different light fixtures and different settings on the light fixtures. See the **General Versus Egress Photometrics** section for more information about these differences.

To run general photometric calculations, go to

Ribbon: DM Electrical->Point-by-Point Photometrics-> Calculate General Photometrics

Pulldown Menu: DM Electrical->Point-by-Point Photometrics->Calculate General Photometrics

Calculate Photometrics Dialog Box

Contour Level 1 - 10: Enter the illuminance level at which to draw contours. You can have up to 10 contours. Contours set to 0 will not be drawn.

Layer or Color: Enter the layer or color of the contour. If the layer specified does not exist on the drawing, it will be created. Press the **Set Color** button to open the standard CAD **Select Color** dialog box.

Show Illuminance Levels and Contours: Select this option to show illuminance levels and contours on the drawing. The illuminance levels and contours will be displayed in the calculation area after the calculation is finished.

Show Illuminance Levels Only: Select this option to show illuminance levels on the drawing. The illuminance levels will be displayed in the calculation area after the calculation is finished.

Show Contours Only: Select this option to show contours on the drawing. The contours will be displayed in the calculation area after the calculation is finished.

Decimal Places on Drawing: The number of decimal places to include in the illuminance levels on the drawing.

Decimal Places in Schedule: The number of decimal places to include in the illuminance levels on the schedule. If the *Decimal Places on Drawing* is less than the *Decimal Places in Schedule*, the smaller *Decimal Places on Drawing* will be used for the *Maximum Illuminance* and the *Minimum Illuminance* values in the schedule.

Distance Unit: The base unit for your drawing. See the **Base Units and Photometrics Calculations** article in the knowledge base for more information about determining the correct *Distance Unit* setting for your drawing.
  - **Inch:** One unit on the drawing corresponds to one inch. The standard base unit for US architectural

315

drawings.

- **Foot:** One unit on the drawing corresponds to one foot. The standard base unit for US civil drawings.
- **Millimeter:** One unit on the drawing corresponds to one millimeter.
- **Centimeter:** One unit on the drawing corresponds to one centimeter.
- **Meter:** One unit on the drawing corresponds to one meter.

Illuminance Unit: Whether the illuminance levels are displayed in foot-candles or lux.

Reflections: Whether the calculation includes reflections from walls and ceilings.
- **No Reflection:** No reflections are included in the calculation. This option is faster, though the calculated values will be slightly lower than reality for indoor calculations and near walls outside.
- **Single Reflection:** Light is reflected once during the calculations. This option takes longer, but results in higher values near solids. The light is reflected off of one surface to the ground. Light that hits a second surface is not reflected. For example, light that goes from the light to a wall to the ground is included. Light that goes from the light to the floor to a wall, then back to the floor, is not included.

Edit Title, Column Labels, and Column Order: Press these buttons to modify the layout of the photometric schedule for general calculation areas and general calculation lines on the drawing. See the **Edit General Photometric Area Schedule List** section and **Edit General Photometric Line Schedule List** section for more information.

Zoom to Schedule After Insertion: Whether the display will be moved to the schedule associated with the calculation area.

Calculating Photometrics

After all of the information has been entered, press the **OK** button.

If there is only one calculation area on the drawing, the illuminance levels in the area will be calculated.

If there is more than one calculation area on the drawing, you will be prompted to specify which area to calculate.

Select calculation area to calculate / <Calculate all boundaries>:

> *Select calculation area:* Select a point inside an area to calculate the illuminance levels in just that area.

> *Calculate all boundaries:* Press `ENTER` to calculate the illuminance levels in all areas on the drawing.

The illuminance levels and contours in the area that is calculated will be updated. The values in the corresponding photometric calculation schedule will also be updated.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

There are several options that set the default values for this dialog box. See the **General Point-by-Point Photometrics Options** and **Egress Point-by-Point Photometrics Options** sections for more information.

Insert Egress Photometric Calculation Area
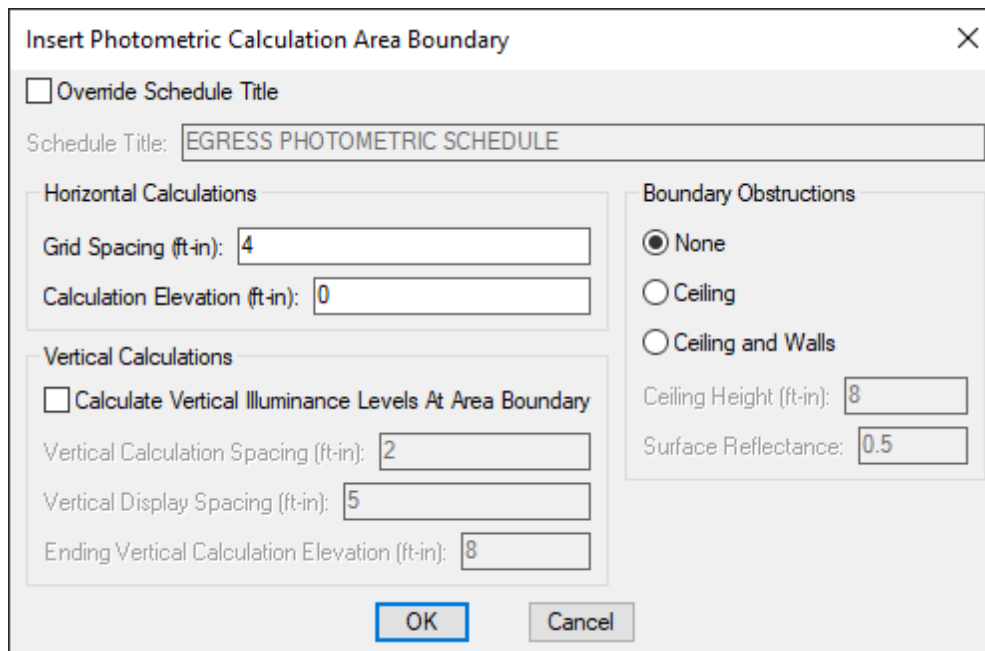
The `Insert Egress Photometric Calculation Area` command is used to insert a flat area on the drawing where egress illuminance levels are to be calculated.

To insert an egress area boundary on the drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` ▦ `Draw Egress Photometric Calculation Area`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Draw Egress Photometric Calculation Area`

Insert Photometric Calculation Area Boundary Dialog Box

See the **Common Photometric Boundary Information** section for information about the values in this dialog box.

Inserting the Egress Calculation Area on the Drawing

To insert the calculation area on the drawing, press the **OK** button. The settings you specified in the dialog box will be used for the area.

You will be prompted to draw the calculation area on the drawing.

Specify first point of calculation area or [<Select existing boundary>]:

> ***First point of calculation area:*** Specify a point on the drawing.Continue entering points on the drawing until the calculation area is complete. Press **ENTER** to finish drawing the calculation area.

Select existing boundary: Press **ENTER** to draw the calculation area by tracing a polyline on the drawing. Specify the polyline on the drawing to be traced. You will be asked whether to erase the polyline after the calculation area is inserted.

You will then be prompted to insert the calculation schedule, group the calculation area with another area, or press **ENTER** to finish the command.

Specify insertion point for calculation schedule or point inside boundary to group with:

Insert a Calculation Schedule: To insert a calculation schedule, select a point on the drawing that is not inside of another calculation area. A calculation schedule will be inserted at the point specified.

Group the Area with Another Group: To group the area with another group, select a point on the drawing that is inside of another calculation area. The two calculation areas will be grouped. If the selected calculation area already has a calculation schedule on the drawing, it will be used as the calculation schedule for the group. If it does not have a calculation schedule, no calculation schedule will be used for the group.

Finish the Command: Press **ENTER** to finish the command without inserting a calculation schedule or grouping the calculation area with another group. No calculation schedule will be inserted and the calculation area will be in its own separate group.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Default grid spacing: Sets the default value for the *Grid Spacing* setting.

Insert Egress Sloped Photometric Calculation Area

The `Insert Egress Sloped Photometric Calculation Area` command is used to insert a sloped area on the drawing, such as stairs, where egress illuminance levels are to be calculated.

To insert a sloped egress area boundary on the drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Insert Egress Sloped Photometric Calculation Area`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Insert Egress Sloped Photometric Calculation Area`

Insert Sloped Photometric Calculation Area Boundary Dialog Box

See the **Common Photometric Boundary Information** section for information about the values in this dialog box.

Inserting the Sloped Egress Calculation Area on the Drawing

To insert the calculation area on the drawing, press the **OK** button. The settings you specified in the dialog box will be used for the area.

You will be prompted to draw the calculation area on the drawing. The area will be inserted as a rectangle. The top elevation will be on one side of the rectangle and the bottom elevation will be on the opposite side of the rectangle. The two connecting sides will slope from the top side to the bottom side.

You will draw two lines to define the area. The first line should start at the top elevation of the slope and end at the bottom elevation of the slope. The top and bottom edges of the boundary will be drawn perpendicular to this line. The sides of the boundary will be drawn parallel to this line.

The second line you draw should start at one side of the slope and end at the opposite side. The locations of the sides of the rectangle will be determined by the start and end points of this line.

You will first be prompted to specify a point on the slope that is at the top elevation.

Specify top of calculation area:

You will then be prompted to specify a point on the slope that is at the bottom elevation.

Specify bottom of calculation area:

You will then be prompted to specify a point on one side of the slope.

Specify point on edge of calculation area:

You will then be prompted to specify a point on the opposite side of the slope.

Specify point on opposite edge of calculation area:

A rectangular boundary will be inserted on the drawing enclosing the specified points. The outline of the boundary will be thicker at the higher elevation to help you visualize how it slopes.

You will then be prompted to insert the calculation schedule, group the calculation area with another area, or press **ENTER** to finish the command.

Specify insertion point for calculation schedule or point inside boundary to group with:

Insert a Calculation Schedule: To insert a calculation schedule, select a point on the drawing that is not inside of another calculation area. A calculation schedule will be inserted at the point specified.

Group the Area with Another Group: To group the area with another group, select a point on the drawing that is inside of another calculation area. The two calculation areas will be grouped. If the selected calculation area already has a calculation schedule on the drawing, it will be used as the calculation schedule for the group. If it does not have a calculation schedule, no calculation schedule will be used for the group.

Finish the Command: Press **ENTER** to finish the command without inserting a calculation schedule or grouping the calculation area with another group. No calculation schedule will be inserted and the calculation area will be in its own separate group.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Default grid spacing: Sets the default value for the *Grid Spacing* setting.

Insert Egress Photometric Calculation Line

The `Insert Egress Photometric Calculation Line` command is used to insert a line on the drawing where egress illuminance levels are to be calculated.

To insert an egress calculation line on the drawing, go to

Ribbon: DM Electrical->Point-by-Point Photometrics-> Insert Egress Photometric Calculation Line

Pulldown Menu: DM Electrical->Point-by-Point Photometrics->Insert Egress Photometric Calculation Line

Insert Calculation Line Dialog Box

See the **Common Photometric Boundary Information** section for information about the values in this dialog box.

Inserting the Egress Calculation Line on the Drawing

To insert the calculation line on the drawing, press the **OK** button. The settings you specified in the dialog box will be used for the line.

You will be prompted to draw the calculation line on the drawing.

Specify first point of calculation line or [<Select existing line>]:

> **First point of calculation line:** Specify a point on the drawing.Continue entering points on the drawing until the calculation line is complete. Press **ENTER** to finish drawing the calculation line.

> **Select existing line:** Press **ENTER** to draw the calculation line by tracing a polyline on the drawing. Specify the polyline on the drawing to be traced. You will be asked whether to erase the polyline after the calculation line is inserted.

If you checked *Calculate Vertical Illuminance Levels*, you will then be prompted to specify on which side of the line the vertical illuminance levels will be displayed.

Specify side to place vertical illuminance values:

You will then be prompted to insert the calculation schedule, group the calculation line with an area, or press **ENTER** to finish the command.

Specify insertion point for calculation schedule or point inside boundary to group with:

Insert a Calculation Schedule: To insert a calculation schedule, select a point on the drawing that is not inside of a calculation area. A calculation schedule will be inserted at the point specified.

Group with an Area: To group the line with an area, select a point on the drawing that is inside of a calculation area. The two will be grouped. If the selected calculation area already has a calculation schedule on the drawing, it will be used as the calculation schedule for the group. If it does not have a calculation schedule, no calculation schedule will be used for the group.

Finish the Command: Press **ENTER** to finish the command without inserting a calculation schedule or grouping the calculation line with an area. No calculation schedule will be inserted and the calculation line will be in its own separate group.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Default grid spacing: Sets the default value for the *Illuminance Level Spacing* setting.

Insert Egress Sloped Photometric Calculation Line

The `Insert Egress Sloped Photometric Calculation Line` command is used to insert a sloped line on the drawing where egress illuminance levels are to be calculated.

To insert an egress sloped calculation line on the drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` 🖊 `Insert Egress Sloped Photometric Calculation Line`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Insert Egress Sloped Photometric Calculation Line`

Insert Sloped Calculation Line Dialog Box

See the **Common Photometric Boundary Information** section for information about the values in this dialog box.

Inserting the Egress Calculation Line on the Drawing

To insert the calculation line on the drawing, press the **OK** button. The settings you specified in the dialog box will be used for the line.

You will be prompted to specify a point that is at the top elevation.

Specify top of calculation line:

You will then be prompted to specify a point that is at the bottom elevation.

Specify bottom of calculation line:

Press **ENTER** to finish drawing the calculation line. The line will be thicker at the higher elevation to help you visualize how it slopes.

If you checked *Calculate Vertical Illuminance Levels*, you will then be prompted to specify on which side of the line the vertical illuminance levels will be displayed.

Specify side to place vertical illuminance values:

You will then be prompted to insert the calculation schedule, group the calculation line with an area, or press **ENTER** to finish the command.

Specify insertion point for calculation schedule or point inside boundary to group with:

Insert a Calculation Schedule: To insert a calculation schedule, select a point on the drawing that is not inside of a calculation area. A calculation schedule will be inserted at the point specified.

Group with an Area: To group the line with an area, select a point on the drawing that is inside of a calculation area. The two will be grouped. If the selected calculation area already has a calculation schedule on the drawing, it will be used as the calculation schedule for the group. If it does not have a calculation schedule, no calculation schedule will be used for the group.

Finish the Command: Press **ENTER** to finish the command without inserting a calculation schedule or grouping the calculation line with an area. No calculation schedule will be inserted and the calculation line will be in its own separate group.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Default grid spacing: Sets the default value for the *Illuminance Level Spacing* setting.

Calculate Egress Photometrics

The `Calculate Egress Photometrics` command is used to calculate point-by-point photometrics inside of egress calculation areas. All of the settings available in the command are the same as the `Calculate General Photometrics` command.

The two commands use different light fixtures and different settings on the light fixtures. See the **General Versus Egress Photometrics** section for more information about these differences.

To run egress photometric calculations, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Calculation Egress Photometrics`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics-> Calculation Egress Photometrics`

See the **Calculate General Photometrics** section for more information about how to use this command.

Edit Title, Column Labels, and Column Order: Press these buttons to modify the layout of the photometric schedule for egress calculation areas and egress calculation lines on the drawing. See the **Edit Egress Photometric Area Schedule List** section and **Edit Egress Photometric Line Schedule List** section for more information.

Query Calculation Area or Solid

The `Query Calculation Area or Solid` command is used to view and edit the information associated with calculation areas and photometric solids.

To view and edit these items, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Query Calculation Area or Solid`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Query Calculation Area or Solid`

You will be prompted to identify which calculation area boundary or photometric solid is to be edited.

Select calculation area boundary or light-obstructing object:

Depending upon your selection, one of the following dialog boxes will appear.

Editing a Calculation Area

Changes can be made to the calculation area in the same way as when it was inserted. See the **Common Photometric Boundary Information** section for information about the values in this dialog box.

Set Grid Origin: Press this button to change the point from which illuminance values are calculated for the selected calculation area. The dialog box will close and you will be prompted to specify a point on the drawing.

Specify illuminance grid origin for calculation area:

Set Grid Angle: Press this button to change the angle at which illuminance values are calculated for the selected calculation area. The dialog box will close and you will be prompted to specify an angle on the drawing.

Specify illuminance grid angle for calculation area / <0>:

Specify two points on the drawing, enter an angle value, or press **ENTER** to set an angle of 0.

Recalculate Illuminance Levels: Whether the queried calculation area will be recalculated when you press the **OK** button.

Editing a Calculation Line

Changes can be made to the calculation line in the same way as when it was inserted. See the **Common Photometric Boundary Information** section for information about the values in this dialog box.

Recalculate Illuminance Levels: Whether the queried calculation line will be recalculated when you press the **OK** button.

Editing a Photometric Solid

Changes can be made to the photometric solid in the same way as when it was inserted. See the **Common Photometric Solid Information** section for information about the values in this dialog box.

Aim Light Fixture

The `Aim Light Fixture` command is used to aim a light fixture at a specific point on the ground.

To aim a light fixture, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` ⊕ `Aim Light Fixture`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Aim Light Fixture`

Select the light fixture to be aimed.

Select light fixture:

You will then be prompted to specify the location on the ground where the light fixture will be aimed.

Select point to aim light fixture towards:

The light fixture will be rotated on the drawing in the direction of the specified point. If the light fixture has multiple heads, the head facing the insertion angle of the light fixture will face the specified point, and the other heads will rotate by the same angle.

The *Tilt* will be adjusted based upon the light fixture elevation to point toward the specified point.

The *Roll* will be set to 0.

See the **Insert Light Fixture** section for more information about the *Tilt* and *Roll* settings.

Insert Photometric Solid: Shape

The `Insert Photometric Solid: Shape` command is used to insert light-obstructing objects that are closed but not rectangular. If the *Starting Elevation* is **0**, any illuminance levels inside the shape will not be printed. Most photometric solids will be inserted using this command, as most buildings and other objects on your plans will require a non-rectangular shape.

To define a photometric solid shape, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Insert Photometric Solid: Shape`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Insert Photometric Solid: Shape`

Insert Photometric Solid: Shape Dialog Box
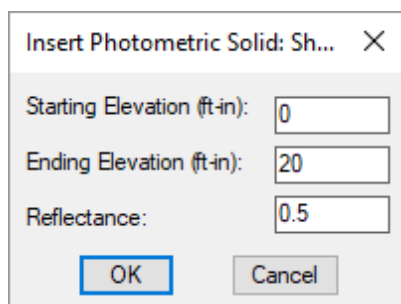
See the **Common Photometric Solid Information** section for more information about the values in this dialog box.

Enter the relevant information into the dialog box and press the **OK** button. You will be prompted to draw the solid shape on the drawing.

Specify first point of solid shape:

You will then be prompted to specify the next point of the solid.

Specify next point:

You can specify additional points or press **ENTER** to complete the shape. The solid shape will be displayed on the drawing and included in the photometric calculations.

Insert Photometric Solid: Rectangle
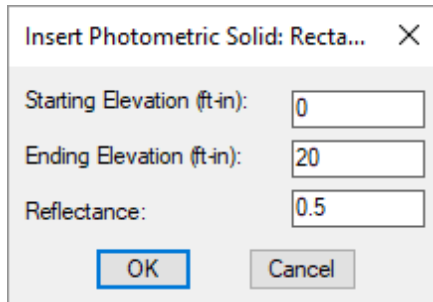
The `Insert Photometric Solid: Rectangle` command is used to insert light-obstructing objects that are rectangular. If the *Starting Elevation* is **0**, any illuminance levels inside the rectangle will not be printed.

To define a photometric solid rectangle, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Insert Photometric Solid: Rectangle`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Insert Photometric Solid: Rectangle`

Insert Photometric Solid: Rectangle Dialog Box

See the **Common Photometric Solid Information** section for more information about the values in this dialog box.

Enter the relevant information into the dialog box and press the **OK** button. You will be prompted to specify the first corner of the solid rectangle on the drawing.

Specify first corner of solid rectangle:

You will then be prompted to specify the second corner of the solid rectangle.

Specify second corner of solid rectangle:

The solid rectangle will be displayed on the drawing and included in the photometric calculations.

Insert Photometric Solid: Line
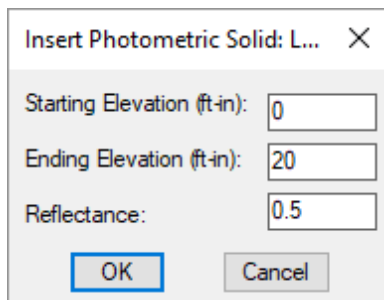
The `Insert Photometric Solid: Line` command is used to insert light-obstructing objects that are shaped like a line. These objects are not closed.

To define a photometric solid line, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` ⟋ `Insert Photometric Solid: Line`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Insert Photometric Solid: Line`

Insert Photometric Solid: Line Dialog Box

See the **Common Photometric Solid Information** section for more information about the values in this dialog box.

Enter the relevant information into the dialog box and press the **OK** button. You will be prompted to specify the

starting point of the solid line.

Specify first point of line:

You will then be prompted to specify the ending point of the solid line.

Specify next point:

You can specify additional line points or press **ENTER** to end the line. The solid line will be displayed on the drawing and included in the photometric calculations.

Insert Photometric Mask

The `Insert Photometric Mask` command is used to define an area in a calculation area where the illuminance values should not be calculated.

To insert a mask on the drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Insert Photometric Mask`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Insert Photometric Mask`

You will be prompted to specify the starting point of the mask.

Specify points for mask:
Specify start point:

You will then be prompted to specify the next point of the mask.

Specify next point:

You can specify additional points or press **ENTER** to complete the mask. The mask will be displayed on the drawing. The illuminance levels inside the area will not be calculated or displayed.

Add Photometric Vertex

To add a vertex to a photometric area, solid, or mask, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Add Photometric Vertex`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Add Photometric Vertex`

You will be prompted to identify the location on the area boundary, solid, or mask where you want a vertex to be added.

Select break point on photometric area, solid, or mask:

A new vertex will be inserted. The appearance of the object will not change, but you will have a new grip that you can use to drag the object into the correct shape.

Remove Photometric Vertex

To remove a vertex from a photometric area, solid, or mask, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` Remove Photometric Vertex

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Remove Photometric Vertex`

You will be prompted to identify the vertex to be removed.

Select point to remove on photometric area, solid, or mask:

The vertex will be removed.

Add Calculation Area to Group

The `Add Calculation Area to Group` command is used to group multiple calculation areas together. Calculations areas in a group share a common calculation schedule.

To add a calculation area to a calculation area group, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` Add Calculation Area to Group

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Add Calculation Area to Group`

You will be prompted to identify a calculation area in the group to which you want to add another calculation area.

Select calculation boundary:

You will then be prompted to identify a calculation area to be added to the group.

Select boundary to group with the first boundary:

The second calculation area will be added to the same group as the first calculation area. If the second calculation area was in another group, it will be removed from the group but all of the other areas in the group will remain. If the second area had a calculation schedule inserted on the drawing, it will be erased. The calculation schedule for the group will include the illuminance values from the second area.

Remove Calculation Area from Group

The `Remove Calculation Area from Group` command is used to remove a calculation area from a calculation area group.

To remove a calculation area from a calculation area group, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` Remove Calculation Area to Group

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Remove Calculation Area to`

```
Group
```

You will be prompted to identify which calculation area is to be removed from a calculation area group.

Select calculation boundary:

The calculation area will be removed from the group.

You will then be prompted for the insertion point for the calculation schedule for the selected area.

Specify insertion point for photometric schedule:

Specify an insertion point to insert the calculation schedule at that point, or press **ENTER** to finish the command.

Highlight Calculation Area Group

The `Highlight Calculation Area Group` command is used to highlight all of the calculation areas in a calculation area group.

To highlight grouped calculation areas on the drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Highlight Calculation Area Group`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Highlight Calculation Area Group`

You will be prompted to identify a grouped calculation area to highlight.

Select calculation boundary:

All of the calculation areas that are grouped with the selected calculation area will be highlighted on the drawing.

To remove the highlights, use the standard CAD **REGEN** command.

Add Photometric Calculation Schedule

To add a photometric calculation schedule to a drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` `Add Photometric Calculation Schedule`

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Add Photometric Calculation Schedule`

You will be prompted to identify the calculation area boundary to which you want to add a photometric calculation schedule.

Select boundary to add schedule to:

You will then be prompted for the insertion location of the photometric calculation schedule.

Specify insertion point for photometric schedule:

The location you specify will be used as the top-left corner for the photometric calculation schedule.

See the **Edit General Photometric Area Schedule List**, **Edit General Photometric Line Schedule List**, **Edit Egress Photometric Area Schedule List**, and **Edit Egress Photometric Line Schedule List** sections for more information about editing the photometric schedule layout.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Delete Photometric Calculation Schedule

To delete a photometric calculation schedule from the drawing, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` Delete Photometric Calculation Schedule

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Delete Photometric Calculation Schedule`

You will be prompted to identify which photometric calculation schedule is to be deleted.

Select schedule to delete:

The photometric calculation schedule will be removed from the drawing.

Edit Photometric Calculation Schedule Note

To edit a photometric calculation schedule note, go to

Ribbon: `DM Electrical->Point-by-Point Photometrics->` EDIT NOTE Edit Photometric Calculation Schedule Note

Pulldown Menu: `DM Electrical->Point-by-Point Photometrics->Edit Photometric Calculation Schedule Note`

You will be prompted to specify the boundary or schedule for which the schedule note is to be edited.

Select boundary or schedule to add note to:

The **Edit Photometric Schedule Note** dialog box will appear.

Schedule Note: This note is displayed below the photometric schedule on the drawing.

Press the **OK** button to close the dialog box. The note at the bottom of the calculation schedule will be updated.

Building Definition

This section describes how to define the building for the project. The room definitions are used for indoor photometric calculations. They can also be used to label circuits in distribution equipment schedules.

You do not need to model the building in your project if you are not going to use the features the building model provides.

Common Building Definition Information

The values that can be specified for floors and rooms within a building are listed below. Some values can be set in the room schedule only. Others can have a default set in the schedule that can be changed for individual floors or rooms.

Schedule-Only Settings

This section describes room values that can only be set in the schedule. These cannot be changed for individual rooms. If you need to include a room that has a different setting for one of these values, you must create a new record in the schedule.

Foot-candles Desired: The amount of light that needs to be provided in the room.

Lighting Power Density: The watts per square foot allowed in the room according to the local energy code.

Illuminance Category: The illuminance category for the group as defined in the *IESNA Lighting Handbook*. The categories are based upon the type of task being performed in the area. Lower letters correspond to lower foot-candle levels. Higher letters correspond to higher foot-candle levels.

Individual Floor and Room Settings

This section describes building definition values that be changed for individual floors and rooms. The value in the room schedule is the default value.

Room / Room Number: The name of the room.

Room Type: The room type taken from the room project schedule. Determines the default settings for the room.

Floor: The floor with which the room is associated, which can affect the ceiling height and room cavity ratio.

Area: The area of the room in square feet. This value is based upon the room outline inserted on the drawing. To change the value, you must change the room outline.

Room Cavity Ratio: The room cavity ratio is a measure of how effectively light fixtures will illuminate the room. It is calculated based upon the geometry of the room using the following formula:

Room Cavity Ratio = 2.5 x (*Ceiling Height - Workspace Height*) x *Perimeter / Area*

Description: The description of the room. The default value is the *Room Type*.

Ceiling Height: The height of the ceiling for this room. The default is set in the floor definition. See the **Distance Input Format** section for more information about specifying this height.

Workspace Height: The elevation of the workspace above the floor level. Used to calculate the room cavity ratio for indoor photometric calculations. See the **Distance Input Format** section for more information about specifying this elevation.

Wall Cavity Reflectance: The reflective value of the walls in the room type. Used with the IES file associated with a light fixture to calculate the coefficient of utilization of the fixture for indoor photometric calculations.

Effective Ceiling Cavity Reflectance: The reflective value of the ceiling in the room type. Used with the IES file associated with a light fixture to calculate the coefficient of utilization of the fixture for indoor photometric calculations.

Effective Floor Cavity Reflectance: The reflective value of the floor in the room type. Used with the IES file associated with a light fixture to calculate the coefficient of utilization of the fixture for indoor photometric calculations.

Dirt Depreciation Factor: The factor by which photometric calculations are reduced due to dirt in the room.

Floor List
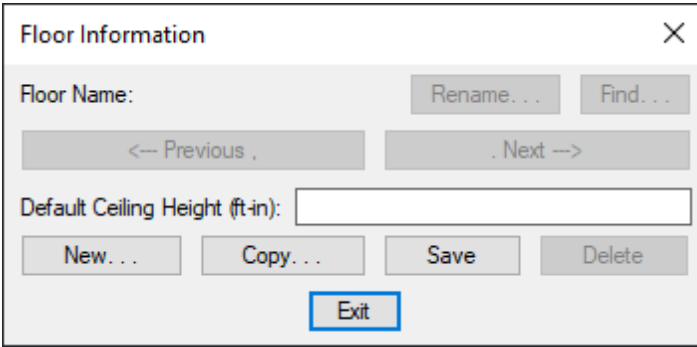
The `Floor List` command is used to create and modify floors in the project.

To modify the list of floors, go to

Ribbon: `DM Electrical->Buildings and Rooms->` FLIR `Floor List`

Pulldown Menu: `DM Electrical->Building Definition->Floor List`

Floor Information Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

Default Ceiling Height: The default height of the ceiling for rooms on this floor. The ceiling height is used to determine the room cavity ratio and mounting height for indoor photometrics. This value can be overridden for individual rooms. See the **Distance Input Format** section for more information about specifying this height.
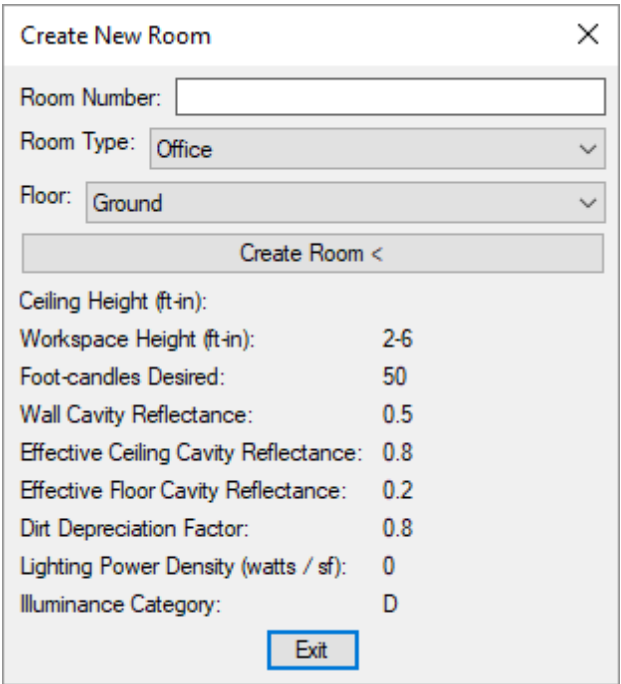
Create Room

To create a room on the current drawing, go to

Ribbon: `DM Electrical->Buildings and Rooms->` `Create Room`

Pulldown Menu: `DM Electrical->Building Definition->Create Room`

Create New Room Dialog Box

Room Number: The name of the room.

Room Type: The room type taken from the room project schedule. Determines the default settings for the room. See the **Room Project Schedule** section for more information.

Floor: The floor with which the room is associated. See the **Floor List** section for more information.

Create Room: Press this button to create the room on the drawing.

You will be prompted to specify the point at which you will begin drawing the room.

Specify first room point:

You will then be prompted to specify additional room points.

Specify next room point / <Finish>:

Press **ENTER** to complete the room outline.

The room will be displayed on the drawing. You will be prompted to select the location for the room label.

Specify room info location:

The room label will be inserted on the drawing with a leader pointing to the room.

Edit Rooms

To edit a room's definition, go to

Ribbon: DM Electrical->Buildings and Rooms-> ^{EDIT} Edit Rooms

Pulldown Menu: DM Electrical->Building Definition->Edit Rooms

Room Electrical Design Criteria Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, Save**, and **Delete** buttons.

There is no **Copy** button in this dialog box. To copy a room, use the standard CAD `COPY` command to copy it on the drawing.

Select from Drawing: Press this button to select the room from the drawing. You will be prompted to select a point inside the room.

Specify point inside room:

The dialog box will be reopen with the selected room active.

Check the box beside a setting to change it. See the **Common Building Definition Information** section for more information about the room settings.

Query Room

To view and edit a specific room's definition, go to

Ribbon: `DM Electrical->Buildings and Rooms->` `Query Room`

Pulldown Menu: `DM Electrical->Building Definition->Query Room`

You will be prompted to identify the room on the drawing that is to be queried.

Specify point inside room to query:

The **Room Electrical Design Criteria** dialog box will appear.

Room Electrical Design Criteria Dialog Box

Changes can be made to the room in the same way as when using the `Edit Room` command. See the **Edit Room** section for more information.

Update Room Labels

To update the information displayed in room labels, go to

Ribbon: `DM Electrical->Buildings and Rooms->` ⟳ `Update Room Labels`

Pulldown Menu: `DM Electrical->Building Definition->Update Room Labels`

All of the room labels on the drawing will be updated. Any calculated values will be updated to reflect any changes to the design.

Insert Wall Point

To add a point to a room outline, go to

Ribbon: `DM Electrical->Buildings and Rooms->` `Insert Wall Point`

Pulldown Menu: `DM Electrical->Building Definition->Insert Wall Point`

You will be prompted to locate a point either inside or outside the wall onto which a point is to be inserted.

First fence point crossing wall:

You will then be prompted to provide a point on the opposite side of the wall. The line between the two points should cross the wall where you want the new point to be inserted.

Second fence point:

A new point will be inserted on the room outline. The appearance of the outline will not change, but you will have two new grips that you can use to drag the outline into the correct shape.

Remove Wall Point

To remove a point from a room outline, go to

Ribbon: `DM Electrical->Buildings and Rooms->` `Remove Wall Point`

Pulldown Menu: `DM Electrical->Building Definition->Remove Wall Point`

You will be prompted to draw a crossing box around the room point to remove.

First crossing point around room vertex to remove:

You will then be prompted to specify the opposite corner of the crossing box.

Second crossing point:

The point on the room outline that is surrounded by the box will be removed.

Show or Hide Label Leaders

To show or hide room label leaders, go to

Ribbon: `DM Electrical->Buildings and Rooms->` `Show or Hide Label Leaders`

Pulldown Menu: `DM Electrical->Building Definition->Show or Hide Label Leaders`

You will be prompted to select one or more room outlines or room labels for which the leaders will be shown or

hidden.

Select objects:

Label leaders will be shown if they were not displayed previously or hidden if they were shown previously.

Draw Room

The `Draw Room` command is used to draw a room that already exists in the database. If the room is inserted on the drawing, this command will erase and reinsert the room.

Do not use this command to create a new room. Use the `Create Room` command instead. See the **Create Room** section for more information.

To draw a room, go to

Ribbon: `DM Electrical->Buildings and Rooms->` 🖊 `Draw Room`

Pulldown Menu: `DM Electrical->Building Definition->Draw Room`

Redraw Room Dialog Box
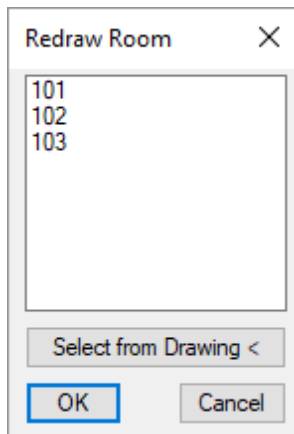
Room List: Select the room to draw from the list. The list displays all of the rooms in the project.

Select from Drawing: Press this button to select the room to draw from the drawing. You will be prompted to select a point inside the room.

Specify point inside room:

The dialog box will reopen with the specified room selected in the room list.

Drawing the Room

Select the room to draw and press the **OK** button. If the room was previously inserted on the drawing, it will be removed. You will be prompted to specify the point at which you will begin drawing the room.

Specify first room point:

You will then be prompted to specify additional room points.

Specify next room point / <Finish>:

Press **ENTER** to complete the room outline.

The room will be displayed on the drawing. You will be prompted to select the location for the room label.

Specify room info location:

The room label will be inserted on the drawing with a leader pointing to the room.

Insert Room Outlines on this Drawing

The `Insert Room Outlines on this Drawing` command is used to insert room outlines that have already been inserted on another drawing or in another alignment point area in the project. When a room is changed in any area, the room will be updated in all of the other areas at the same time.

To insert rooms in another area or drawing, go to

Ribbon: `DM Electrical->Buildings and Rooms->` 🔳 `Insert Room Outlines on this Drawing`

Pulldown Menu: `DM Electrical->Building Definition->Insert Room Outlines on this Drawing`

Insert Room Outlines Dialog Box
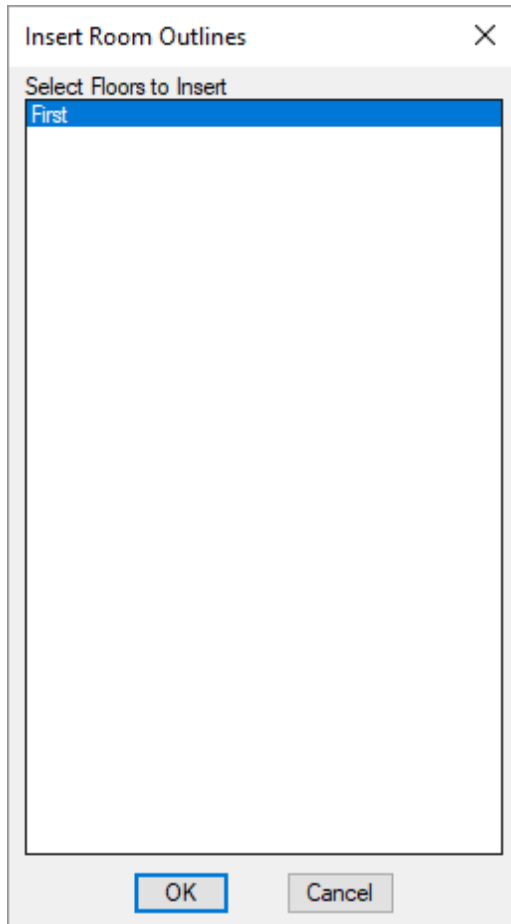
Select the floor to be inserted onto the drawing and press the **OK** button.

If there is only one alignment point on the drawing, it will be selected automatically. If there are multiple alignment points on the drawing, you will be prompted to select the alignment point area to draw the rooms in.

Specify point in alignment point area to use:

All of the rooms on the selected floor will be drawn in the selected alignment point area. The room locations will be based upon the location of the alignment point in the area.

Room Project Schedule

The room project schedule contains the room types that can be used in the current project.

To view or modify room types, go to

Ribbon: `DM Electrical->Buildings and Rooms->` ᴿᴼᴼᴹ P `Room Project Schedule`

Pulldown Menu: `DM Electrical->Building Definition->Room Project Schedule`

Room Lighting Design Criteria Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Common Building Definition Information** section for more information about the values that can be specified for rooms using this dialog box.

See the **Import from Master Schedule** section for more information about importing rooms to your project from the master schedule.

Room Master Schedule

The room master schedule contains all of the room types that could be used in a project. See the **Master and Standards Databases** section for more information about using master databases.

To create and modify room types in the master schedule, go to

Ribbon: `DM Electrical->Buildings and Rooms->` ROOM `M` `Room Master Schedule`

Pulldown Menu: `DM Electrical->Building Definition->Room Master Schedule`

Room Electrical Design Criteria Master Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

See the **Common Building Definition Information** section for more information about the values that can be specified for rooms using this dialog box.

See the **Import from Project Schedule** section for more information about importing rooms to your project from the master schedule.

Turn Room Layers On

To turn room layers on so that room outlines and room labels are displayed on the drawing, go to

Ribbon: `DM Electrical->Buildings and Rooms->` `Turn Room Layers On`

Pulldown Menu: `DM Electrical->Building Definition->Turn Room Layers On`

The room outline and room label layers will be turned on and will be visible on the drawing.

By default, these layers are non-plotting and do not have to be turned off before plotting.

Turn Room Layers Off

To turn room layers off so that room outlines and room labels are not displayed on the drawing, go to

Ribbon: `DM Electrical->Buildings and Rooms->` ⬓ `Turn Room Layers Off`

Pulldown Menu: `DM Electrical->Building Definition->Turn Room Layers Off`

The room outline and room label layers will be turned off and will not be visible on the drawing.

By default, these layers are non-plotting and do not have to be turned off before plotting.

Transfer Building Information from HVAC

The `Transfer Building Information from HVAC` command is used to copy floor and room definitions from a Design Master HVAC project. Using this command allows you to define the building once and use it in both the HVAC and electrical project.

To transfer building information from a Design Master HVAC project, go to

Ribbon: `DM Electrical->Buildings and Rooms->` UTIL `->Transfer Building Information from HVAC`

Pulldown Menu: `DM Electrical->Building Definition->Transfer Building Information from HVAC`

The dialog box that appears depends upon whether you have previously run this command in this project.

Select HVAC Database to Transfer Rooms From Dialog Box

The first time you run the command, you will be prompted to choose the HVAC database from which the building information will be transferred.
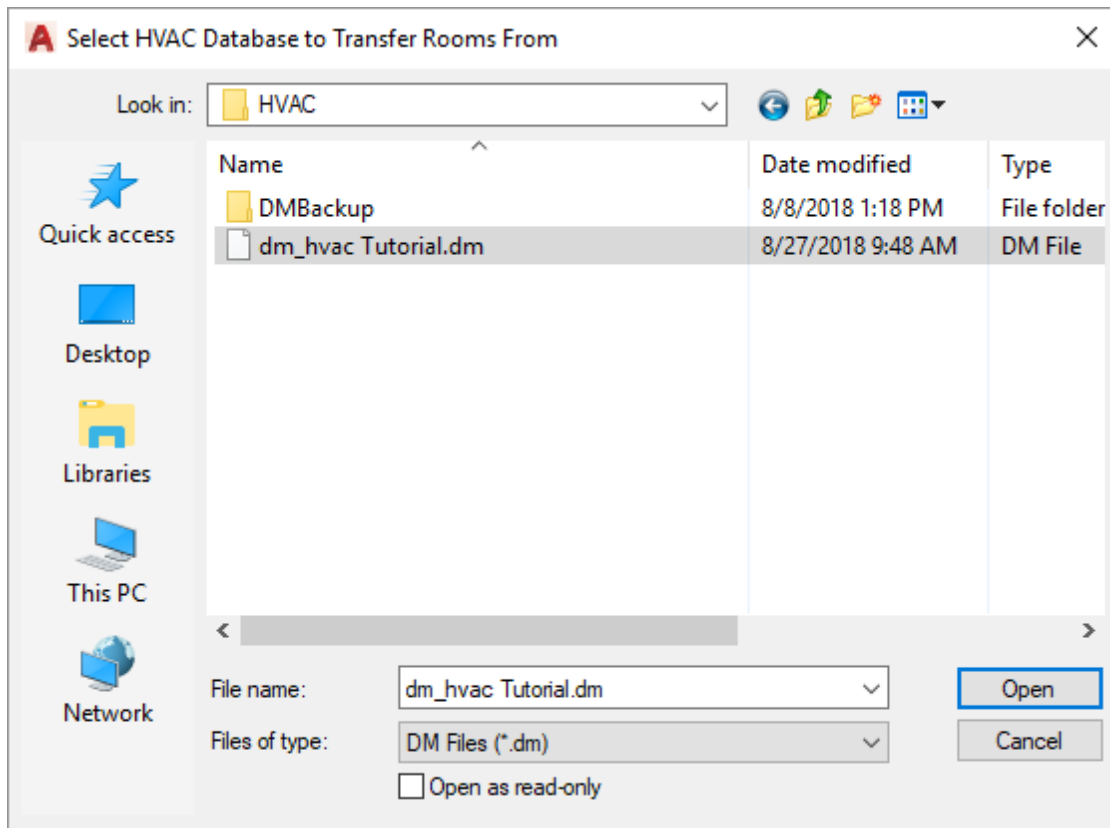
Select the desired HVAC project from which to copy room definitions and press the **Open** button. The **Transfer Rooms** dialog box will appear.

Transfer Rooms Dialog Box
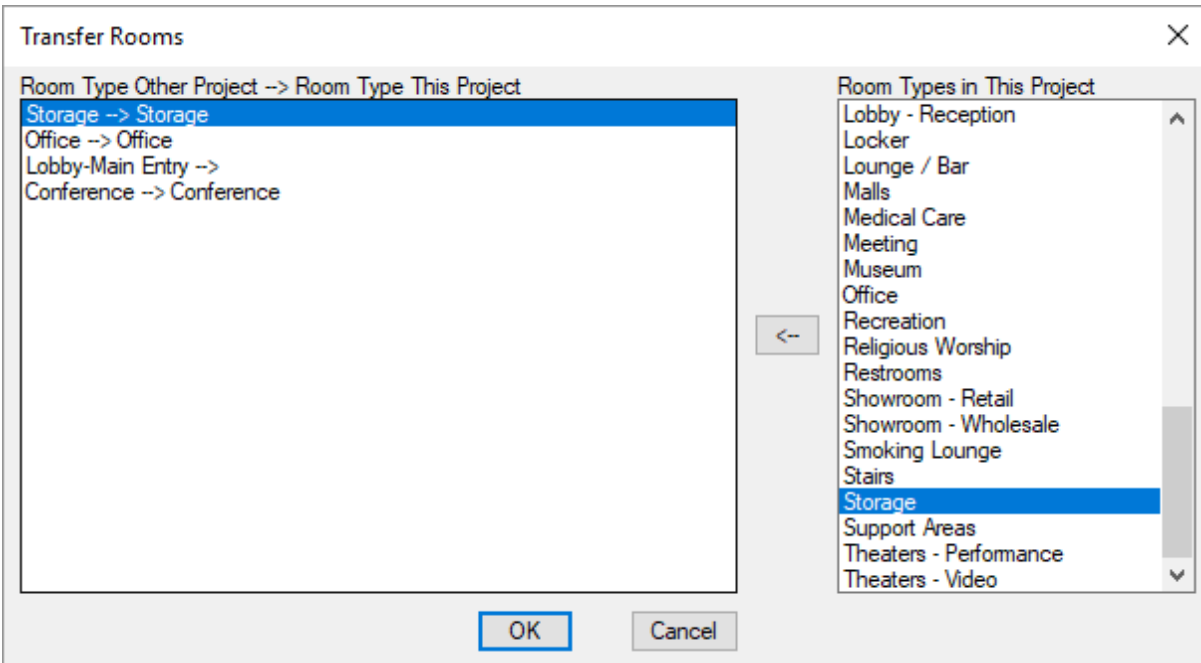
This dialog box is used to match room types between the HVAC project and the electrical project.

Room Type Other Project --> Room Type This Project: Lists all of the room types in the HVAC project and the room type that will be used in the electrical project.

Room Types in This Project: Lists the room types available in the electrical project.

<--: Press this button to assign the room type selected in the *Room Types in This Project* list to the room type selected in the *Room Type Other Project --> Room Type This Project* list.

Press the **OK** button to copy the rooms and floors from the HVAC project.

Design Master Dialog Box

After you run the command the first time, you will be prompted to transfer building information from the previously selected HVAC database.
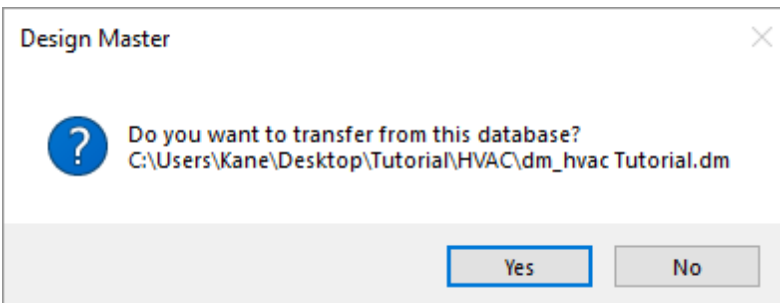
Yes: Press this button to import the building information again from the previously selected HVAC database.

No: Press this button to import the building information from a different HVAC database. The **Select HVAC Database to Transfer Floors From** dialog box, described above, will appear.

Takeoffs

This section describes how to generate a quantity takeoff directly from the device counts and wire measurements stored in the project database. Takeoffs may be performed at any time during the design process and can be quickly redone as the design changes. Each takeoff is generated as an HTML file and will automatically be opened in your web browser. It can also be opened in Microsoft Excel.

This feature is not intended to be used as a complete estimating tool. It is intended as a starting point for creating an estimate that can then be finished using a complete electrical estimating program.

There are three categories for which a takeoff may be performed. A takeoff may be performed on selected areas on a drawing, on distribution equipment and feeders for the project, or on equipment that is stored only in the database and not on a drawing file.

Follow the links below to learn more about the takeoff tools available:

- To perform a takeoff on an area on a drawing, you must first **Insert a Takeoff Area**. These takeoff areas will remain on the drawing until erased and can be used again for future takeoffs. The definition of the takeoff area can be modified using the **Query Takeoff Area** command.
- Once you have created a takeoff area, use the **Print Takeoff Area** command to perform the takeoff.
- Distribution equipment and feeders are not included in the quantities in the takeoff area on your drawing. To perform a takeoff of these items, use the **Print Distribution Equipment Takeoff** command.
- To perform a takeoff of the equipment that is in the database but not on a drawing, use the **Print Equipment in Database Only Takeoff** command.

Common Takeoff Information

Takeoff Area Name: The name of the takeoff area. The name is displayed when the takeoff for the area is printed.

Include Circuit Length by Circuit: Whether the *Circuit Length By Circuit* section is included when the takeoff for the area is printed. See the **Print Takeoff Area** section for more information.

Include Feeder Length by Feeder: Whether the *Feeder Length By Feeder* section is included when the takeoff is printed. See the **Print Distribution Equipment Takeoff** section for more information.

Layer Systems to Count: Which layers are included when the takeoff for the area is printed. See the **Layers** section for more information about layers.

Insert Takeoff Area

To create a takeoff area, go to

Ribbon: `DM Electrical->Takeoffs->` `Insert Takeoff Area`

Pulldown Menu: `DM Electrical->Takeoffs->Insert Takeoff Area`

Insert Takeoff Area Dialog Box
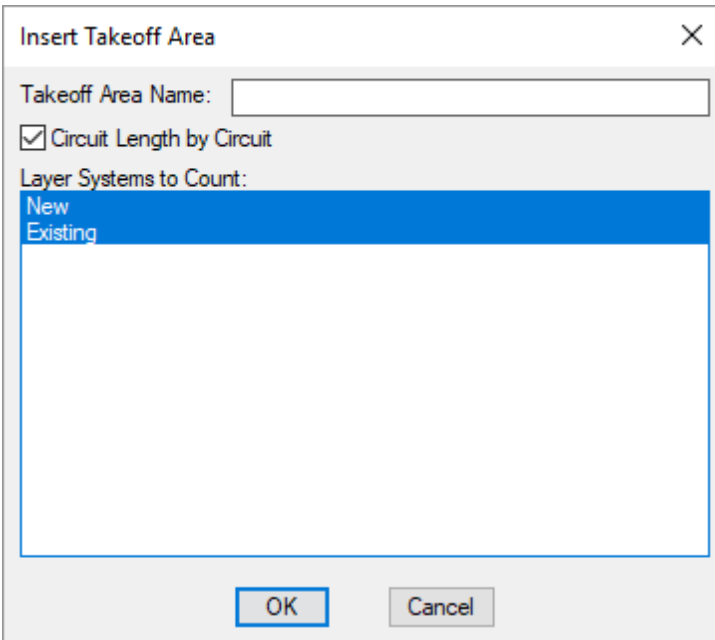
See the **Common Takeoff Information** section for more information about how this dialog box works.

Enter a name for the takeoff area and press the **OK** button.

You will be prompted to specify the first point of the takeoff area.

First point of takeoff area:

You will then be prompted to specify the next point.

Specify next point:

Continue inserting points until the takeoff area outline is defined. Press **ENTER** to complete the takeoff area outline.

A polyline will appear representing your takeoff area. The polyline is on a non-plotting layer and will not display on printed drawing files.

When a takeoff is performed in the area, all of the branch circuit devices in the area will be included. Circuits will be included in the takeoff if the device the homerun is drawn from is inside the area.

Query Takeoff Area

To query a takeoff and verify the area in which you are working, go to

Ribbon: `DM Electrical->Takeoffs->` `Query Takeoff Area`

Pulldown Menu: `DM Electrical->Takeoffs->Query Takeoff Area`

You will be prompted to identify a takeoff area.

Select point inside takeoff area:

The **Query Takeoff Area** dialog box will appear.

Query Takeoff Area Dialog Box
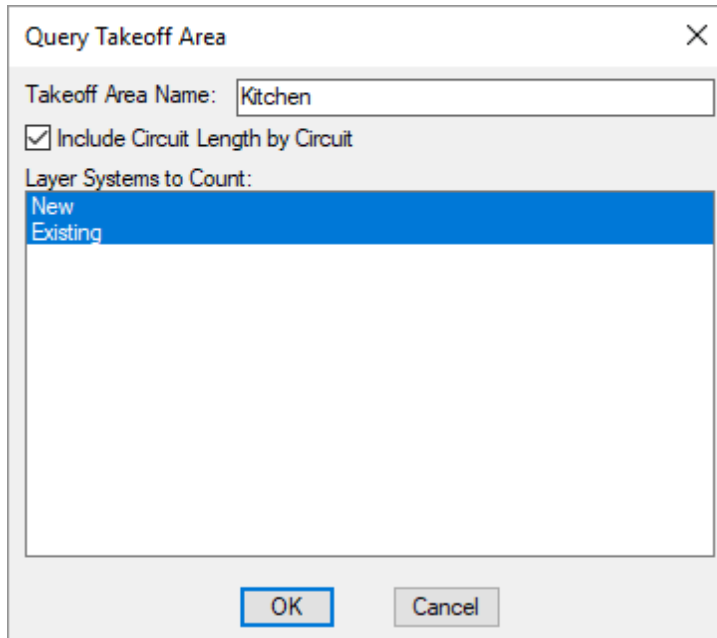
Changes can be made to the takeoff area in the same way as when it was inserted. See the **Common Takeoff Information** section for more information about how this dialog box works.

If you specify a point inside multiple takeoff areas, the **Select Takeoff Area** dialog box will appear.
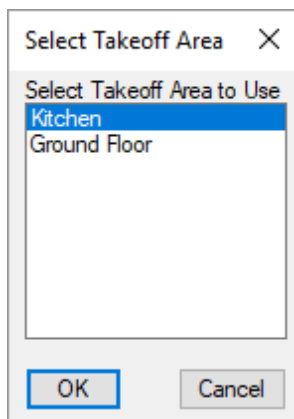
Select the desired takeoff area and press the **OK** button.

Print Takeoff Area

To perform a takeoff, go to

Ribbon: `DM Electrical->Takeoffs->` `Print Takeoff Area`

Pulldown Menu: `DM Electrical->Takeoffs->Print Takeoff Area`

You will be prompted to identify which takeoff area is to be used for the takeoff calculations.

Specify point inside takeoff area:

If you specify a point inside multiple takeoff areas, the **Select Takeoff Area** dialog box will appear.
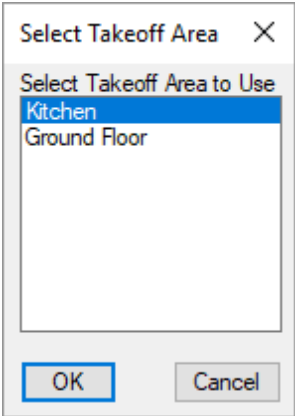
Select the takeoff area you want to use and press the **OK** button. The takeoff will be generated as an HTML file and displayed in your web browser. An example takeoff is shown below.

Electrical Takeoff: Ground Floor			
Receptacles: General			
Duplex	5		
Quad	3		
Circuit Length (Branch Wiring + Homerun)			
Wire Size	**Length**		**Loops**
1/2"C,1#12,#12N,#12G	170'		4
Branch Wiring Length			
Wire Size	**Length**		**Loops**
1/2"C,1#12,#12N,#12G	10'		2
Homerun Circuit Length			
Wire Size	**Length**		**Loops**
1/2"C,1#12,#12N,#12G	161'		2
Circuit Length By Circuit			
Description	**Wire Size**	**Length**	**Loops**
L1-9	1/2"C,1#12,#12N,#12G	84'	2
L1-11	1/2"C,1#12,#12N,#12G	87'	0

Related Options

[**Feeder make-up:**](#) Sets additional length to be added to feeders for connections.

This option can be further customized with the **Ignore make-up on feeders shorter than this** option.

[**Branch circuit make-up**](#)**:** Sets additional length to be added to branch circuits for connections.

This option can be further customized with the **Add branch circuit make-up for each device** option.

Print Distribution Equipment Takeoff

To perform a takeoff of distribution equipment and feeders in your project, go to

Ribbon: `DM Electrical->Takeoffs->` `[DE]` `Print Distribution Equipment Takeoff`

Pulldown Menu: `DM Electrical->Takeoffs->Print Distribution Equipment Takeoff`

Print Distribution Equipment Takeoff Dialog Box
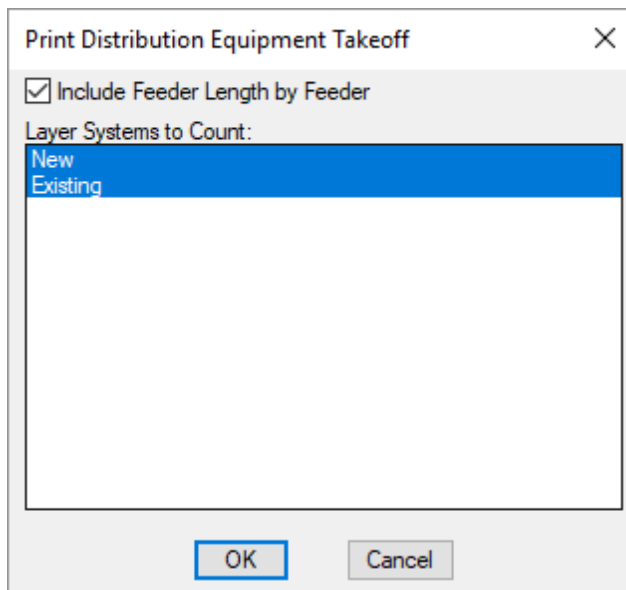
See the **Common Takeoff Information** section for more information about how this dialog box works.

Press the **OK** button. The takeoff will be generated as an HTML file and displayed in your web browser. The distribution equipment is sorted and grouped by type. Examples of distribution equipment takeoff information and the format in which the information is displayed are provided below.

Panels

Panels

H1

Volts: 480Y/277V 3P 4W
Bus Amps: 400
Main Disconnect Type: Breaker
Main Disconnect Amps: 400
Mounting: Flush
Neutral: 100%
Number of Poles: 42
Lugs: Standard
AIC Rating: 22,000
Feeder Size: 3-1/2"C,3#600kcmil,#600kcmilN,#2G
Feeder Length: 20'-10" (Calculated)

Breakers	
20/1	2
60/3	1
Unused Spaces	38

Transformers

Transformers

T1

KVA: 45
Primary Volts: 480Y/277V 3P 4W
Secondary Volts: 208Y/120V 3P 4W
Lugs: Standard
AIC Rating: 22,000
Feeder Size: 1-1/4"C,3#4,#4N,#10G
Feeder Length: 21'-4" (Calculated)

Breaker Totals

Breaker Totals

Less than 300V	
22,000 AIC	
20/1	8
30/3	1
Greater than 300V	
22,000 AIC	
60/3	1

Breakers are listed twice: once with the device with which they are associated, and again in a separate section that totals all of the breakers in the project. The breaker quantities do not include main disconnect breakers, which are listed separately as part of the distribution equipment description. The project totals are grouped by voltage and AIC rating.

Feeder Lengths

Feeders Lengths	
Calculated	
1-1/4"C,3#4,#4N,#10G	22'
2"C,3#3/0,#3/0N,#6G	18'
2-1/2"C,3#4/0,#4/0N,#4G	15'
3-1/2"C,3#600kcmil,#600kcmilN,#2G	21'

Feeders are also listed multiple times: with the equipment with which they are associated, in a total for the project, and by feeder if *Feeder Length by Feeder* was checked. The length of the feeder serving each equipment is listed with the equipment. The feeder length is labeled as either being **Calculated** or **Fixed**. Feeder lengths labeled **Calculated** were calculated based upon the location of the distribution equipment on the drawings. Feeder lengths labeled **Fixed** were input directly by the user and are not based upon the drawings.

Related Options

Feeder make-up: Sets additional length to be added to feeders for connections.

This option can be further customized with the **Ignore make-up on feeders shorter than this** option.

Display neutral wire count separately from phase wire count: Sets how the neutral wire is displayed in wire callouts.

Conduit location: Sets how the conduit location is displayed in wire callouts.

Print Equipment in Database Only Takeoff

To perform a takeoff of the equipment that is in the project database but not on a drawing, go to

Ribbon: `DM Electrical->Takeoffs->` EQU `Print Equipment in Database Only Takeoff`

Pulldown Menu: `DM Electrical->Takeoffs->Print Equipment in Database Only Takeoff`

Print Equipment in Database Only Takeoff Dialog Box

See the **Common Takeoff Information** section for more information about how this dialog box works.

Press the **OK** button. The takeoff will be generated as an HTML file and displayed in your web browser. This takeoff will list only equipment that is stored in the database but not shown on the drawing. An example takeoff is shown below.

Equipment: Mechanical	
RTU-1 Volts: 480V 3P 4W Ground: Yes Isolated Ground: No Load: 31.5 kVA Disconnect Type: Fused	6
RTU-2, RTU-5 Volts: 480V 3P 4W Ground: No Isolated Ground: No Load: 18.3 kVA Disconnect Type: Fused	5
RTU-3, RTU-4, RTU-6 Volts: 480V 3P 4W Ground: No Isolated Ground: No Load: 19.9 kVA Disconnect Type: Fused	9

Each distinct equipment connection is listed in a separate row. The description of the equipment connection is listed on the left. The number of equipment connections matching that description is listed on the right.

Lighting Groups

This section describes lighting groups, which are used for energy calculations.

Common Lighting Group Information

The values that can be specified for lighting groups are listed below.

Subtract Area from Room Area: Whether the area occupied by the lights in this lighting group is subtracted from the room area. This will affect energy calculations.

Height: The height of the lighting group, if it is a vertical area. See the **Distance Input Format** section for more information about specifying this height.

LF Mounting Height: The height at which the light fixtures in the lighting group are mounted. See the **Distance Input Format** section for more information about specifying this height.

LF Throw Distance: The distance from light fixtures to the lighting group area. See the **Distance Input Format** section for more information about specifying this distance.

Lighting Power Density: The amount of light allowed for the area.

Illuminance Category: The illuminance category for the group as defined in the *IESNA Lighting Handbook*. The categories are based upon the type of task being performed in the area. Lower letters correspond to lower foot-candle levels. Higher letters correspond to higher foot-candle levels.
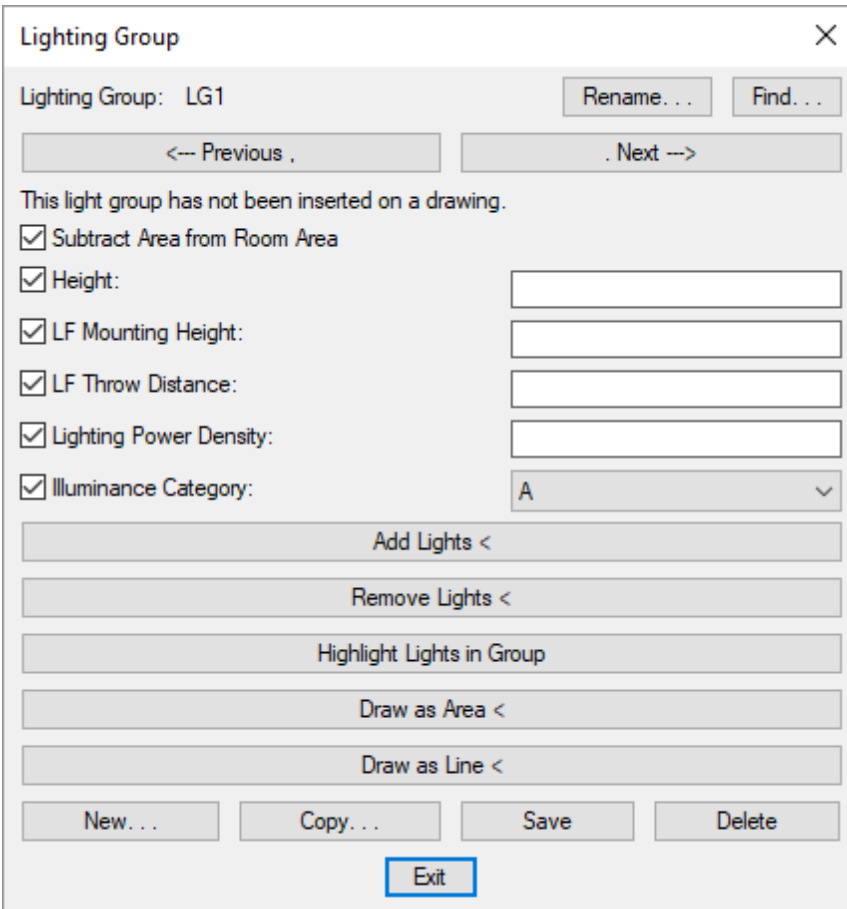
Create and Edit Lighting Groups

To create or edit a lighting group, go to

Ribbon: DM Electrical->Lighting Groups-> Create and Edit Lighting Groups

Pulldown Menu: DM Electrical->Lighting Groups->Create and Edit Lighting Groups

Lighting Group Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, New**, **Copy**, **Save**, and **Delete** buttons.

Check the box beside a setting to change it. See the **Common Lighting Group Information** section for more information about the lighting group settings.

Add Lights: See the **Add Lights To Group** section for more information.

Remove Lights: See the **Remove Lights from Groups** section for more information.

Highlight Lights in Group: See the **Highlight Lights in Group** section for more information.

Draw as Area: See the **Draw Lighting Group** section for more information.

Draw as Line: See the **Draw Lighting Group** section for more information.

Update Lighting Group Labels

To update lighting group labels or insert/update the lighting group table, go to

Ribbon: `DM Electrical->Lighting Groups->` ♻ `Update Lighting Group Labels`

Pulldown Menu: `DM Electrical->Lighting Groups->Update Lighting Group Labels`

Inserting the Lighting Group Table the First Time

If the lighting group table is not currently inserted on the drawing, you will be prompted for the insertion location of the table.

Insertion Point for Lighting Group Table:

The location you specify will be used as the top-left corner for the lighting group table.

Updating the Table Already on the Drawing

If the lighting group table is already inserted on the drawing, it will be updated in its current location.

You must update the table on the drawing when you make changes to the lighting groups. The table will not update automatically.

The lighting group table will also be updated when the **Update All Schedules and One-Line Diagram** command is run.

If you have manually inserted graphics, such as revision clouds, over the schedule graphic, be sure to check their location after updating the table.

Add Lights To Group

To add individual light fixtures to a lighting group, go to

Ribbon: `DM Electrical->Lighting Groups->` `Add Lights to Group`

Pulldown Menu: `DM Electrical->Lighting Groups->Add Lights to Group`

Select Lighting Group Dialog Box

Select the **Lighting Group** to which light fixtures are to be added and press the **OK** button. You will be prompted to identify which light fixtures on the drawing are to be added to the lighting group.

Select objects:

The light fixtures selected will be added to the selected lighting group and the appropriate lighting group label will be added next to the added light fixtures.

Related Options

Lighting group block: Sets the blocks used for lighting group labels.

Remove Lights from Groups

To remove individual light fixtures from a lighting group, go to

Ribbon: DM Electrical->Lighting Groups-> Remove Lights from Groups

Pulldown Menu: DM Electrical->Lighting Groups->Remove Lights from Groups

You will be prompted to identify which light fixtures are to be removed from a lighting group.

Select objects:

The selected light fixtures will be removed from the lighting group and the lighting group labels will be removed from the light fixtures.

Highlight Lights in Group

To highlight light fixtures in a particular lighting group, go to

Ribbon: DM Electrical->Lighting Groups-> Highlight Lights in Group

Select Lighting Group Dialog Box

Select the *Lighting Group* to be highlighted and press the **OK** button. All of the light fixtures in the selected lighting group will be highlighted.

To remove the highlights, use the standard CAD **REGEN** command.

Draw Lighting Group

To draw the area encompassing a lighting group for energy calculations, go to

Ribbon: `DM Electrical->Lighting Groups->` Draw Lighting Group

Select Lighting Group to Draw Dialog Box
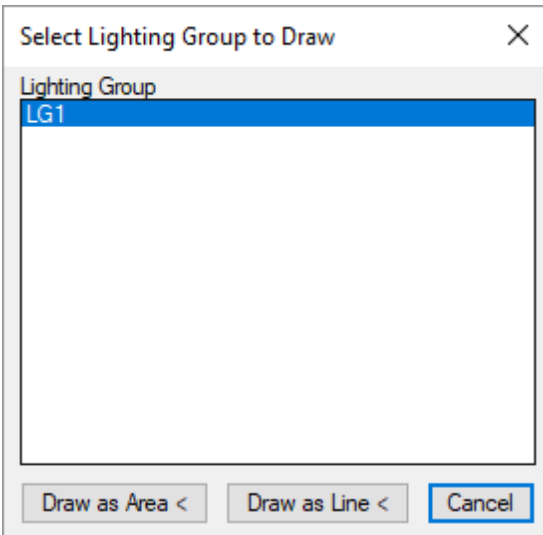
Select the *Lighting Group* for which an area is to be drawn.

Drawing Lighting Group as an Area

Press the **Draw as Area** button to draw an area encompassing the selected lighting group. You will be prompted to identify the location of the first point on the drawing.

Specify first lighting group point:

You then will be prompted to specify additional points.

Specify next lighting group point:

Continue inserting points until the area is defined. Press **ENTER** to complete the area. The area will be displayed on the drawing. You will be prompted to specify the insertion point for the lighting group area label on the drawing.

Specify lighting group info location:

The lighting group area label will be inserted at the specified location on the drawing.

Drawing Lighting Group as a Line

Press the **Draw as Line** button to indicate the area by tracing the length of a wall and entering the height. You will be prompted to identify the location of the first point on the drawing.

Specify first lighting group point:

You then will be prompted to specify additional points.

Specify next lighting group point:

Continue inserting points to indicate the length of the wall or walls. Press **ENTER** to complete the line. The line will

363

be displayed on the drawing. You will be prompted to specify the insertion point for the lighting group area label on the drawing.

Specify lighting group info location:

The lighting group area label will be inserted at the specified location on the drawing.

Insert Lighting Group Vertex

To add a vertex to a drawn lighting group area, go to

Ribbon: `DM Electrical->Lighting Groups->` Insert Lighting Group Vertex

Pulldown Menu: `DM Electrical->Lighting Groups->Insert Lighting Group Vertex`

You will be prompted to locate a point either inside or outside of the lighting group area onto which a vertex is to be inserted.

First fence point crossing lighting group boundary:

You will then be prompted to provide a point on the opposite side of the lighting group boundary. The line between the two points should cross the boundary where you want the new point to be inserted.

Second fence point:

A new vertex will be inserted on the boundary. The appearance of the outline will not change, but you will have two new grips that you can use to drag the area into the correct shape.

Remove Lighting Group Vertex

To remove a vertex from a lighting group area boundary, go to

Ribbon: `DM Electrical->Lighting Groups->` Remove Lighting Group Vertex

Pulldown Menu: `DM Electrical->Lighting Groups->Remove Lighting Group Vertex`

You will be prompted to draw a crossing box around the lighting group boundary vertex to remove.

First crossing point around lighting group vertex to remove:

You will then be prompted to specify the opposite corner of the crossing box.

Second crossing point:

The vertex on the lighting group boundary that is surrounded by the box will be removed.

Show or Hide Label Leaders

To show or hide lighting group area label leaders, go to

Ribbon: `DM Electrical->Lighting Groups->` ⬑ `Show or Hide Label Leaders`

Pulldown Menu: `DM Electrical->Lighting Groups->Show or Hide Label Leaders`

You will be prompted to select one or more lighting group boundaries or labels for which the leaders will be shown or hidden.

Select objects:

Label leaders will be shown if they were not displayed previously or hidden if they were shown previously.

Turn Lighting Group Layers On

To turn on all lighting group layers so that lighting group area boundaries and labels are displayed on the drawing, go to

Ribbon: `DM Electrical->Lighting Groups->` 🟨 `Turn Lighting Group Layers On`

Pulldown Menu: `DM Electrical->Lighting Groups->Turn Lighting Group Layers On`

The layers that the lighting group area boundaries and labels are inserted on will be turned on.

By default, these layers are non-plotting and do not have to be turned off before plotting.

Turn Lighting Group Layers Off

To turn off all lighting group layers so that lighting group area boundaries and labels are not displayed on the drawing, go to

Ribbon: `DM Electrical->Lighting Groups->` ⬜ `Turn Lighting Group Layers Off`

Pulldown Menu: `DM Electrical->Lighting Groups->Turn Lighting Group Layers Off`

The layers that the lighting group area boundaries and labels are inserted on will be turned off.

By default, these layers are non-plotting and do not have to be turned off before plotting.

Utilities

This section describes the utility commands that are available to assist in maintaining the integrity of your project database and drawings.

Coordinate Drawings and Database

The `Coordinate Drawings and Database` command is used to compare the project database with one or more drawings and confirm that the information matches.

To coordinate a set of selected drawings with the database, go to

Ribbon: `DM Electrical->Utilities->` `->Coordinate Drawings and Database`

Select Drawings Dialog Box
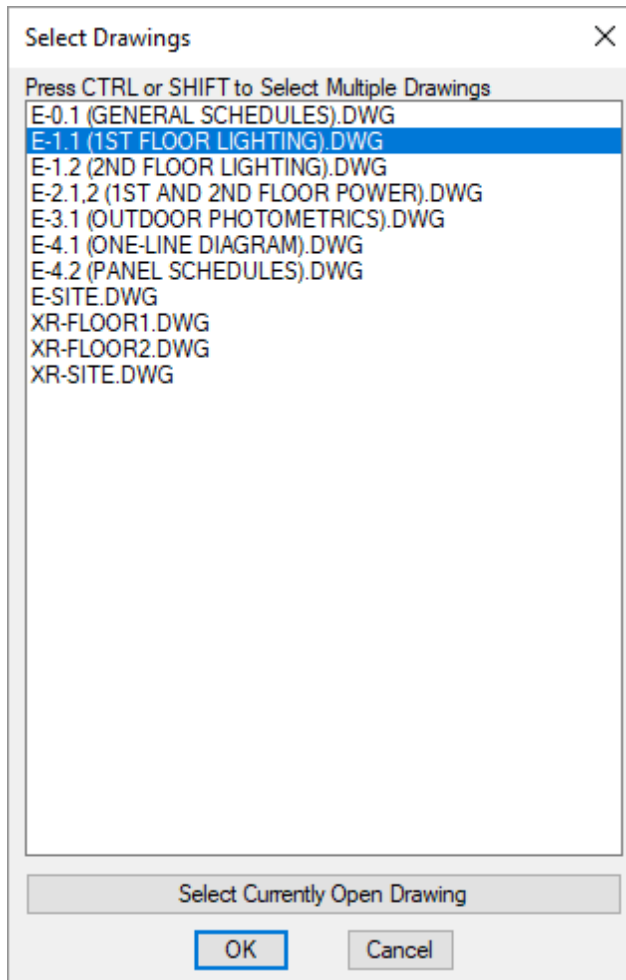
Drawing List: Select the drawings you want to coordinate with the database. The current drawing is selected when the command is first run. Use the **SHIFT** and **CTRL** keys to select multiple drawings in the list. All of the selected drawings will be coordinated with the database.

Select Currently Open Drawing: Press this button to select the current drawing.

Press the **OK** button to coordinate the selected drawings with the database.

The following actions will be performed:
- Devices on the drawing will be added to the database.
- Devices in the database will be added to the drawing. If the command causes a device to be inserted directly on top of another device, the new device will be inserted at the edge of the drawing. Any devices inserted from the database will have a line drawn to them. Review the newly inserted devices and move, delete, or otherwise modify them as appropriate. Once they are on the drawing, you can treat them like standard devices.
- Devices in both the drawing and the database will be reviewed to ensure their locations and settings

match.
- Any changes to scheduled devices, such as new graphics for a device type, will be made on the drawing.

When this command is finished, the drawing and the database will match. All devices on the drawing will accurately reflect the information stored in the database.

Devices reinserted on the drawing from the database will be highlighted with lines from the origin to their insertion point. Review these devices to determine whether they should be kept. Use the standard CAD `ERASE` command to remove excess devices from the drawing.

The coordination lines can be removed either by using the standard CAD `ERASE` command and selecting the coordination lines, or by using the **Erase Coordination Lines** command.

When to Use the Coordinate Drawings and Database Command

When you make changes to a drawing, these changes are immediately recorded in the database. Changes are not recorded on the drawing until it is saved. If your CAD program crashes before the drawing is saved, the database will have newer and better information than the drawing. Use the `Coordinate Drawings and Database` command to match the drawing and database, and you can often recover information that was lost.

Changes to customization settings are not immediately reflected on the drawings. If you change your customization settings in the middle of a project, use the `Coordinate Drawings and Database` command to update the drawings.

There are other times when this command can fix errors in your project. A backup of the current drawing and database is made before the command is run in case something unexpected happens during the command.

Erase Coordination Lines

To erase coordination lines generated when using the **Coordinate Drawings and Database** command, go to

Ribbon: `DM Electrical->Utilities->  ->Erase Coordination Lines`

Pulldown Menu: `DM Electrical->Utilities->Erase Coordination Lines`

The coordination lines will be erased from the current drawing.

Coordination lines may also be erased using the standard CAD `ERASE` command.

Check for Drawings to Update

The `Check for Drawings to Update` command is used to check for and update any drawings that need to have changes made to them. Certain changes made on one drawing in a project will require changes to be made to other drawings in the project. Typically, these changes happen automatically.

This command will check for any drawings that need to have changes made and give you a chance to update them all.

To check if any drawings in the current project need to be updated, go to

Ribbon: `DM Electrical->Utilities->  ->Check for Drawings to Update`

Pulldown Menu: `DM Electrical->Utilities->Check for Drawings to Update`

The drawings that need to be updated will be identified. You will be given a chance to update them automatically if there are any.

Calculate All Circuit Loads

The `Calculate All Circuit Loads` command is used to recalculate all of the loads on all of the circuits on all of the distribution equipment in the project. It guarantees that all of the loads shown in distribution equipment schedules are correct.

To recalculate all circuit loads, go to

Ribbon: `DM Electrical->Utilities->` ^{UTIL} `->Calculate All Circuit Loads`

Pulldown Menu: `DM Electrical->Utilities->Calculate All Circuit Loads`

All of the loads on all of the circuits in the project will be reset to 0. The load on each equipment will then be added to the circuit it is connected to. Once all of the branch circuits are updated, the loads will be transferred between distribution equipment, starting from the bottom equipment and working up toward the utility.

Generally, this command does not need to be run. Circuit loads are updated automatically as you modify your design. Recalculating all of the loads can take a long time, especially on large projects.

You can use the **Recalculate Selected Circuit** and **Recalculate All Circuits** buttons on the **Circuiting** dialog box to recalculate smaller portions of the project. See the **Circuiting** section for more information.

Delete Extra Devices from Database

The `Delete Extra Devices From Database` command is used to delete devices from the database that are on drawings that no longer exist in the project.

Normally, you would use the standard CAD `ERASE` command to erase devices. They will be erased from the drawing and database.

If a drawing is moved or deleted before all of the devices are erased, the records still exist in the database. You cannot erase the device from the drawing because it no longer exists. Using this command, these devices can be deleted from the database.

To delete any devices that exist in the database that either do not have an associated drawing or are on a drawing that no longer exists, go to

Ribbon: `DM Electrical->Utilities->` `->Delete Extra Devices From Database`

Pulldown Menu: `DM Electrical->Utilities->Delete Extra Devices From Database`

The drawing that each device is on will be compared with the list of drawings in the project. If a drawing in the database does not exist, the **Delete devices?** dialog box will appear.
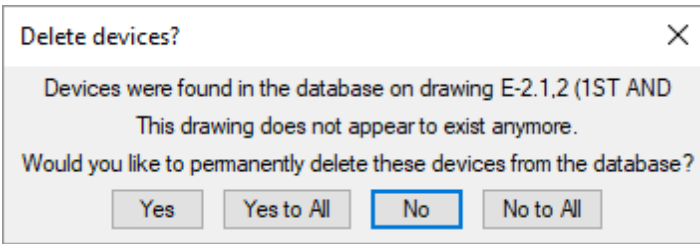
Yes: Press this button to delete the devices from the listed drawing.

Yes to All: Press this button to delete the devices from all of the drawings that are not found.

No: Press this button to not delete the devices from the listed drawing. Use this button if the drawing should exist and you plan to recreate it.

No to All: Press this button to not delete the devices from all of the drawings that are not found.

Rename Database

The `Rename Database` command is used to change the name of the database file associated with the current drawing. It will change the suffix of the file when viewed in Windows Explorer. Nothing else in the project will be changed.

To rename the database, go to

Ribbon: `DM Electrical->Utilities->  ->Rename Database`

Pulldown Menu: `DM Electrical->Utilities->Rename Database`

Rename Electrical Database Dialog Box
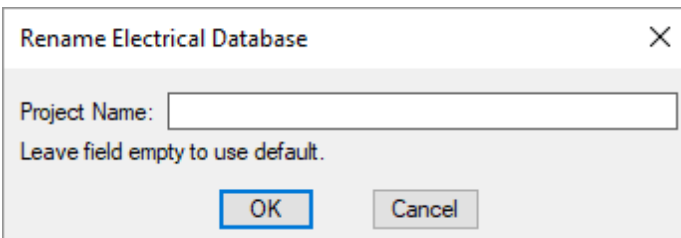
Project Name: The new name of the database.

Enter the new *Project Name* and press the **OK** button. The suffix of the database file associated with the current drawing will be changed. No other changes will be made in the project.

Repair Database

The `Repair Database` command is used to attempt to repair a project database that has become unusable due to crashes, multiple errors, or other irregularities.

This command can take a long time to run and is intended to be used as a last resort before restoring the database

from a backup. Only run this command if you have been instructed to do so by Design Master support.

To repair the database, go to

Ribbon: `DM Electrical->Utilities->` ᵁᵀᴵᴸ `->Repair Database`

Pulldown Menu: `DM Electrical->Utilities->Repair Database`

A **Design Master** dialog box will appear that briefly explains the process and possible outcomes.
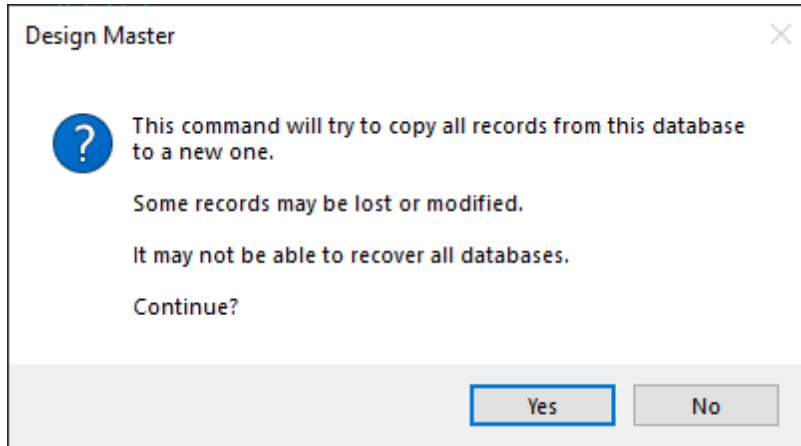
Press the **Yes** button to attempt the repair.

Delete DM Backups Over 30 Days Old

The `Delete DM Backups Over 30 Days Old` command is used to delete files from the DMBackup folder. Backups are made on a regular basis for projects and when certain utility commands are run. When a project is opened, any backups over 30 days old are automatically deleted. However, if a project is not opened, backups over 30 days old will not be deleted.

This command will erase all of the backups that it finds in any subfolders of the selected folder. If you have a folder that contains all your projects, you can select the main folder and have all backups from all projects deleted.

Backups less than 30 days old will not be deleted. You must use Windows Explorer to delete these folders.

To delete backup files that are over 30 days old, go to

Ribbon: `DM Electrical->Utilities->` `->Delete DM Backups Over 30 Days Old`

Pulldown Menu: `DM Electrical->Utilities->Delete DM Backups Over 30 Days Old`

Select Folder Dialog Box
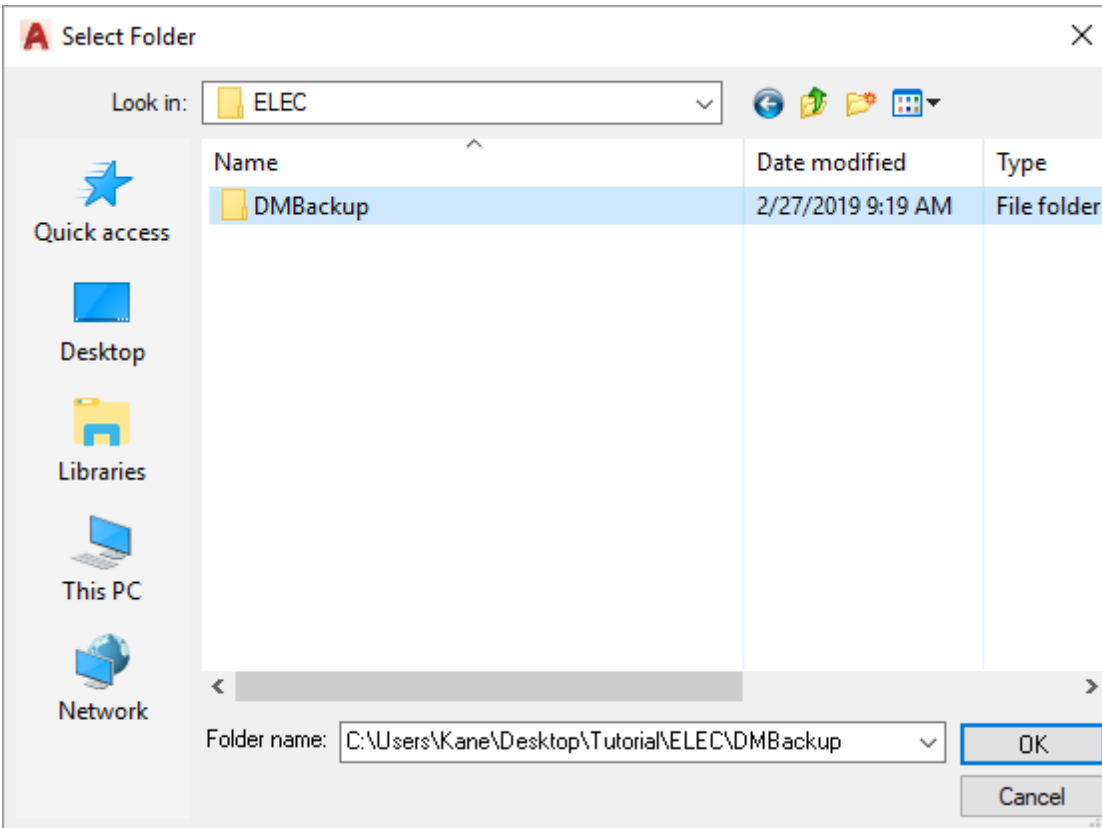
Select the folder to delete backups from. All backups over 30 days old will be deleted.

Find Lost Toolbars

The `Find Lost Toolbars` command is used to move all of the toolbars to a visible location on the screen. If your CAD program moves the toolbars to a location that cannot be seen, the toolbars are made unusable. Moving all of the toolbars is the simplest solution to this problem.

To find lost toolbars, go to

Ribbon: `DM Electrical->Utilities->  ->Find Lost Toolbars`

Pulldown Menu: `DM Electrical->Utilities->Find Lost Toolbars`

Once this command is entered, all of the toolbars will be displayed. Move any toolbars you want visible to the correct location. Close any toolbars that you do not use.

Reload Ribbons

The `Reload Ribbons` command is used to reload the Design Master Electrical ribbon tabs. Use this command if the tabs have disappeared from your ribbon.

To reload the ribbon tabs, go to

Pulldown Menu: `DM Electrical->Utilities->Reload Ribbons`

Once this command is run, the ribbon tabs should be reloaded and visible on your ribbon.

Update Breaker Database

The `Update Breaker Database` command is used to update the database of breaker and fuse curves used for selective coordination. This command guarantees you will have the most complete list of breakers and fuses.

To update the breaker database, go to

Ribbon: `DM Electrical->Utilities->` UTIL `->Update Breaker Database`

Pulldown Menu: `DM Electrical->Utilities->Update Breaker Database`

The breaker database will be updated. A **Design Master Electrical** dialog box will appear confirming the update was successful.
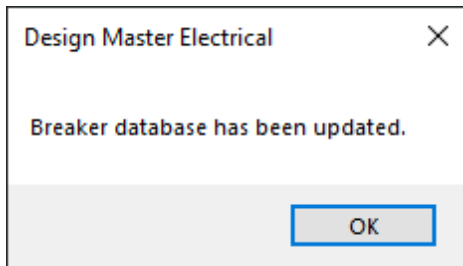
Generally, this command does not need to be run. The breaker database is automatically updated once each day that a command attempts to access it. You should only need to run this command if you need immediate access to curves that have been added to the breaker database since the last automatic update.

Delete Drawing

The `Delete Drawing` command is used to delete a drawing that contains Design Master Electrical devices and remove all of the associated information from the database.

If you delete a drawing using Windows Explorer, the database will not be updated. Any devices that existed on the drawing will still exist in the database. It is possible to delete these devices using the **Delete Extra Devices from Database** command, but this method can easily create larger problems.

Deleting the drawing using this command will make sure that the necessary changes to the database are made.

You do not need to use this command if the drawing does not contain Design Master Electrical devices, though there is no harm if you do.

To delete a drawing, go to

Ribbon: `DM Electrical->Utilities->` `->Delete Drawing`

Pulldown Menu: `DM Electrical->Utilities->Delete Drawing`

Select Drawing to Delete Dialog Box
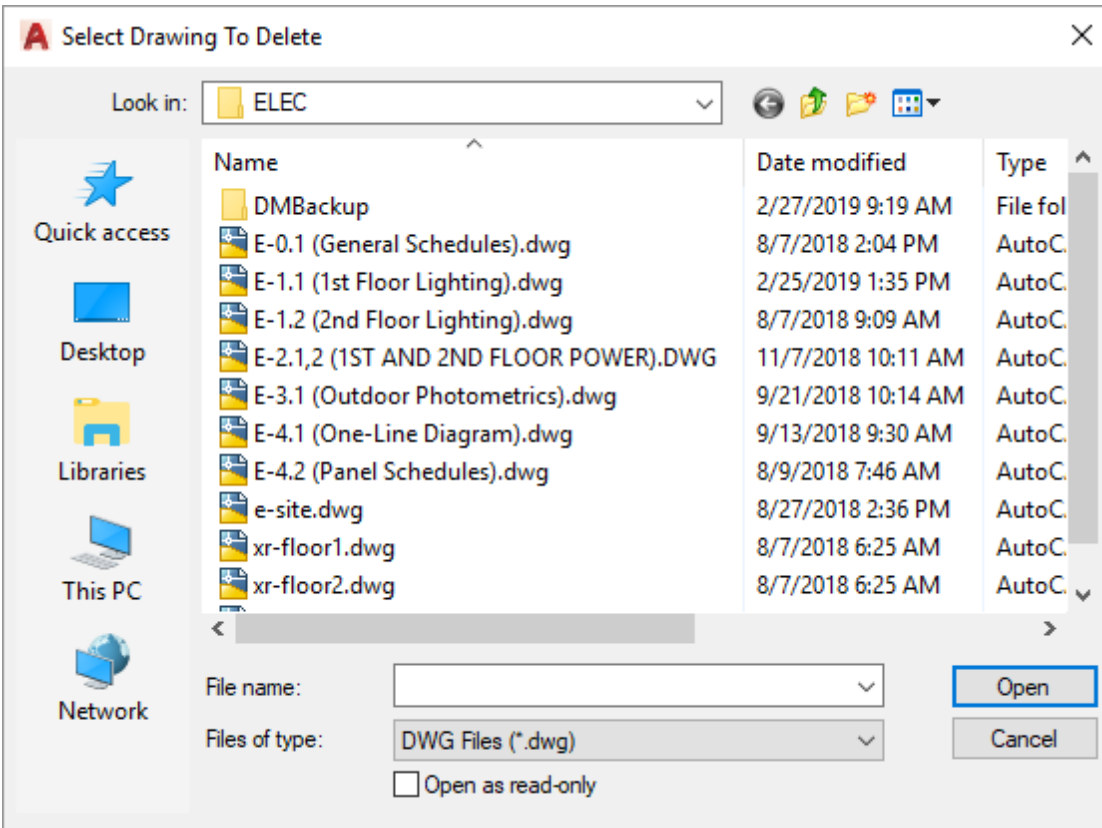
Select the drawing to delete and press the **Open** button.

The drawing will be deleted. All of the devices on the drawing will be deleted from the database.

A backup of the drawing and the database will be made before the command is run in case a drawing is deleted accidentally.

Rename Drawing

The `Rename Drawing` command is used to rename the current drawing. The database will be updated so that any devices associated with the drawing will have their drawing location information changed.

If you rename a drawing using Windows Explorer or the standard CAD `SAVEAS` command, the database does not know if you intend to copy or rename the drawing. It is very easy to create significant errors in your drawings and database if you choose the wrong option.

Renaming the drawing using this command will prevent these errors from happening.

To rename the current drawing, go to

Ribbon: `DM Electrical->Utilities->  ->Rename Drawing`

Pulldown Menu: `DM Electrical->Utilities->Rename Drawing`

Rename Drawing Dialog Box
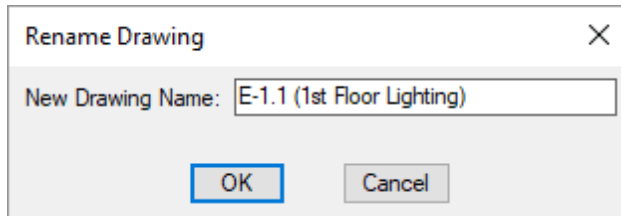
New Drawing Name: The name of the new drawing.

Enter the *New Drawing Name* into the dialog box and press the **OK** button. The name of the current drawing will be changed.

Copy Drawing

The `Copy Drawing` command is used to copy the current drawing. Copies of all of the devices on the drawing will be created in the database.

If you copy a drawing using Windows Explorer or the standard CAD `SAVEAS` command, the database does not know if you intend to copy or rename the drawing. It is very easy to create significant errors in your drawings and database if you choose the wrong option.

Copying the drawing using this command will prevent these errors from happening.

To copy the current drawing, go to

Ribbon: `DM Electrical->Utilities->  ->Copy Drawing`

Pulldown Menu: `DM Electrical->Utilities->Copy Drawing`

Copy Drawing Dialog Box
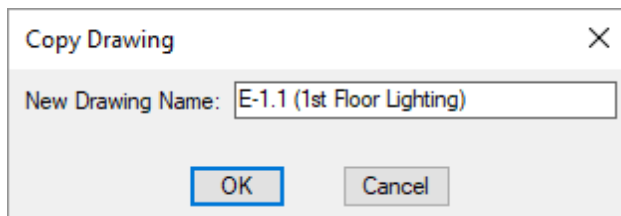
New Drawing Name: The name of the new drawing.

Enter the *New Drawing Name* into the dialog box and press the **OK** button. A copy of the current drawing will be made using the new name.

Copy Drawing and Remove Links to Database

The `Copy Drawing and Remove Links to Database` command is used to make a copy of a drawing and remove all of the links between the Design Master Electrical entities on the drawing and the database. The appearance of the drawing will not change. All entities on the drawing will remain. However, they will be

converted to standard CAD entities and their connection to the database will be removed.

To copy a drawing and remove all of the links to the database, go to

Ribbon: `DM Electrical->Utilities->  ->Copy Drawing and Remove Links to Database`

Pulldown Menu: `DM Electrical->Utilities->Copy Drawing and Remove Links to Database`

Save Drawing as Dialog Box
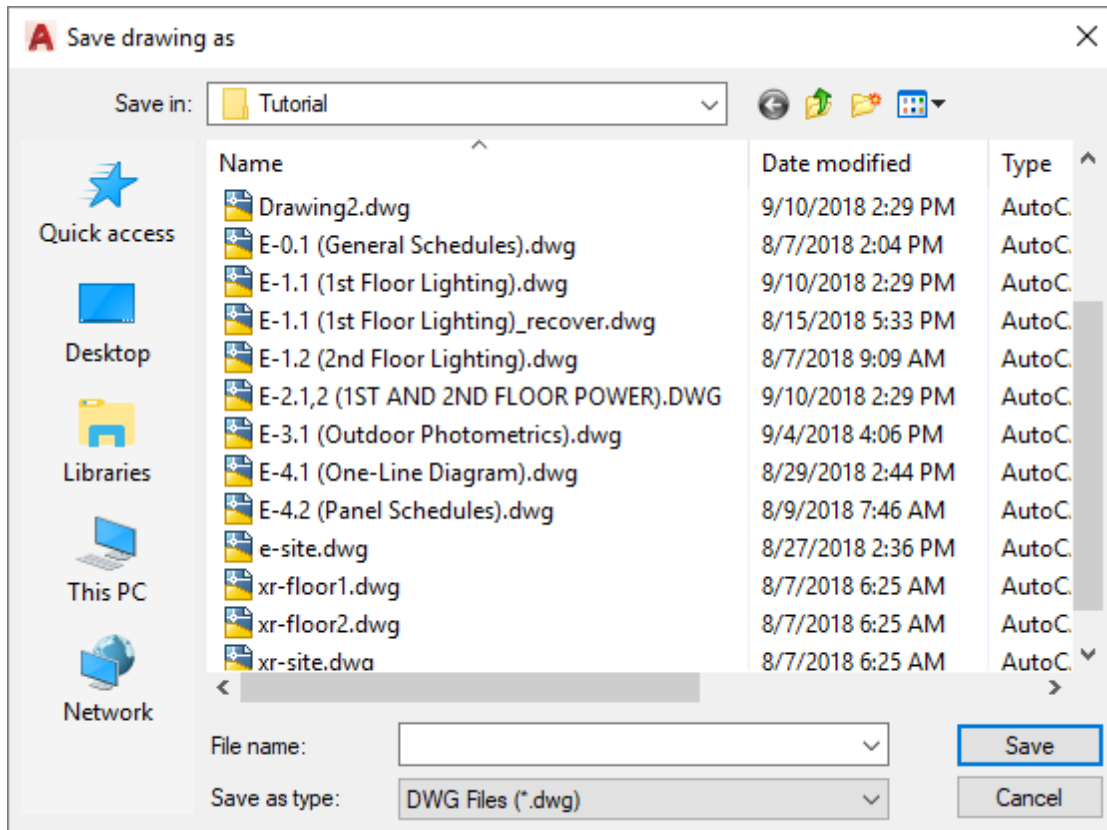
Select a location for the copy, enter a file name, and press the **OK** button. A copy of the current drawing will be made using the new name. All of the Design Master Electrical links to the project database will be removed.

The project database will not be modified during this command.

Copy Drawing and Remove All DM Entities

The `Copy Drawing and Remove All DM Entities` command is used to make a copy of a drawing with all of the Design Master Electrical entities on it erased.

This command is useful when you want to create a new drawing with the same settings as an existing drawing in the project. Using other methods, the copied drawing will copy the Design Master Electrical entities. Cleaning up these extra devices can cause problems if not done correctly. This command will remove all of the devices for you, leaving you with a new drawing that you can immediately start using.

To copy a drawing and remove all of the Design Master Electrical entities, go to

Ribbon: `DM Electrical->Utilities->  ->Copy Drawing and Remove All DM Entities`

Pulldown Menu: `DM Electrical->Utilities->Copy Drawing and Remove All DM Entities`

Save Drawing as Dialog Box
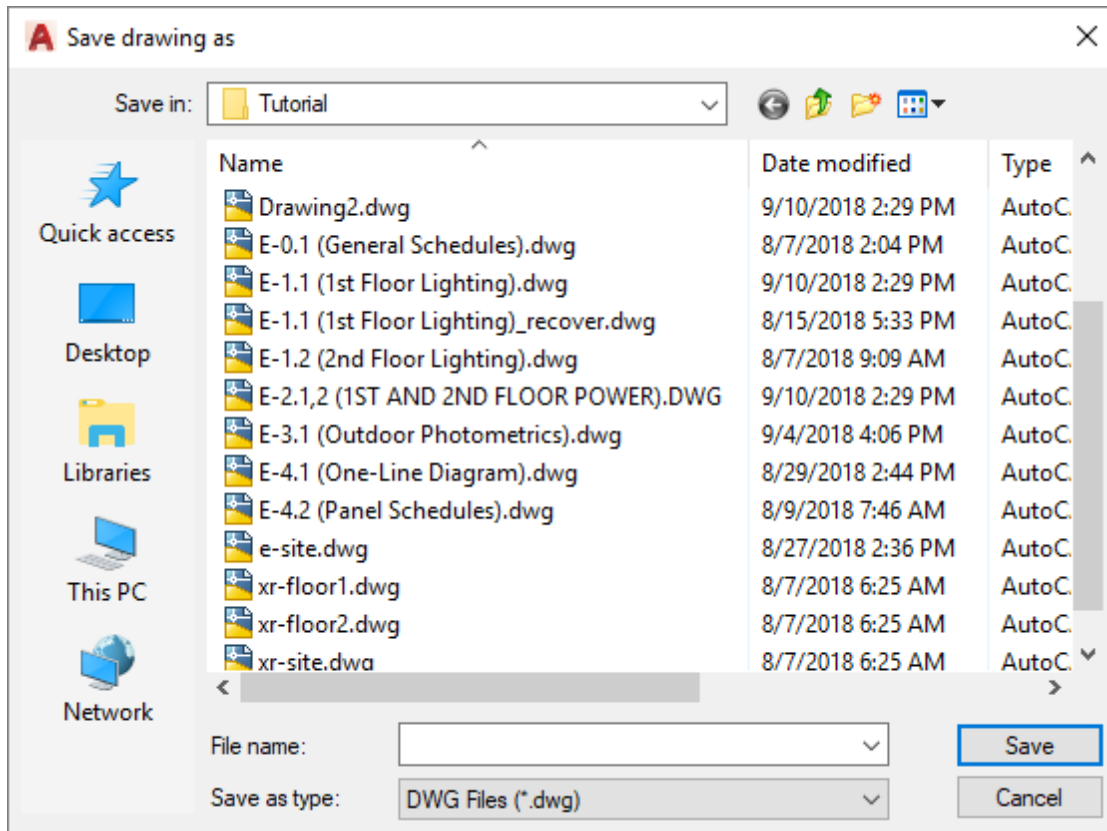
Select a location for the copy, enter a file name, and press the **OK** button. A copy of the current drawing will be made using the new name. All of the Design Master Electrical entities on the new drawing will be erased.

The project database will not be modified during this command.

Disconnect Electrical Database from Drawing

The `Disconnect Electrical Database from Drawing` command is used to remove all of the links between the Design Master Electrical entities on the drawing and the database. The appearance of the drawing will not change. All entities on the drawing will remain. However, they will be converted to standard CAD entities and their connection to the database will be removed.

To disconnect the database from the drawing, go to

Ribbon: `DM Electrical->Utilities->  ->Disconnect Electrical Database from Drawing`

Pulldown Menu: `DM Electrical->Utilities->Disconnect Electrical Database from Drawing`

The project database will not be modified during this command.

Merge Project

The `Merge Project` command is used to merge panels and drawings from another project into the current project.

To merge another project with the current project, go to

Ribbon: `DM Electrical->Utilities->` UTIL `->Merge Project`

Pulldown Menu: `DM Electrical->Utilities->Merge Project`

Select Electrical Project to Merge Box Dialog Box
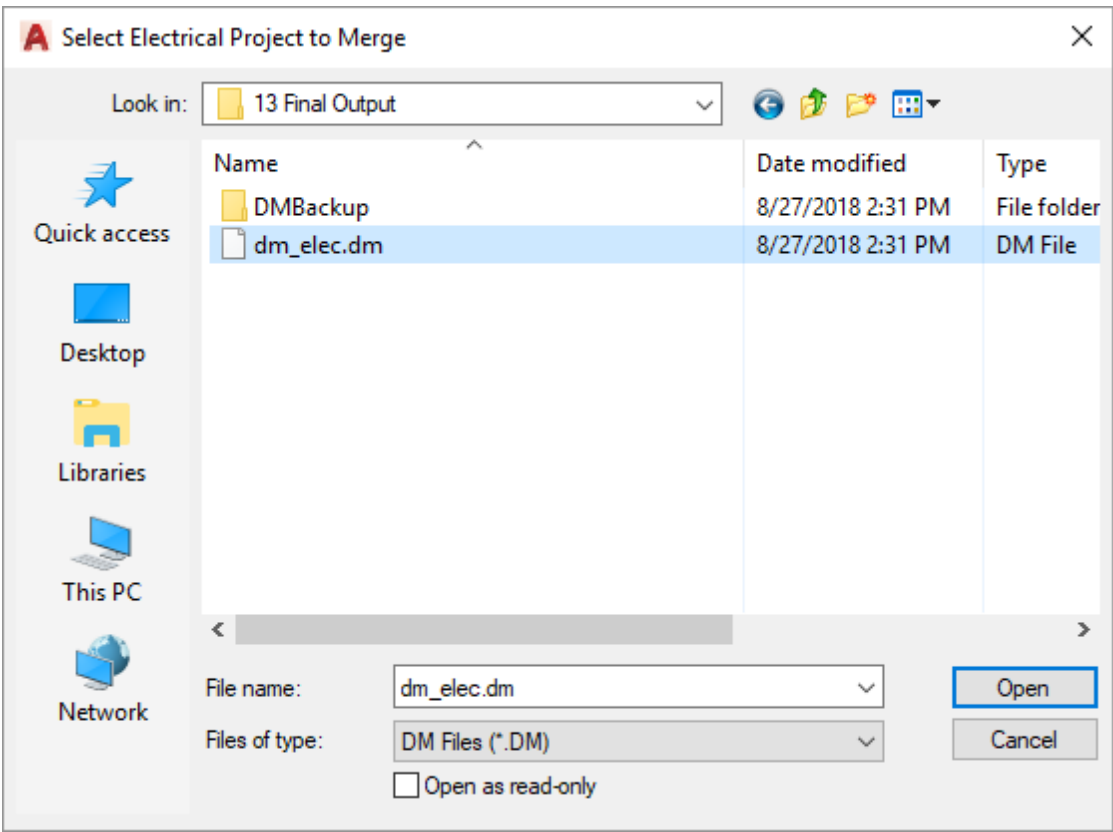
Browse to the project you want to merge with the current project, select the "dm_elec.dm" file, and press the **Open** button.

Merge Projects Dialog Box
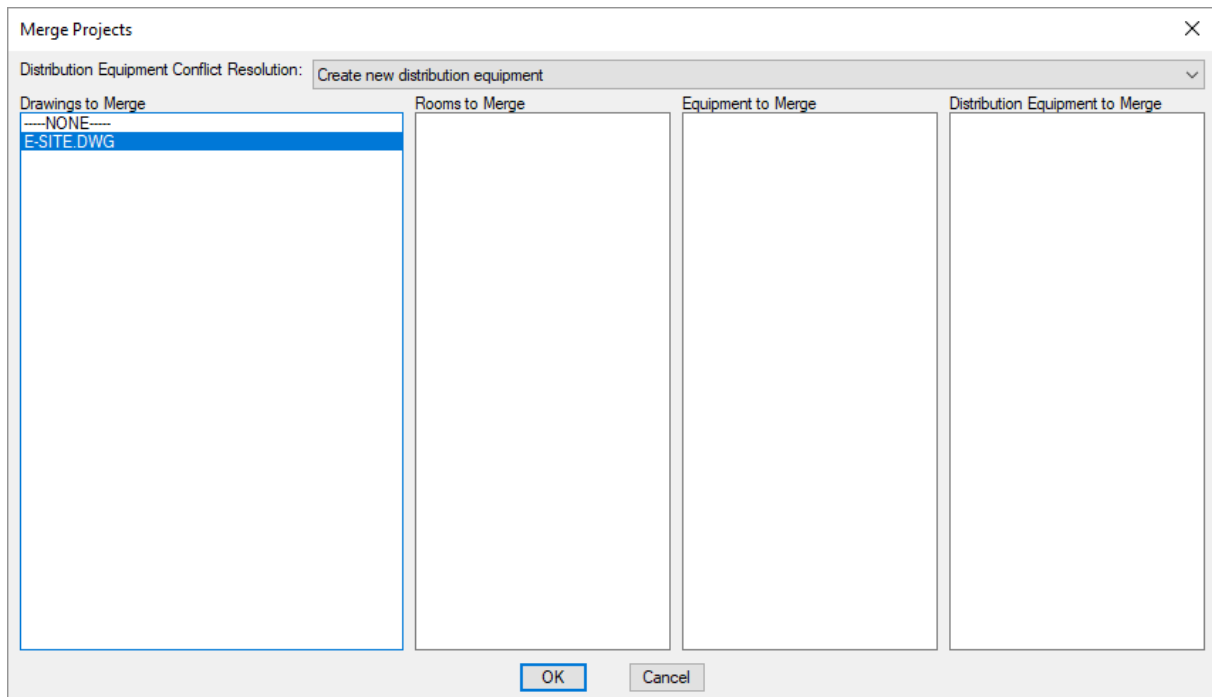
Distribution Equipment Conflict Resolution: How conflicts between distribution equipment with the same name will be handled when the projects are merged.

- **Create new distribution equipment:** The distribution equipment from the other project will be copied to the current project as a new distribution equipment with **(Copy X)** added to the callout. Devices connected to the distribution equipment in the other project will be connected to the new distribution equipment in the current project.
- **Merge circuits:** The distribution equipment from the other project will not be copied to the current project. All devices that are connected to the distribution equipment from the other project will be connected to the same circuit on the equipment in the current project. Any devices that are already connected to the equipment in the current project will stay connected.
- **Use first available space:** The distribution equipment from the other project will not be copied to the current project. All devices that are connected to the distribution equipment from the other project will be connected to the distribution equipment in the current project. If the circuit the device is connected to is a space, it will be connected to the same circuit in the current project. If the circuit the device is connected to already has devices connected to it in the current project, it will be connected to the next available space on the distribution equipment. If there are no free spaces available on the distribution equipment, a copy of the distribution equipment will be created and the devices will be connected to that.

Drawings to Merge: Select the drawings from the other project to merge with the current project. Use the `SHIFT` and `CTRL` keys to select multiple drawings. The selected drawings will be copied from the other project to the current project. All of the devices on the drawing will be updated to work correctly with the new project.

Schedule entries will be created for devices as appropriate. If a schedule entry with the same name exists in both projects, the two definitions will be compared. If any part of the definitions is different, a copy of the schedule entry will be created in the current project. The copied schedule entry will be used for devices that are merged from the other project.

If you select a drawing with the same name as a drawing already in the project, you will be asked what to do with the drawing from the other project.
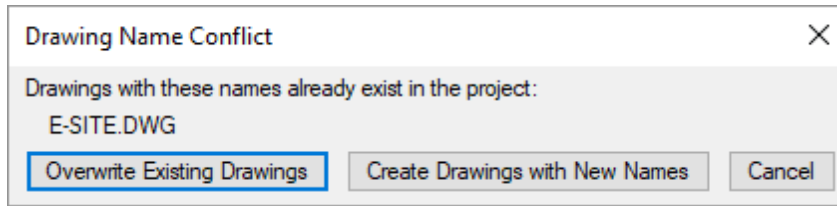
Overwrite Existing Drawings: Press this button to erase the drawing from the current project and replace it with the drawing from the other project. All of the devices on the drawing in the current project will be removed from the database.

Create Drawings with New Names: Press this button to create a drawing with a new name in the current project.

Rooms to Merge: Select the rooms from the other project to merge with the current project. Use the `SHIFT` and `CTRL` keys to select multiple rooms. Rooms that exist on the drawings selected in the *Drawings to Merge* list will be included in the merge even if they are not selected in this list.

Equipment to Merge: Select the equipment from the other project to merge with the current project. Use the `SHIFT` and `CTRL` keys to select multiple equipment. Only equipment that exists in the other project database but is not inserted on a drawing will be listed. Equipment that is on a drawing will be merged if the drawing is merged.

Distribution Equipment to Merge: Select the distribution equipment from the other project to merge with the current project. Use the `SHIFT` and `CTRL` keys to select multiple distribution equipment. Distribution equipment that is inserted on a drawing or connected to devices on a drawing selected in the *Drawings to Merge* list will be included in the merge even if they are not selected in this list.

Copy or Back Up Project

The `Copy or Back Up Project` command is used to make a complete copy or backup of the current project. When you copy or back up a project, it is important to copy all of the drawings and the Design Master database files. This command will copy all of the necessary files. Use this command if you want to make a backup of the current state of the project or if you want to make a copy to try a different design approach.

To copy or back up a project, go to

Ribbon: `DM Electrical->Utilities->  ->Copy or Back Up Project`

Pulldown Menu: `DM Electrical->Utilities->Copy or Back Up Project`

Select Folder to Copy or Back Up Project To Dialog Box
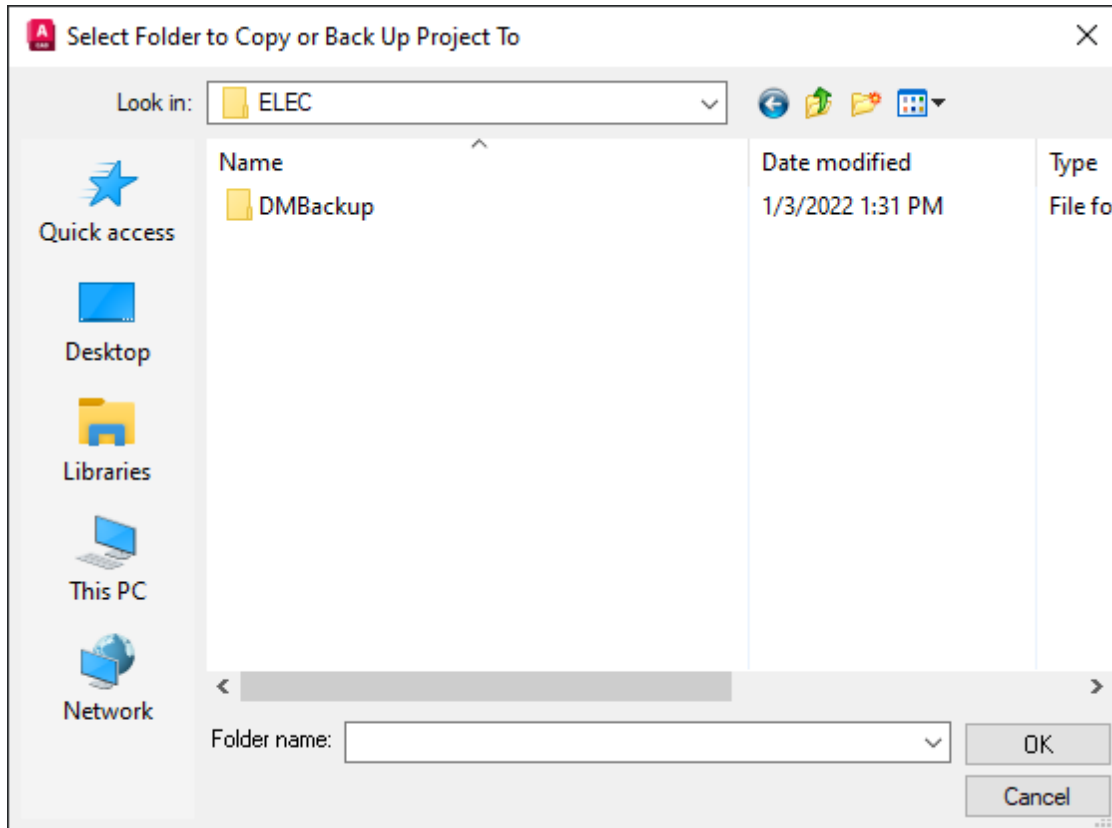
Select the folder to copy or back up the project to and press the **OK** button. All files associated with the project will be copied and saved to the folder.

Check In Floating License

The `Check In Floating License` command is used with floating licenses to immediately check in the license so that another person can use the software. Floating licenses are checked out in 10-minute intervals. If no commands are run for 10 minutes, the license is automatically released and another person can use the software.

If another person attempts to use the software before the license is released, a warning message will be displayed for the new user, stating that no license is available. The new user can choose to ignore the warning and immediately start using the software. However, a license overuse exception will be recorded on the license server. If there are too many overuse exceptions, you will need to purchase another license.

Using this command, the original user can check in the floating license when they are done with it. The new user can then use the software immediately without causing a license overuse exception to be recorded.

To check in the floating license, go to

Ribbon: `DM Electrical->Utilities->  ->Check In Floating License`

Pulldown Menu: `DM Electrical->Utilities->Check In Floating License`

The floating license will be checked in and will be available for another person to use.

Help

This section describes the commands available for obtaining support, maintaining current licenses, and installing software updates.

Common Help Information

You are prompted to provide the following information when sending us a support request. We use this information to respond to your support request.

Name: Your name.

Email Address: Your email address. If possible, we will respond to your support request by email.

Phone Number: Your phone number. If we need additional information beyond what we can gather by email, we will call you.

Additional Information: A description of the problem you are having. The more information you can provide about your problem, the more likely we will be able to solve it for you. For example, specific devices that exhibit the problem you are having.

Your Design Master Electrical version, CAD program version, and Windows version will be sent automatically with the support request. You do not need to include that information here.

User Manual

To open the Design Master Electrical User Manual (which you are currently viewing), go to

Ribbon: `DM Electrical->Utilities->  ->User Manual`

Pulldown Menu: `DM Electrical->Help->User Manual`

The user manual will be opened in your web browser.

Tutorial

To open the Design Master Electrical Tutorial, go to

Ribbon: `DM Electrical->Utilities->  ->Tutorial`

Pulldown Menu: `DM Electrical->Help->Tutorial`

The tutorial will be opened in your web browser. The tutorial provides instructions for and examples of common tasks that can be performed using our software.

Knowledge Base

To open the Design Master Electrical Knowledge Base, go to

Ribbon: `DM Electrical->Utilities->  ->Knowledge Base`

Pulldown Menu: `DM Electrical->Help->Knowledge Base`

The knowledge base will be opened in your web browser. This page has useful links and information to help you when you have trouble with our software.

Remote Support

The `Remote Support` command allows us to connect to your computer over the internet. We will be able to see your screen and perform actions on your computer.

To allow us to connect to your computer, go to

Ribbon: `DM Electrical->Utilities->  ->Remote Support`

Pulldown Menu: `DM Electrical->Help->Remote Support`

The remote support program will run and allow us to connect to your computer.

An ID number will be displayed, which you will need to read aloud to us.

We use TeamViewer for our remote support. You can visit their website at **http://www.teamviewer.com**. Information about security is available at **http://www.teamviewer.com/en/products/security.aspx**.

Send Project to Design Master Support
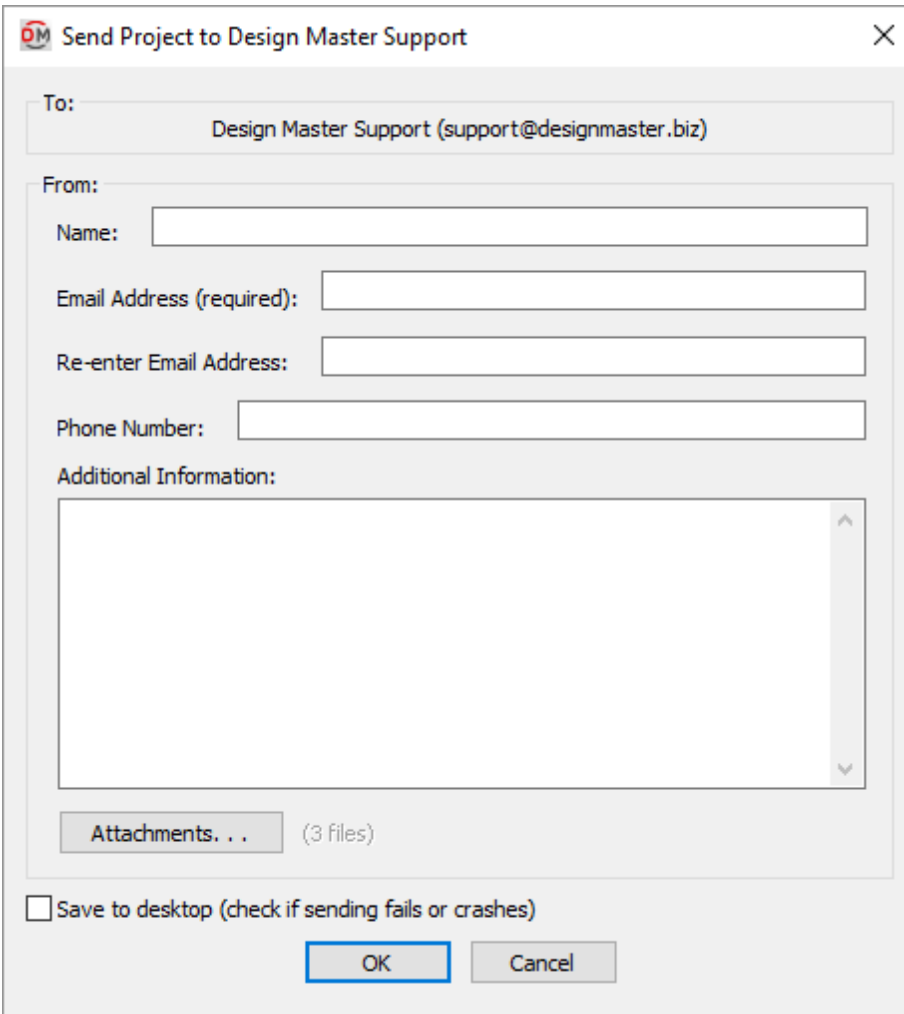
The `Send Project to Design Master Support` command is used to send us the current drawing and the associated Design Master Electrical database. Providing us with your project helps us recreate the problem you are having and find a solution to it. It is our preferred way for you to report a problem or ask a question. Many problems are specific to a project and having that immediately helps us assist you.

To send a copy of your project's current drawing and database to us, go to

Ribbon: `DM Electrical->Utilities->  ->Send Project to Design Master Support`

Pulldown Menu: `DM Electrical->Help->Send Project to Design Master Support`

Send Project to Design Master Support Dialog Box

See the **Common Help Information** section for more information about the *Name*, *Email Address*, *Phone Number* and *Additional Information* fields.

Attachments: Press this button to change the files that will be sent with the support request.

The **Design Master Attachments** dialog box will appear.
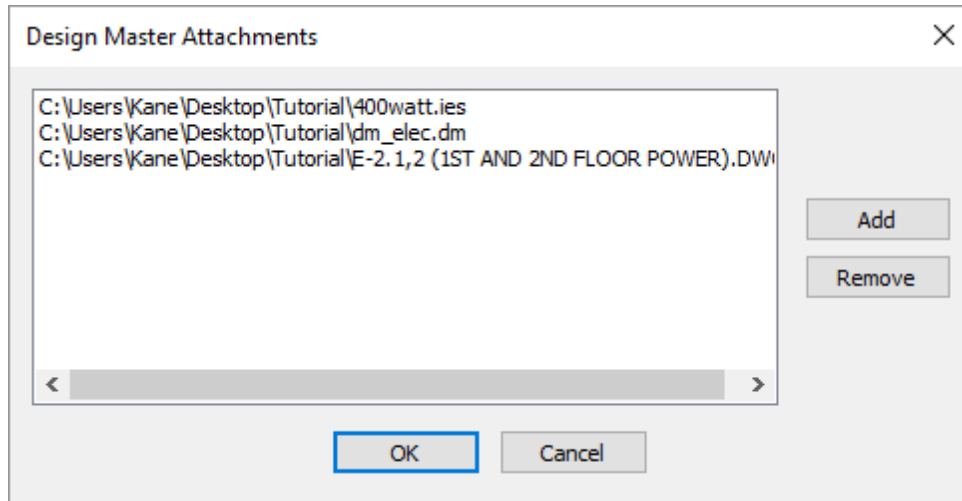
By default, the current drawing and associated Design Master Electrical database will be included in the list of files to send.

Add: Press this button to select additional files to send.

Remove: Press this button to remove the selected file from the list to send.

Save to desktop: Whether the support request is sent directly to us or saved to your desktop for you to email.

By default, this box is not checked. Support requests are automatically sent with no further input from you.

Some companies block outgoing messages from programs other than your email client. When this happens, you will be given a chance to save a ZIP file to your desktop that you can send to us manually. Detecting this configuration can take a long time.

When this box is checked, the attempt to send the message directly is skipped and you are immediately prompted to save a ZIP file to your desktop. Skipping the attempt to send saves time.

Go to Design Master Support Website

To open the Design Master Support Website, go to

Ribbon: `DM Electrical->Utilities->  ->Go to Design Master Support Website`

Pulldown Menu: `DM Electrical->Help->Go to Design Master Support Website`

The support page will be opened in your web browser. This page has useful links and information to help you when you have trouble with our software.

Send Email to Design Master Support

To send an email to us, go to

Ribbon: `DM Electrical->Utilities->  ->Send Email to Design Master Support`

Pulldown Menu: `DM Electrical->Help->Send Email to Design Master Support`

A draft email addressed to **support@designmaster.biz** will be opened in your preferred email client. Enter your message and send the email.

Send Master and Standards Databases to Design Master Support

The `Send Master and Standards Databases to Design Master Support` command is used to send us your master and standards customization database files. We will occasionally ask for these files when helping you solve problems related to your customization.

To send a copy of your master and standards databases to us, go to

Ribbon: `DM Electrical->Utilities->  ->Send Master and Standards Databases to Design Master Support`

Pulldown Menu: `DM Electrical->Help->Send Master and Standards Databases to Design Master Support`

Send Master and Standards Databases to Design Master Support Dialog Box

See the **Common Help Information** section for more information about the *Name, Email Address, Phone Number* and *Additional Information* fields.

Your master database and all of your standards databases will be sent.

Save to desktop: Whether the support request is sent directly to us or saved to your desktop for you to email.

By default, this box is not checked. Support requests are automatically sent with no further input from you.

Some companies block outgoing messages from programs other than your email client. When this happens, you will be given a chance to save a ZIP file to your desktop that you can send to us manually. Detecting this configuration can take a long time.

When this box is checked, the attempt to send the message directly is skipped and you are immediately prompted to save a ZIP file to your desktop. Skipping the attempt to send saves time.

Set Email Address
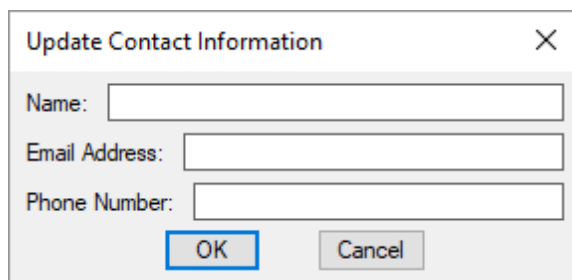
The Set Email Address command is used to set your default email address and other contact information used when sending support cases to us.

To set your default contact information, go to

Ribbon: `DM Electrical->Utilities->  ->Set Email Address`

Pulldown Menu: `DM Electrical->Help->Set Email Address`

Update Contact Information Dialog Box

See the **Common Help Information** section for more information about the *Name*, *Email Address* and *Phone Number* fields.

Installation Settings

The `Installation Settings` command is used to display the current installation settings. It can be used to verify the installation settings are correct, and modify them if necessary.

To view and modify the installation settings, go to

Ribbon: `DM Electrical->Utilities->  ->Installation Settings`

Pulldown Menu: `DM Electrical->Help->Installation Settings`

There are three different installation types.

Local

A local install should be used when there is only one person using Design Master Electrical in your office. All program and customization files are stored directly on the computer. This install type should be used even if the one person is installing the software on a laptop.

See the **Local Install** section for more information about this install type.

Network

A network install should be used when there are multiple people using Design Master Electrical in your office. The program files are stored on the computer and are automatically updated from the network. The customization files are stored on the network and are shared between users.

See the **Network Install** section for more information about this install type.

Network Laptop

A network laptop install should be used if the computer is a laptop and multiple people are using Design Master Electrical in your office. The program files are stored on the laptop and are automatically updated from the network. The customization files are stored on both the network and the laptop. The network customization files will be used when the laptop is connected to the network. The local customization files will be used when the laptop is away from the office and disconnected from the network.

See the **Network Laptop Install** section for more information about this install type.
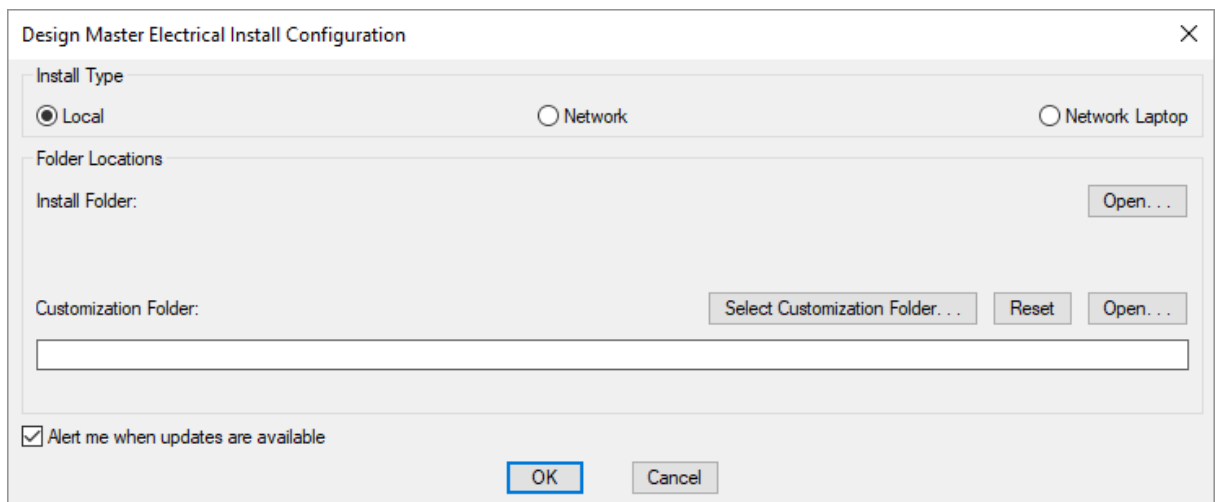
The local customization will automatically be updated with any changes to the network customization once a day. Use the `Update Laptop Customization` command to manually update the customization stored on the computer. See the **Update Laptop Customization** section for more information.

Local Install

This section describes configuring Design Master Electrical for a local install. A local install should be used when there is only one person using Design Master Electrical in your office. All program and customization files are stored directly on the computer. This install type should be used even if the person is installing the software on a laptop.

If you have floating licenses, this install type should not be used.

Configuring an Installation on a Local Computer

Install Type: The type of installation. The dialog box will change depending upon the type selected.
- **Local:** The installation type described in this section.
- **Network:** See the **Network Install** section for more information about this install type.
- **Network Laptop:** See the **Network Laptop Install** section for more information about this install type.

Install Folder: The folder where Design Master Electrical is installed. This folder is forced to be located in your Application Data folder.

Customization Folder: The folder where the customization files are stored.

Select Customization Folder: Press this button to select a new location for the customization folder.

Reset: Press this button to reset the customization to the default location, located in your Application Data folder.
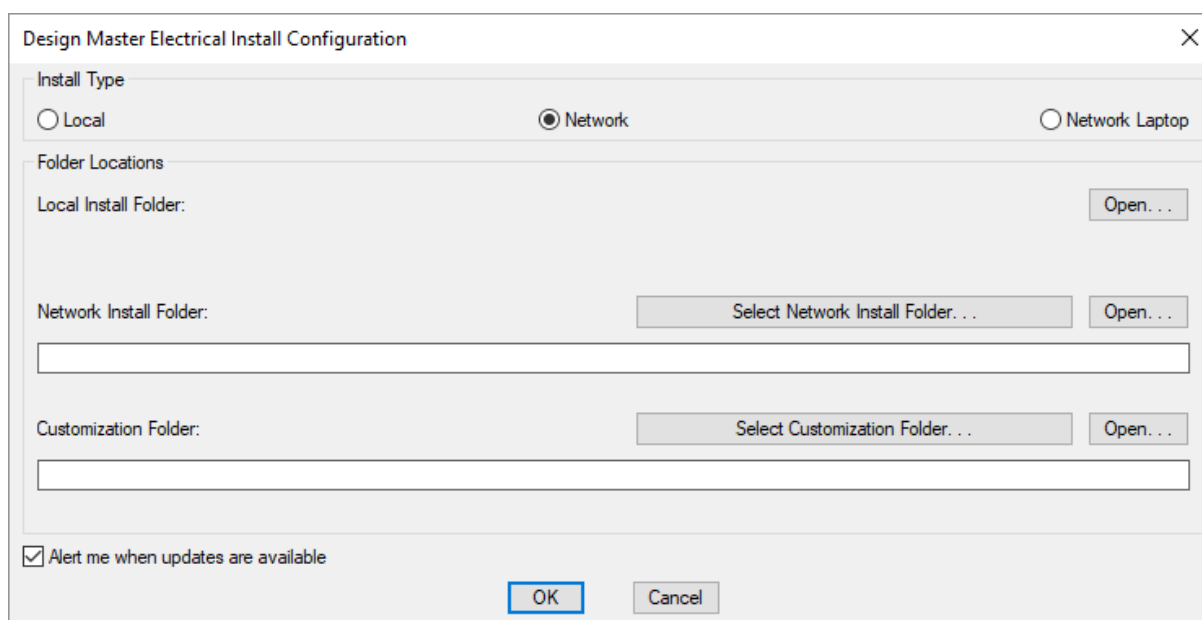
Open: There are two **Open** buttons in the dialog box. Press this button to open Windows Explorer to one of the folders listed in the dialog box. The first button opens the install folder. The second button opens the customization folder.

Alert me when updates are available: Whether updates are checked for automatically. If this is checked, an update check will occur the first time you run a command each day. An alert will appear if an update is available.

Network Install

This section describes configuring Design Master Electrical for a network install. A network install should be used when there are multiple people using Design Master Electrical in your office. The program files are stored on the computer and are automatically updated from the network. The customization files are stored on the network and are shared between users.

Configuring an Installation on a Network

Install Type: The type of installation. The dialog box will change depending upon the type selected.
- **Local:** See the **Local Install** section for more information about this install type.
- **Network:** The installation type described in this section.
- **Network Laptop:** See the **Network Laptop Install** section for more information about this install type.

Local Install Folder: The folder on the local computer where Design Master Electrical is installed. This folder is forced to be located in your Application Data folder.

Network Install Folder: The folder on the network where Design Master Electrical is installed. This folder should be on the network. All computers in your office should use the same network install folder.

Updates should be installed to this folder. Updates to this folder will automatically be copied to the local install folder.

Customization Folder: The folder where the customization files are stored. This folder should be on the network. All computers in your office should use the same customization folder.

Select Customization Folder: Press this button to select a new location for the customization folder.
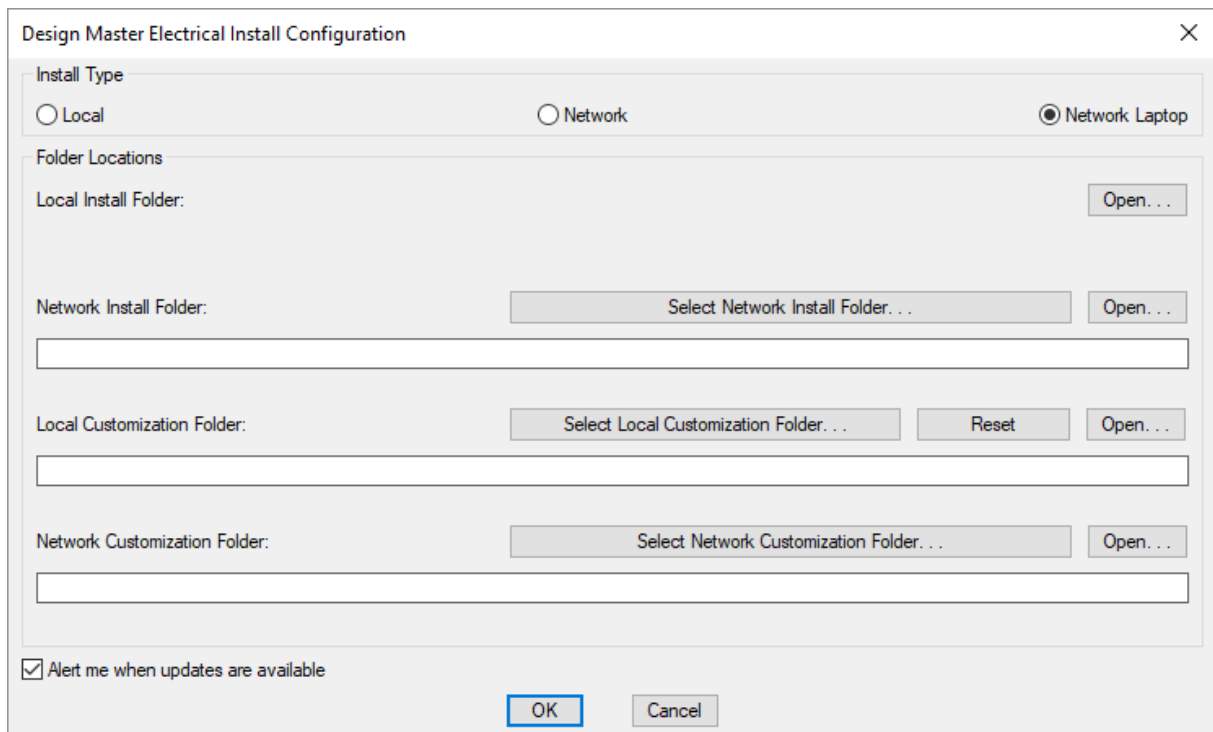
Open: There are three **Open** buttons in the dialog box. Press this button to open Windows Explorer to one of the folders listed in the dialog box. The first button opens the local install folder. The second button opens the local install folder. The third button opens the customization folder.

Alert me when updates are available: Whether updates are checked for automatically. If this is checked, an update check will occur the first time you run a command each day. An alert will appear if an update is available.

Network Laptop Install

This section describes configuring Design Master Electrical for a network laptop install. A network laptop install should be used if the computer is a laptop and multiple people are using Design Master Electrical in your office. The program files are stored on the laptop and are automatically updated from the network. The customization files are stored on both the network and the laptop. The network customization files will be used when the laptop is connected to the network. The local customization files will be used when the laptop is away from the office and disconnected from the network.

Configuring an Installation on a Laptop

Install Type: The type of installation. The dialog box will change depending upon the type selected.
- **Local:** See the **Local Install** section for more information about this install type.

390

- **Network:** See the **Network Install** section for more information about this install type.
- **Network Laptop:** The installation type described in this section.

Local Install Folder: The folder on the local computer where Design Master Electrical is installed. This folder is forced to be located in your Application Data folder.

Network Install Folder: The folder on the network where Design Master Electrical is installed. This folder should be on the network. All computers in your office should use the same network install folder.

Updates should be installed to this folder. Updates to this folder will automatically be copied to the local install folder.

Local Customization Folder: The folder where the local customization files are stored. This customization folder is used if a network connection is not available.

Select Local Customization Folder: Press this button to select a new location for the customization folder.

Reset: Press this button to reset the customization to the default location, located in your Application Data folder.

Network Customization Folder: The folder where the customization files are stored. This folder should be on the network. All computers in your office should use the same customization folder.

This folder is used when the computer is connected to the network. The files in this folder are copied to the local customization folder once each day.

Select Network Customization Folder: Press this button to select a new location for the customization folder.

Open: There are four **Open** buttons in the dialog box. Press this button to open Windows Explorer to one of the folders listed in the dialog box. The first button opens the local install folder. The second button opens the local install folder. The third button opens the local customization folder. The fourth button opens the network customization folder.

Alert me when updates are available: Whether updates are checked for automatically. If this is checked, an update check will occur the first time you run a command each day. An alert will appear if an update is available.

Update Local Install from Network

The `Update Local Install from Network` command is used to update the program files on your computer from the program files on the network if you have a **network install** or **network laptop install**.

This process should take place automatically. Only use this command if there is a problem.

To update the local install from the network, go to

Ribbon: `DM Electrical->Utilities->  ->Update Local Install from Network`

Pulldown Menu: `DM Electrical->Help->Update Local Install from Network`

The program files on your computer will be updated.

Check for Updates

The `Check for Updates` command compares the version of Design Master Electrical you have installed with the latest version that is available on our website. You will be given instructions for installing a newer version if one is available.

To check for updates, go to

Ribbon: `DM Electrical->Utilities->  ->Check for Updates`

Pulldown Menu: `DM Electrical->Help->Check for Updates`

The dialog box that appears depends upon whether an update is available from our website.

Update Not Available

If an update is not available on our website, the following dialog box will appear.
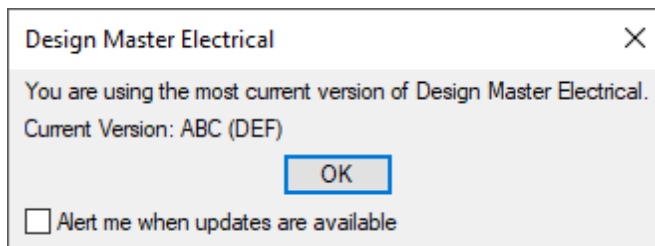
Current Version: The current version of the software. **ABC** will be replaced with the release number. **DEF** will be replaced with the build number.

Alert me when updates are available: Whether updates are checked for automatically. If this is checked, an update check will occur the first time you run a command each day. An alert will appear if an update is available.

Update Available

If the versions are different, you will be prompted to install the latest version.
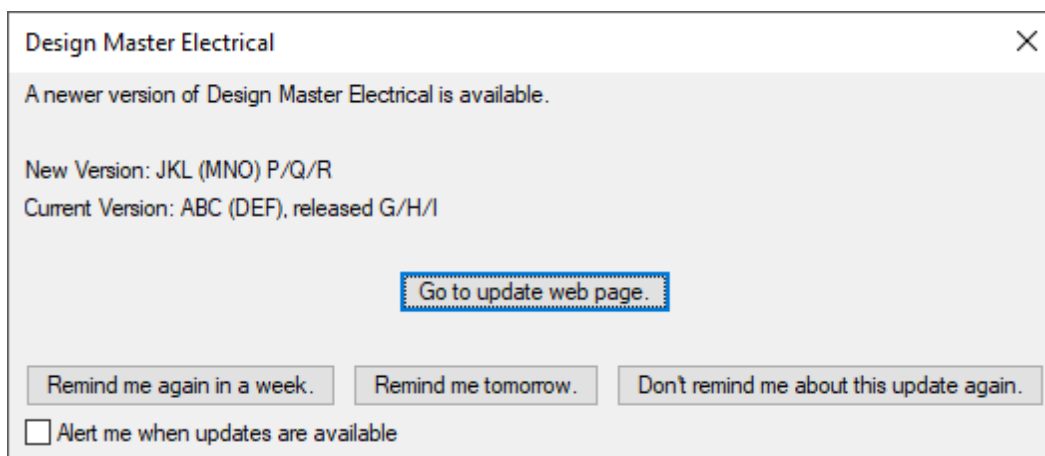
New Version: The new version of the software available from our website. **JKL** will be replaced with the new release number. **MNO** will be replaced with the new build number. **P/Q/R** is the date the new version was released.

Current Version: The current version of the software. **ABC** will be replaced with the currently installed release number. **DEF** will be replaced with the currently installed build number. **G/H/I** is the date the currently installed version was released.

Go to update web page: Press this button to go to a page on our website about the new release. The page will include a list of new features, install instructions, and a download link.

Remind me again in a week: Press this button to close the dialog box. You will be reminded about the new version again in one week.

Remind me tomorrow: Press this button to close the dialog box. You will be reminded about the new version again tomorrow.

Don't remind me about this update again: Press this button to close the dialog box. You will not be reminded about the new version again.

Alert me when updates are available: Whether updates are checked for automatically. If this is checked, an update check will occur the first time you run a command each day. An alert will appear if an update is available.

Install Patch

The `Install Patch` command is used to install patches we send to you. These patches will be ZIP files sent in response to crash reports or support requests. We send patches to specific users to solve problems they are currently having. They are not posted to our website.

We regularly release updated versions of Design Master Electrical that include all of the patches we sent to customers. These updates are available from our website. You do not need to use this command to install these updates.

Once you have installed a patch, you will not need to install it again. All future patches or updates you receive will include the patch.

To install a software patch, go to

Ribbon: `DM Electrical->Utilities->  ->Install Patch`

Pulldown Menu: `DM Electrical->Help->Install Patch`

Select Patch File Dialog Box
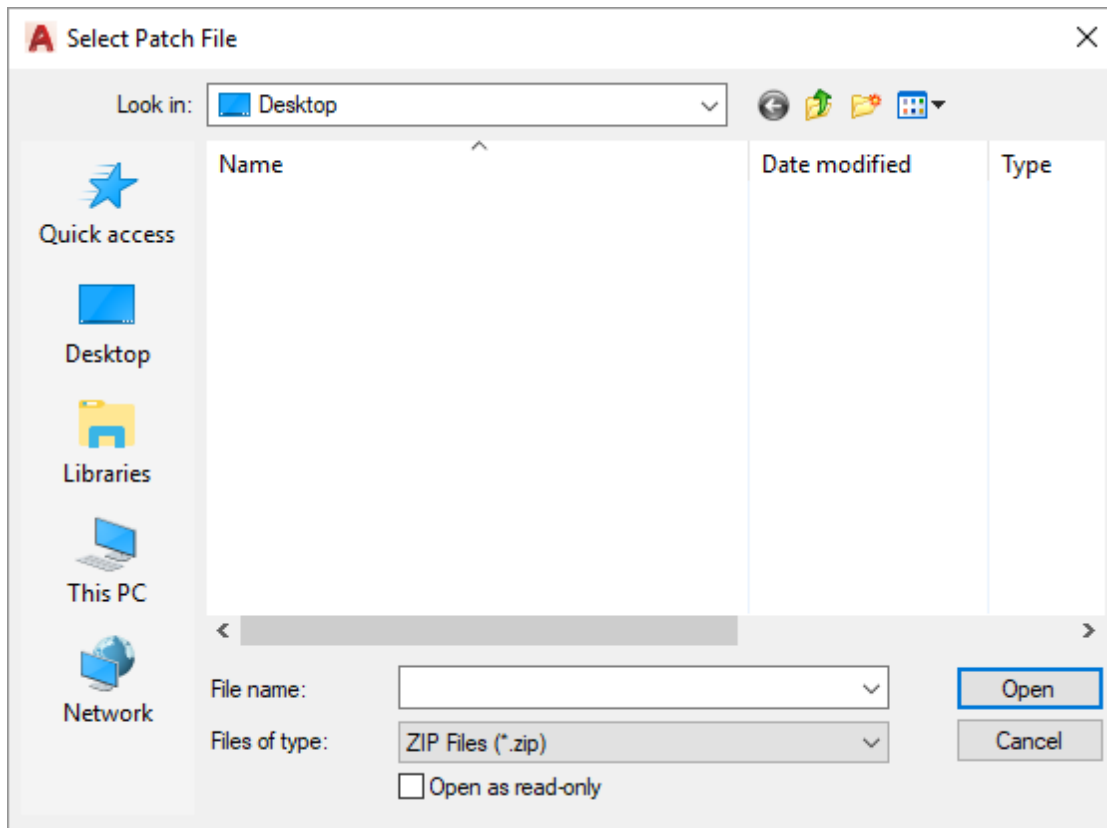
Select the patch ZIP file and press the **Open** button. The patch will be installed on your computer. If you have a **network install**, it will also be installed on the network. Installing it to the network will update all of the other computers in your office.

The patch file will be erased after it is installed. It is erased to discourage you from installing it again in the future.

Restart your CAD program after installing the patch. The error you were seeing should now be resolved.

Install License

The `Install License` command is used to activate the license for Design Master Electrical.

To activate a license, go to

Ribbon: `DM Electrical->Utilities->  ->Install License`

Pulldown Menu: `DM Electrical->Help->Install License`

Install Design Master Electrical License Dialog Box

Enter your *Name*, *Email Address*, *License ID*, and *Activation Password* in the fields provided. If you did not receive a password, leave the *Activation Password* field blank.

Press the **Activate** button to activate your license.

About

To view general information about the Design Master Electrical software, go to

Ribbon: `DM Electrical->Utilities->  ->About`

Pulldown Menu: `DM Electrical->Help->About`

A dialog box will appear that includes the version of software you are running and your current license status.

Customization

Enter topic text here.This section describes the commands available to customize the software. A wide variety of customization options are included to allow you to match the drawings that are created to your existing company standards.

General Customization Commands

Most of the customization sections include four commands:
- `Edit Project List`
- `Edit Master List` or `Edit Standards List`
- `Copy Project List to Master` or `Copy Project List to Standards`
- `Copy Master List to Project` or `Copy Standards List to Project`

The command name and how it functions varies slightly depending upon whether the default settings are stored in the master database or a standards database.

This section describes these general commands and how they work for the customization commands.

Edit Project List

The `Edit Project List` command is the first command listed in each customization section. It is used to modify the customization settings in the current project database. It allows you to make changes to a specific project without affecting other projects.

Edit Master List

The `Edit Master List` command is one of the commands that may be listed second in each customization section. It is used to modify the customization settings in the master database. These settings will be used when you start a new project. Changes here do not affect any projects already created.

Edit Standards List

The `Edit Standards List` command is one of the commands that may be listed second in each customization section. It is used to modify the customization settings in a standards database. These settings can be used when you start a new project. Changes here do not affect any projects already created.

If only one standards database exists, the command will immediately start editing that database using the appropriate dialog box.

If more than one standards database exists, you will be prompted to select the standards database to edit. After you select the standards database to edit, the appropriate dialog box will appear with the settings from that standards database displayed.

Copy Project List to Master

The `Copy Project List to Master` command will copy the settings from the project database to the master database. Often, it is simplest to start your first project and customize it as you go. When you are finished, you can use this command to copy the settings for use with future projects.

The **Copy Project to Master** dialog box will appear.
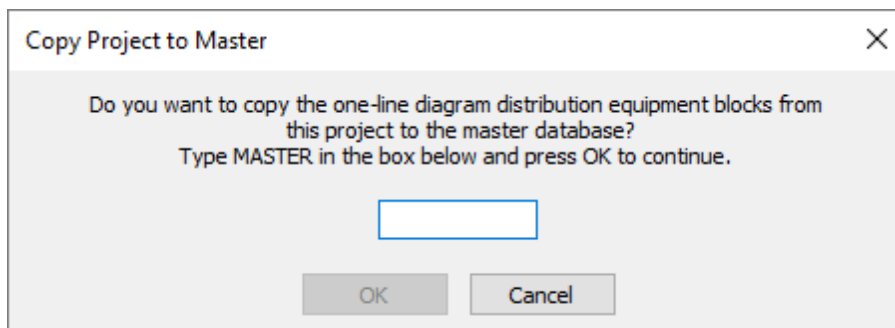
To copy the settings, type **MASTER** and press the **OK** button. This step helps prevent you from accidentally overwriting the settings in your master database.

Copy Project List to Standards

The `Copy Project List to Standards` command will copy the settings from the project database to a standards database. Often, it is simplest to start your first project and customize it as you go. When you are finished, you can use this command to copy the settings for use with future projects.

If only one standards database exists, the settings will be copied to that standards database. The **Copy Project to Standards** dialog box will appear immediately.

If more than one standards database exists, you will be prompted to select a standards database to which the settings will be copied. After you make your selection, the **Copy Project to Standards** dialog box will appear.
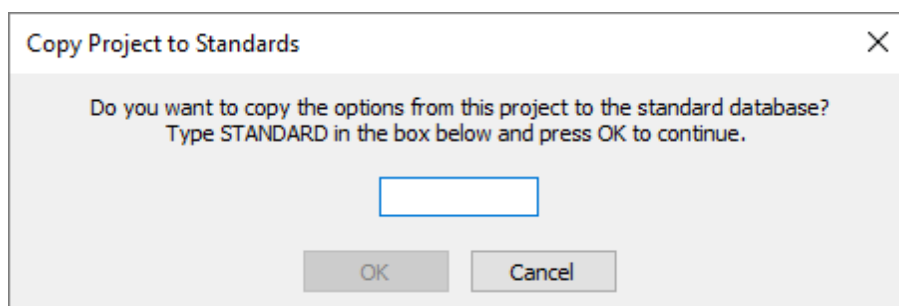
To copy the settings, type **STANDARD** and press the **OK** button. This step helps prevent you from accidentally overwriting the settings in your standards database.

Copy Master List to Project

The `Copy Master List to Project` command will copy the settings from the master database to the project database. You can use this to update projects you have already started with new settings you have set in your master database.

Copy Standards List to Project

The `Copy Standards List to Project` command will copy the settings from a standards database to the project database. You can use this to update projects you have already started with new settings you have set in your standards database.

If only one standards database exists, the settings will be copied from that standards database.

If more than one standards database exists, you will be prompted to select the standards database from which the settings will be copied.

User Options

The `User Options` command allows you to customize settings that are specific to you. These settings are stored on your computer. They are not stored in the project, master, or standards database. Some of these settings are shared with other Design Master programs you have installed on your computer.

To update your user settings, go to

Ribbon: `DME: Customization->User Customization->` OPT USER `User Options`

Pulldown Menu: `DM Electrical->Customization->User Options`

Design Master User Settings Dialog Box

The **Design Master User Options** dialog box contains a list of all of the user options and their current settings. The options are arranged in groups in the list. The groups are described in the sections below:

- **User Support Information**
- **Drawing Options**
- **Electrical**
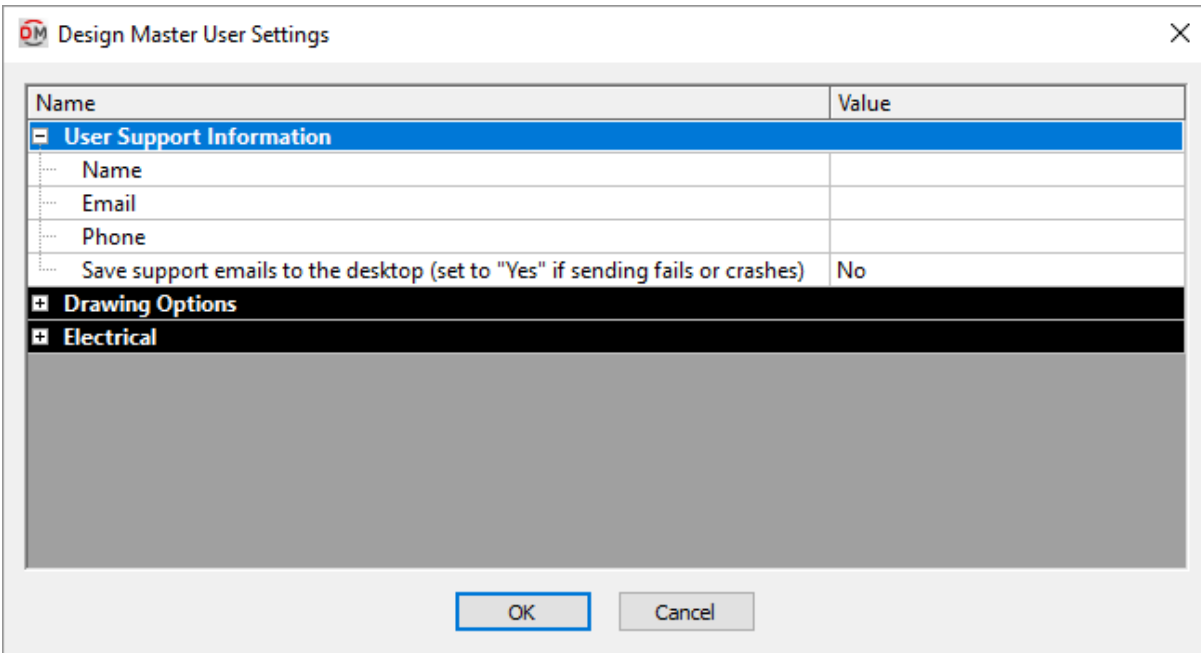
Editing a Setting

To edit an option setting, select the *Value* in the list and enter a new value.

Press the **OK** button to save your changes.

User Support Information

This section describes the options available in the **User Support Information** group in the **Design Master User Settings** dialog box.

Name: Your name.

Email: Your email address. This is the email address we will use when replying to support requests from you.

Phone: Your phone number. This phone number will be sent with any support requests you send. It is not a required field, but having it in the support requests helps us call you if we need additional information.

Save support emails to the desktop (set to "Yes" if sending fails or crashes): Support requests and bug reports can be emailed directly from Design Master Electrical. Some company's networks disable this ability, causing the send to fail or crash. When this happens, the information is saved to a file on your desktop that you can email to us manually.

- **Yes:** Design Master Electrical does not attempt to send the email directly to us. Instead, it saves it to the desktop. Choose this setting if you consistently have trouble sending us support emails through our software. Determining that the send step will fail can take time. When you know it will fail, skipping it will save you time.
- **No:** Design Master Electrical will attempt to send the email directly to us. When possible, this is the setting you should use.

Drawing Options

This section describes the options available in the **Drawing Options** group in the **Design Master User Settings** dialog box.
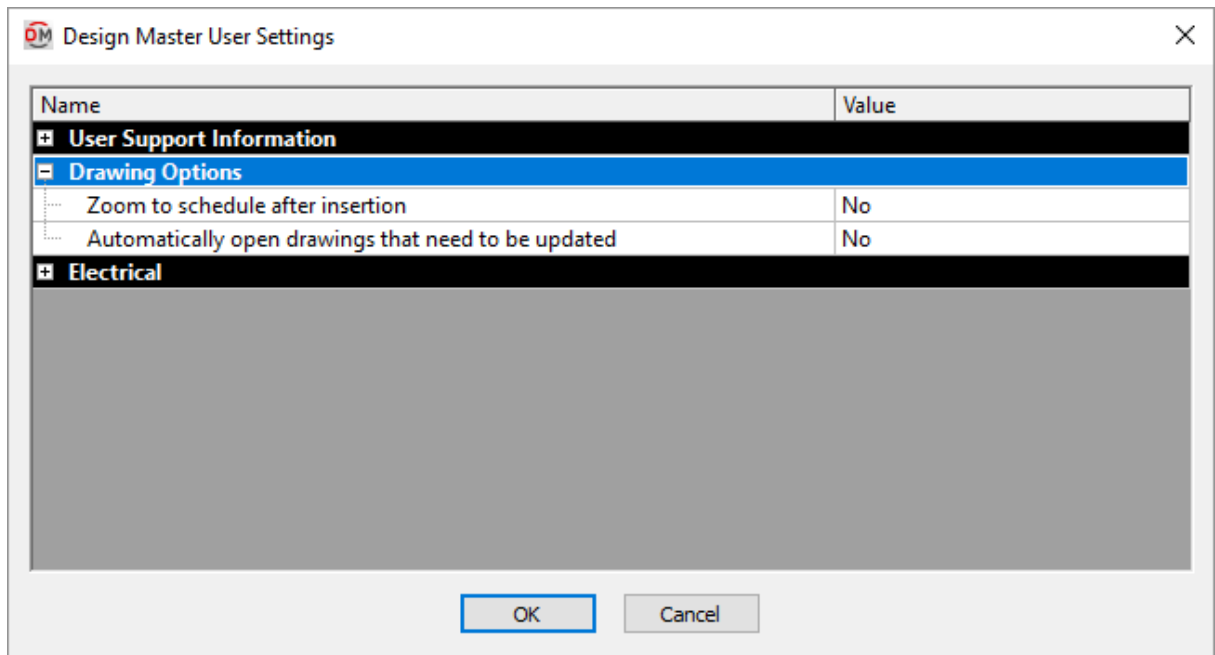
Zoom to schedule after insertion: After you insert a schedule on the drawing, you can specify whether the drawing should zoom to the schedule or remain at the same location.

- **Yes:** The drawing will zoom to the schedule after it has been inserted.
- **No:** The drawing will remain at the same location after a schedule is inserted.

Automatically open drawings that need to be updated: Design Master Electrical projects can span multiple drawings. Changes in one drawing can require that changes be made in another drawing. In order to make the changes in the other drawing, it must be opened.

- **Yes:** When a change in another drawing is required, the other drawing will be opened. This will take a moment, but when it is finished, the current drawing will remain active. If the other drawing is already open, nothing will happen.
- **No:** When a change in another drawing is required, you will be alerted at the command line, but it will not automatically be opened. The alert will continue to appear at the command line until you manually open the drawing and it is updated.

Electrical

This section describes the options available in the **Electrical** group in the **Design Master User Settings** dialog box.
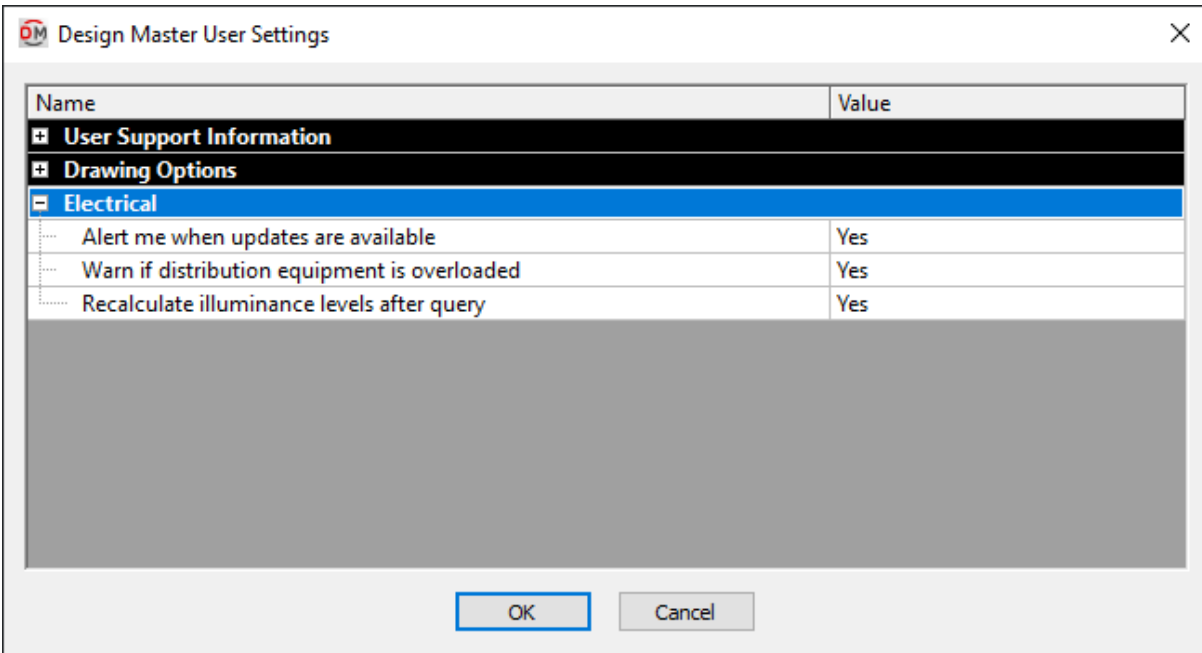
Alert me when updates are available: Whether updates are checked for automatically. If this is checked, an update check will occur the first time you run a command each day. An alert will appear if an update is available.

Warn if distribution equipment is overloaded: Whether a warning dialog box appears if you have distribution equipment in the project that is overloaded. See the **Overloaded Distribution Equipment** section for more information.

If this option is disabled, it will override the *Warn if distribution equipment is overloaded* setting in the **Options** command.

Recalculate illuminance levels after query: Whether a calculation area or line modified using the **Query Calculation Area or Solid** command will be recalculated when you press the **OK** button.

Options

The `Options` commands allow you to customize a variety of settings that affect the calculations in your project and appearance of your drawings.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Standards List`, `Copy Project List to Standards`, and `Copy Standards List to Project` commands work.

To edit the options project list, go to

Ribbon: `DME: Customization->Project Customization->` ^{OPT} P `Options`

Pulldown Menu: `DM Electrical->Customization->Options->Edit Project List`

To edit the options standards list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` OPT **S** `Options`

Pulldown Menu: `DM Electrical->Customization->Options->Edit Standards List`

Training Videos

- **Distribution Equipment Options**
- **One-Line Diagram Options**
- **Circuit Options**
- **Light Fixtures, Receptacles, Equipment, and Switches Options**
- **Homeruns and Loops Options**
- **Tick Marks, Schedules, and Leaders Options**
- **Elevation Labels and 3D Drafting Options**
- **Photometrics and Panel Schedule Options**

Design Master Electrical Options Dialog Box

Key	Value
Distribution Equipment	
Default load calculation method	General
Default general lighting circuit value	No
Default distribution equipment AIC	22,000
Fault Schedule: Highlight AIC ratings that are too low	Yes
Voltage Drop Schedule: Highlight voltage drops	Yes, Feeder or Branch Circuit > 3%, Total > 5%
Voltage Drop Schedule: Circuits to display	Too high
Voltage Drop Schedule: Warn if branch circuit voltage drop cannot be calculated	No
Voltage drop and transformers	Ignore transformers
Calculate voltage drop based upon 80% of panel capacity	No
Panel schedule load display units	kVA
Show fixed number of decimals in panel schedules	No
Number of decimals to display in panel schedule	3
Feeder length calculation method	Right Angles
Circuit length calculation method	Straight Line
Building angle	0
Extra distance added to feeder length (ft-in)	10-0
Ignore extra distance on feeders shorter than this (ft-in)	0-0
Elevation input	Relative to alignment point area (easier)
Warn if distribution equipment is overloaded	No
Distribution equipment overloaded threshold (% of panel capacity)	95
One-Line Diagram	
Distribution equipment sort order	Equipment name
Display lugs on panels with main breakers	Yes
Insert feeder IDs inside feeder lines	Yes
Include equipment feeders in feeder schedule	Yes
Include vertical main breaker on horizontal panels	Yes
Insert labels inside feeder lines	No
Feeder offset multiplier	0.3
Riser height multiplier	0.5
Default panel box height	1.5
Default panel box width	1

The **Design Master Electrical Options** dialog box contains a list of all of the options and their current settings. The options are arranged in groups in the list.

Editing a Setting

To edit an option setting, select the *Value* in the list and enter a new value.

Press the **OK** button to save your changes.

If the change you made to the options does not immediately appear on the drawing, run the **Coordinate Drawings and Database** command. This command will update the drawing to use the new settings you have specified.

Distribution Equipment

This section describes the options available in the **Distribution Equipment** group in the **Design Master Electrical Options** dialog box.

Default load calculation method: When a new distribution equipment is created, the calculation method for load types is set to this value. See the **Feeder and Service Calculation Settings** section for more information.

Default general lighting circuit value: When a new distribution equipment is created, the default general lighting setting for circuits on the equipment is set to this value. See the **Feeder and Service Calculation Settings** section for more information.

Default distribution equipment AIC: When a new distribution equipment is created, the AIC rating is set to this value. The value for each panel can be changed once it is created.

Fault Schedule: Highlight AIC ratings that are too low: Whether AIC ratings that are too low are highlighted in the fault schedule. See the **Fault Schedule** section for more information.

Voltage Drop Schedule: Highlight voltage drops: Whether voltage drops that are too high are highlighted in the voltage drop schedule. See the **Voltage Drop Schedule** section for more information.

Voltage Drop Schedule: Circuits to display: Which circuits are displayed in the voltage drop schedule. See the **Voltage Drop Schedule** section for more information.

Voltage Drop Schedule: Warn if branch circuit voltage drop cannot be calculated: Whether a warning is displayed if the branch circuit voltage drop cannot be calculated correctly due to circuiting errors. See the **Voltage Drop Schedule** section for more information.

Voltage drop and transformers: There are three options for addressing transformers when calculating voltage drop. See the **Voltage Drop and Transformers Options** article in the knowledge base for more information.
- **Ignore transformers:** The voltage drop through the transformer is ignored when the voltage drop for the system is calculated. The voltage drop through the upstream feeders is carried through the transformer.
- **Reset voltage drop at transformers:** The voltage drop is reset to 0% at each transformer. The voltage drop through the transformer is ignored. The voltage drop through the upstream feeders is ignored.
- **Include transformers in voltage drop:** The voltage drop through the transformer is calculated and added to the voltage drop on the feeder. The voltage drop through the upstream feeders is carried through the transformer.

After changing the *Voltage drop and transformers* option, reinsert the voltage drop schedule on the drawing to display voltage drop numbers based upon the new setting. See the **Voltage Drop Schedule** section for more information about inserting the schedule.

Calculate voltage drop based upon 80% of panel capacity: Whether calculations for the voltage drop schedule are based upon 80% of panel capacity or actual panel loads. See the **Voltage Drop Schedule** section for more information.

Panel schedule load display units: This option controls the units used for displaying loads on panel schedules. Changing this value will change the numbers, but it will not change the column labels and headers. You will need to manually customize the distribution equipment schedule blocks. See the **Distribution Equipment Schedule Blocks** section for more information.
- **kVA:** The loads are displayed in kVA.
- **VA:** The loads are displayed in VA.
- **Amps:** The loads are displayed in amps.

Show fixed number of decimals in panel schedules: This option controls how values are displayed in distribution equipment schedules.
- **Yes:** The full value for all numbers will be displayed. All numbers will include a specific number of digits after the decimal.
- **No:** All numbers on the panel schedule will be rounded to 3 significant figures.

Number of decimals to display in panel schedule: If the *Show fixed number of decimals in panel schedules*

option is set to **Yes**, this option specifies the number of digits after the decimal to display.

If the *Show fixed number of decimals in panel schedules* option is set to **No**, this option is ignored and not used.

Feeder length calculation method: When a new distribution equipment is created, the calculation method for feeder length is set to this value. See the **Feeder Length Settings** section for more information.

Circuit length calculation method: When a new distribution equipment is created, the calculation method for circuit length is set to this value. See the **Set Circuit Information** section for more information.

Building angle: When a new distribution equipment is created, the building angle used for feeder and circuit length calculations is set to this value. See the **Feeder Length Settings** section and the **How Building Angle Affects Calculations** article in the knowledge base for more information.

Feeder make-up: This value is the extra length of wire included in feeder lengths to account for the wire used for distribution equipment connections. It is used in the fault calculation, voltage drop calculation, and takeoffs. See the **Distance Input Format** section for more information about specifying this length.

Ignore make-up on feeders shorter than this: If two pieces of distribution equipment are close to each other, additional feeder length is not needed to terminate the wires. If the distance between distribution equipment is shorter than this specified value, the value set for the *Feeder make-up* option is not added to the length of the feeder. See the **Distance Input Format** section for more information about specifying this length.

Elevation input: When distribution equipment is inserted on the drawing, the elevation input method is set to this value. See the **Elevation** section for more information.

Warn if distribution equipment is overloaded: Whether a warning is displayed if you have distribution equipment in the project that is overloaded. See the **Overloaded Distribution Equipment** section for more information.

This setting will affect the project for all users. To change this setting for a single user, see the **Electrical User Options** section.

Distribution equipment overloaded threshold (% of panel capacity): When the load on a distribution equipment exceeds this value as a percentage of the load capacity, the distribution equipment will be considered overloaded.

One-Line Diagram

This section describes the options available in the **One-Line Diagram** group in the **Design Master Electrical Options** dialog box.

Design Master Electrical Options

Key	Value
Distribution Equipment	
One-Line Diagram	
Distribution equipment sort order	Equipment name
Display lugs on panels with main breakers	Yes
Insert feeder IDs inside feeder lines	Yes
Include equipment feeders in feeder schedule	Yes
Include vertical main breaker on horizontal panels	Yes
Insert labels inside feeder lines	No
Feeder offset multiplier	0.3
Riser height multiplier	0.5
Default panel box height	1.5
Default panel box width	1
Default fed from (in automatic riser)	Top
Default feeder connection layout	Connect feeders like a riser
Feeder connection location in riser to panels with no bus	Side
"Main Lugs Only" label	MLO
Default riser blocks	
Default one-line blocks (multiple downstream panels)	
Default one-line blocks (0 or 1 downstream panels)	
Default equipment connection block	dm_elec-oneline-equipment
Default equipment disconnect block	<None>
Default circuit OCP block	<None>
Main breaker block	dm_elec-oneline-breaker
Main fused switch block	dm_elec-oneline-fusedSwitch
Main non-fused switch block	dm_elec-oneline-nonFusedSwitch
Standard lugs block	dm_elec-oneline-standardLugs
Double lugs block	dm_elec-oneline-standardLugs
Feedthrough lugs top block	dm_elec-oneline-standardLugs
Feedthrough lugs bottom block	dm_elec-oneline-standardLugs
Wire break block	dm_elec-oneline-break
Feeder ID format	Short
Automatically create feeder IDs	Yes
Copy feeder IDs to master schedule when created	Yes
Transfer feeder IDs from master schedule if possible	Yes
Circuits, Load Types, and Wire Sizes	
Light Fixtures	
Receptacles	
Equipment	
Switches	
Homeruns and Loops	
Tick Mark Blocks	
Device Elevation Labels	
Conduit and Cable Tray Elevation Labels	
3D Drafting	
Schedules	
Selective Coordination Graph	
General Point-by-Point Photometrics	
Egress Point-by-Point Photometrics	
Print Panel Schedules to HTML	
Leaders	

OK   Cancel

Distribution equipment sort order: When generating the one-line diagram, this option determines how downstream equipment connections are ordered on the diagram.
- **Breaker size:** Generation order is based upon the breaker size.
- **Circuit number:** Generation order is based upon the circuit to which the downstream equipment is connected.
- **Distance:** Generation order is based upon the distance between the upstream and downstream equipment.
- **Equipment name:** Generation order is based upon the name of the equipment.

Display lugs on panels with main breakers: Whether lugs are displayed on panels with main breakers.
- **Yes:** Lugs are displayed on all bus bars.
- **No:** Lugs are not displayed on bus bars with main breakers.

This option does not change whether the lugs exist on the panel, only whether they are displayed.

Insert feeder IDs inside feeder lines: Whether feeder IDs appear on the one-line diagram.

Include equipment feeders in feeder schedule: Whether feeders to equipment connections are included on the feeder schedule.

If *Insert feeder IDs inside feeder lines* is set to **Yes**, this option will be set to **Yes**. If *Insert feeder IDs inside feeder lines* is set to **No**, equipment connection feeders will only appear on the feeder schedule if they are labeled manually.

Include vertical main breaker on horizontal panels: Whether the main breaker on a horizontal custom panel block is inserted vertically.
- **Yes:** The main breaker is inserted perpendicular to the horizontal panel bus.
- **No:** The main breaker is inserted along the horizontal panel bus.

Insert labels inside feeder lines: Whether feeder labels can snap to inside feeder lines when being inserted.

Feeder offset multiplier: This value sets the default distance when offsetting feeders. This value is multiplied by `DIMSCALE` to determine the distance.

Riser height multiplier: This value sets the default distance between elevations when generating a riser diagram.

Default panel box height: When inserting a custom block on the diagram, the *Specific Panel Box Size* height is set to this value. See the **[Insert Distribution Equipment](#)** section for more information.

Default panel box width: When inserting a custom block on the diagram, the *Specific Panel Box Size* width is set to this value. See the **[Insert Distribution Equipment](#)** section for more information.

Default fed from (in automatic riser): Whether distribution equipment blocks are fed from the top or the bottom when generating a riser diagram.

Default feeder connection layout: Sets how feeders are drawn for distribution equipment inserted into the diagram using the **[Insert Distribution Equipment](#)** command. This option can also be set in the **Insert Distribution Equipment** dialog box.

Feeder connection location in riser to panels with no bus: Sets how feeders are drawn for distribution equipment blocks with no bus when generating a riser diagram.

"Main Lugs Only" label: Sets the text for the *Main Trip Amps*, *Main Trip Amps (MLO)*, and *Main Trip Amps and Poles* labels when the main disconnect for the distribution equipment is set to **Main Lugs Only**.

Default riser blocks: Sets the default block for each distribution equipment type when generating a riser diagram.

Default one-line blocks (multiple downstream panels): Sets the default block for each distribution equipment type that is connected to multiple downstream distribution equipment when generating a one-line diagram.
- **<Same as Riser Block>:** Uses the same default block as the *Default riser blocks* option.

Default one-line blocks (0 or 1 downstream panels): Sets the default block for each distribution equipment type that is connected to one or fewer downstream distribution equipment when generating a one-line diagram.
- **<Same as Riser Block>:** Uses the same default block as the *Default riser blocks* option.

Default equipment connection block: Sets the default block for equipment connections on the one-line diagram.

Default equipment disconnect block: Sets the default block for the equipment disconnect on the one-line diagram.

Default circuit OCP block: Sets the default block for OCP on the one-line diagram.

Main breaker block: Sets the default block for main breakers on the one-line diagram.

Main fused switch block: Sets the default block for main fused switches on the one-line diagram.

Main non-fused switch block: Sets the default block for main non-fused switches on the one-line diagram.

Standard lugs block: Sets the default block for standard lugs on the one-line diagram.

Double lugs block: Sets the default block for double lugs on the one-line diagram.

Feedthrough lugs top block: Sets the default block for feedthrough lugs connected at the top on the one-line diagram.

Feedthrough lugs bottom block: Sets the default block for feedthrough lugs connected at the bottom on the one-line diagram.

Wire break block: Sets the default block for wire breaks on the one-line diagram.

Feeder ID format: Sets how feeder IDs are generated. See the **Feeder ID Formats** article in the knowledge base for more information.

Automatically create feeder IDs: Whether feeder IDs are automatically generated.
- **Yes:** Feeder IDs are generated automatically based upon the *Feeder ID format*.
- **No:** When a feeder ID is generated, the **New Feeder Callout ID** dialog box will appear. You can then set the feeder ID, feeder ID group, whether feeder IDs should be automatically generated in the future, and whether the feeder ID should be added to the master schedule.

Copy feeder IDs to master schedule when created: Whether feeder IDs are copied to the master schedule when they are generated.

Transfer feeder IDs from master schedule if possible: When a feeder ID is generated with the same specifications as an existing feeder ID in the master schedule, this option sets whether a new feeder ID is generated or the existing feeder ID is used.

Circuits, Load Types, and Wire Sizes

This section describes the options available in the **Circuits, Load Types, and Wire Sizes** group in the **Design Master Electrical Options** dialog box.

Branch circuit make-up: This value is the extra length of wire included in branch circuit lengths to account for branch circuit connections. It is used in the fault calculation, voltage drop calculation, and takeoffs. See the **Distance Input Format** section for more information about specifying this length.

Add branch circuit make-up for each device: Whether the length specified in *Branch circuit make-up* is added for each device on the circuit.
- **Yes:** The make-up added to the branch circuit length is multiplied for each device on the circuit.
- **No:** The make-up is only added to the branch circuit length once, regardless of the number of devices on the circuit.

Unused circuit description: Specifies how circuits that do not have any devices connected to them are labeled in the panel schedule.
- **Space with breaker size listed:** The circuit description is set to **SPACE**. The breaker size is displayed as the smallest default breaker.
- **Space with no breaker listed:** The circuit description is set to **SPACE**. The breaker size is displayed as a dash.
- **Spare with breaker size listed:** The circuit description is set to **SPARE**. The breaker size is displayed as the smallest default breaker.
- **Spare with no breaker listed:** The circuit description is set to **SPARE**. The breaker size is displayed as a dash.
- **Dash with no breaker listed:** The circuit description is displayed as a dash. The breaker size is displayed as a dash.

Use the **Update Distribution Equipment Schedule** command to update schedules after changing the option.

409

Default circuit layer system: The default layer system associated with circuits and taps on panels.
- **Same as Distribution Equipment:** The circuits will use the same layer system as the distribution equipment with which they are associated.
- **<Layer System>:** The circuits will use the specific layer system chosen.

This default setting can be overridden for specific circuits using the **Circuiting** dialog box. See the **Circuiting** section for more information about setting the layer system for circuits.

Receptacle circuit count limit: This setting is used to alert you when a certain number of receptacles have been placed on a single circuit. This alert happens while circuiting the receptacles. Once the receptacles are circuited, this warning will not be shown.

The count is based upon total load, not the number of devices. Every 0.18 kVA of load is counted as a receptacle. This way, a quad receptacle with 0.36 kVA that is inserted as a single receptacle is counted as two receptacles and not one.

The warning does not prevent you from connecting more receptacles to the circuit than specified for situations where that is necessary.

Circuit amp limit: This setting is used to alert you when the load on a circuit has reached a certain number of amps. This alert happens while circuiting devices. Once the devices are circuited, this warning will not be shown.

The warning does not prevent you from connecting more devices to the circuit than specified for situations where that is necessary.

Include room names in circuit descriptions: Whether room names are included in circuit descriptions on panel schedules. If one of the **Yes** options is selected, all of the rooms containing a device on the circuit are listed in the description. These rooms must be inserted using the **Create Room** command.
- **Yes, include "Room" in description:** The rooms are listed in the circuit description. The word "Room" and the room name is included in the description.
- **Yes, do not include "Room" in description:** The rooms are listed in the circuit description. Only the room name is included in the description.
- **No:** The rooms are not listed in the circuit description.

Size breakers to 100% of noncontinuous load: Whether branch circuits are sized to 100% of noncontinuous loads (per *NEC 210.20(A)*) or to 125% of noncontinuous loads.
- **Yes:** Branch circuits are sized to 100% of noncontinuous loads and 125% of continuous loads. This setting gives you smaller breakers and wire sizes.
- **No:** Branch circuits are sized to 125% of all loads. This setting gives you larger breakers and wire sizes.

Continuous load types:
- Lighting
- Largest Motor (*NEC 430.24*)
- Other Motors (*NEC 430.24*)
- Continuous
- Heating (*NEC 424.4(B)*)
- Cooling, Largest Motor (*NEC 430.24*)
- Cooling, Other Motors (*NEC 430.24*)
- Heating and Cooling Motor (*NEC 430.24*)
- Appliance, Continuous
- Appliance Motor (*NEC 430.24*)
- Dwelling Unit Heating (*NEC 424.4(B)*)
- Dwelling Unit Cooling, Largest Motor (*NEC 430.24*)

- Dwelling Unit Cooling, Other Motors (*NEC 430.24*)
- Dwelling Unit Heating and Cooling Motor (*NEC 430.24*)

Noncontinuous load types:
- Receptacles (*NEC Handbook Exhibit 220.4*)
- Noncontinuous
- Kitchen
- Diverse
- Small Appliance
- Laundry
- Appliance, Noncontinuous
- Electric Dryer
- Electric Cooking
- Marina / Mobile Home / RV

All motors are considered continuous at the branch circuit level.

Custom Load Type 1 and Custom Load Type 2: You can create two custom load types, which can be used on your projects for load types that are not included with the software. The settings for the two custom load types are the same.

Name: The name of the custom load type. It is displayed when selecting the load type of a device and it is printed on the distribution equipment schedule load summary.

Calculation method: The method used for setting the demand factor for the load type.
- **Fixed demand factor:** The demand factor for the load is constant and fixed. (Most of the default load types are fixed.)
- **Variable demand factor:** The demand factor varies depending upon the load. (For example: kitchen and diverse load types)
- **Variable calculated load:** The demand factor varies, but the calculated load is input rather than the demand factor.

Demand factor: The demand factor for the load type. Enabled if *Calculation method* is set to **Fixed demand factor**.

Load type: Whether the load type should be treated as a **Continuous** or **Noncontinuous** load for branch circuit sizing.

Wire Sizing Option: Which sizing option is selected in the **Wire Sizing** dialog box. See the **Wire Sizing** section for more information about creating custom wire sizing tables. This option is most useful in standards database to set the default sizing option for new projects.

Upsize #12 to #10 wire length: Branch circuits that are sized to #12 and exceed this length are automatically upsized to #10 wire.

Show #10 on homerun when wires upsized: Branch circuits that are sized to #12 wire are automatically upsized if they exceed the length specified in *Upsize #12 to #10 wire length*. This option controls whether a #10 is displayed next to upsized homeruns on the drawing.

Display neutral wire count separately from phase wire count: Whether the neutral wire and phase wires are counted separately on wire callouts when they are the same size.

Conduit location: Whether the conduit is listed at the start or end of wire callouts.

Size utility and transformer grounds as service grounds: Whether the ground for utilities and transformers is sized as a service ground or an equipment ground.

Light Fixtures

This section describes the options available in the **Light Fixtures** group in the **Design Master Electrical Options** dialog box.
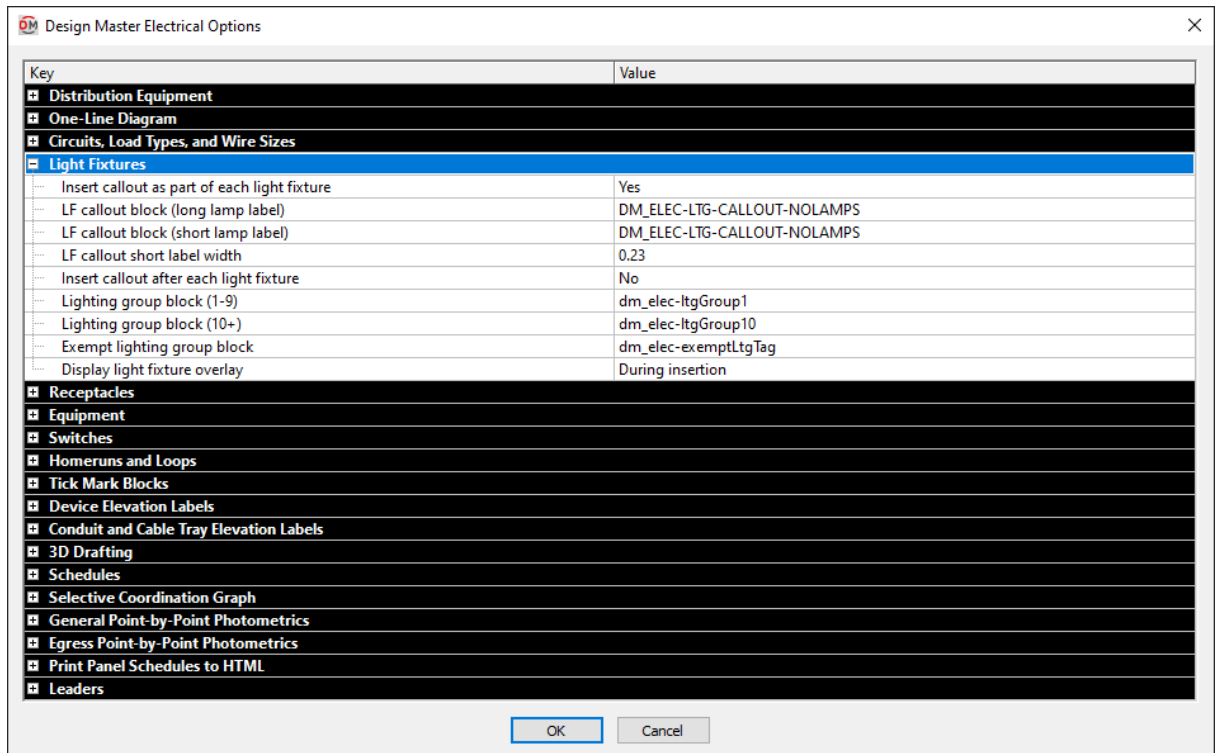

Insert callout as part of each light fixture: Whether a light fixture callout note is automatically inserted as part of each light fixture on the drawing.

LF callout block (long lamp label): The block used when you insert a light fixture callout. It is used when the lamp label is wider than the value specified in *LF callout short label width*.

LF callout block (short lamp label): The block used when you insert a light fixture callout. It is used when the lamp label is smaller than the value specified in the *LF callout short label width*.

LF callout short label width: This width is used to determine which LF callout is inserted for a light fixture.

Insert callout after each light fixture: Whether you will be prompted to a light fixture callout after inserting a light fixture on the drawing. The light fixture callout that is inserted is the same as what is inserted using the **Insert Light Fixture Callout** command.

Lighting group block (1-9): The callout block used when you associate a light fixture with a lighting group. It is used when the group number is less than 10.

Lighting group block (10+): The callout block used when you associate a light fixture with a lighting group. It is used when the group number is 10 or more.

Exempt lighting group block: The block used when a light fixture is marked as exempt. Leave this value blank to not insert a block next to exempt fixtures. See the **Exempt Light Fixtures** section for more information.

Display light fixture overlay: This specifies when a light fixture overlay will be displayed on non-plotting layers.
- **Never:** Light fixture overlays will never be inserted.
- **During insertion:** The overlay will be displayed only when the light fixture is inserted. Once the light fixture is inserted, the overlay is no longer displayed. To view the overlay again, you must insert a new fixture.
- **Always:** The light fixture overlay will always be displayed.

The light fixture overlay is only generated for fixtures that have sufficient photometric information defined. The *General Light* checkbox or *Egress Light* checkbox must be checked. The *Elevation* must be set higher than 0. An IES file must be associated with the fixture callout.

Receptacles

This section describes the options available in the **Receptacles** group in the **Design Master Electrical Options** dialog box.
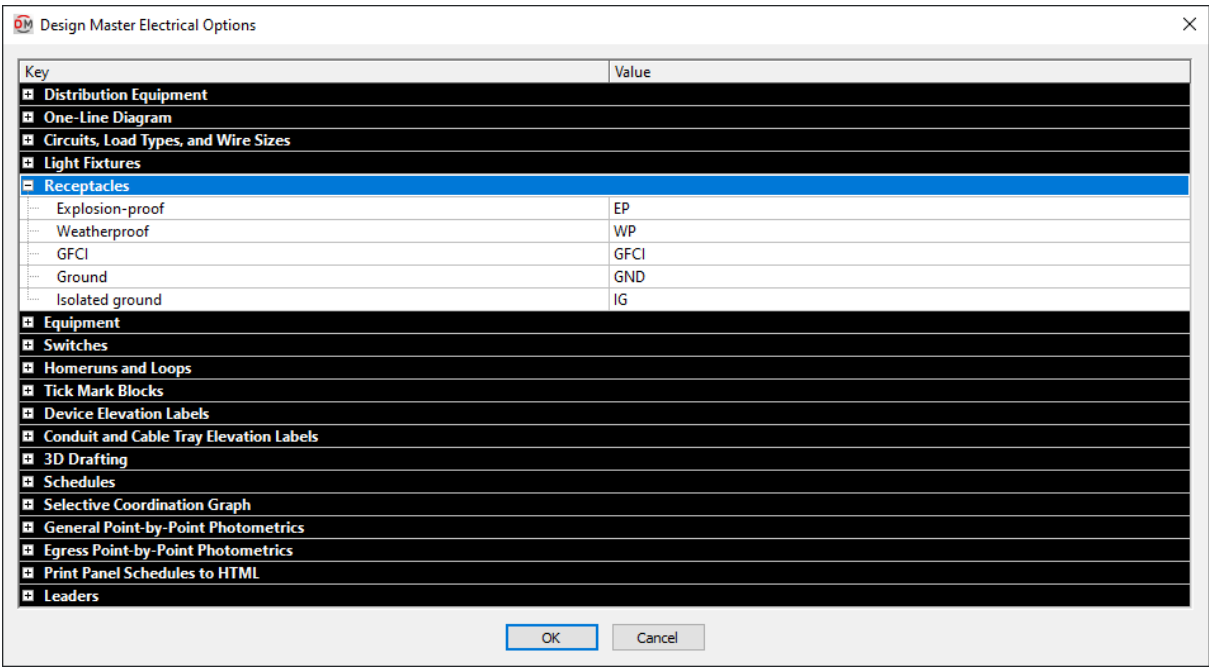
Explosion-proof: This label is inserted with receptacles that have the *Explosion-proof* box checked in the receptacle schedule.

Weatherproof: This label is inserted with receptacles that have the *Weatherproof* box checked in the receptacle schedule.

GFCI: This label is inserted with receptacles that have the *GFCI* box checked in the receptacle schedule.

Ground: This label is inserted with receptacles that have the *Ground* box checked in the receptacle schedule.

Isolated ground: This label is inserted with receptacles that have the *Isolated Ground* box checked in the receptacle schedule.

Equipment

This section describes the options available in the **Equipment** group in the **Design Master Electrical Options** dialog box.
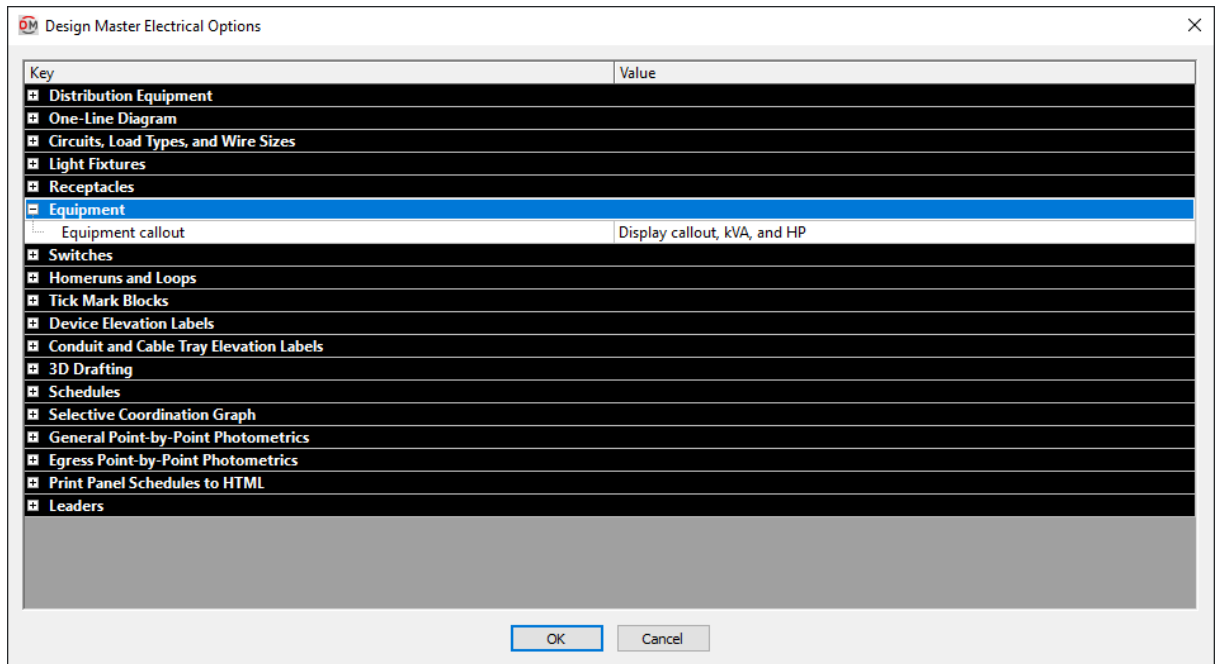
Equipment callout: This option controls the information that is displayed on the callout for each piece of equipment. See the **How to Use or Create a Custom Equipment Callout** article in the knowledge base for more information.

- **Display callout, kVA and HP:** The equipment callout contains two lines. The top line displays the callout. The bottom line displays the loads on the equipment and the voltage. Motor loads with a horsepower value set are displayed in horsepower. All other loads are displayed in kVA.
- **Display callout, amps, and HP:** The equipment callout contains two lines. The top line displays the callout. The bottom line displays the loads on the equipment and the voltage. Motor loads with a horsepower value set are displayed in horsepower. All other loads are displayed in amps.
- **Display callout and amps only:** The equipment callout contains two lines. The top line displays the callout. The bottom line displays the loads on the equipment and the voltage. All loads are displayed in amps.
- **Display callout only:** The equipment callout contains a single line that displays the callout.
- **No callout:** No equipment callout is inserted on the drawing.
- **Select custom block:** Select this option to choose a custom block for your equipment callout. Once the block is selected, the block name will be displayed.

Switches

This section describes the options available in the **Switches** group in the **Design Master Electrical Options** dialog box.
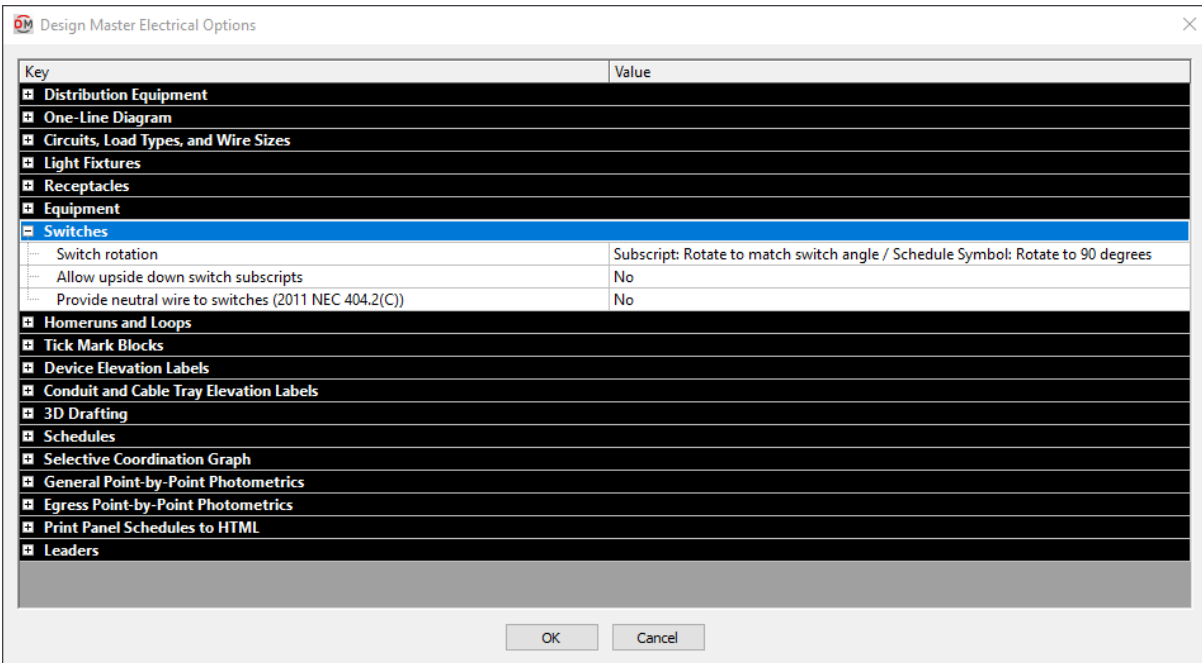
Switch rotation: This option controls how switches and switch subscripts are rotated. Switch subscripts should be rotated to match the angle of the switch. Switches in the schedule should be rotated so that the $ is drawn vertical. Which option you choose to make this happen correctly depends upon how you have your switch blocks defined.

If you are uncertain which option to choose, the simplest solution is to choose an option and insert a switch schedule on the drawing. Try each option until you find one that looks correct.

- **Subscript: Rotate to match switch angle / Schedule Symbol: Rotate to 90 degrees:** The switch subscript is inserted at the same rotation angle as the switch. The switch in the schedule is rotated 90 degrees. Choose this option if your switch rotation is set perpendicular to the wall.
- **Subscript: Rotate to match switch angle + 90 degrees / Schedule Symbol: Rotate to 0 degrees:** The switch subscript is rotated 90 degrees relative to the switch. The switch in the schedule is not rotated. Choose this option if your switch rotation is set parallel to the wall.
- **Subscript: Rotate to match switch angle - 90 degrees / Schedule Symbol: Rotate to 180 degrees:** The switch subscript is rotated -90 degrees relative to the switch. The switch in the schedule is rotated -90 degrees. Choose this option if your switch rotation is set parallel to the wall.
- **Subscript: Rotate to 0 / Schedule Symbol: Rotate to 0 degrees:** The switch subscript is not rotated. The switch in the schedule is not rotated. Choose this option if your switch rotation is set parallel to the wall and you do not want the switch subscript rotated automatically.
- **Subscript: Rotate to 0 / Schedule Symbol: Rotate to 90 degrees:** The switch subscript is not rotated. The switch in the schedule is rotated 90 degrees. Choose this option if your switch rotation is set perpendicular to the wall and you do not want the switch subscript rotated automatically.

Allow upside down switch subscripts: Whether a switch subscript can be displayed upside down.
- **Yes:** The switch subscript will be rotated in exactly the same way as the switch. If the switch is upside down, the subscript will be upside down.
- **No:** The switch subscript will first be rotated in the same way as the switch. If the text is upside down, it will then be rotated an additional 180 degrees so that it is readable.

Provide neutral wire to switches: Whether a neutral wire is provided to switches. The option is required for *2011 NEC 404.2(C)*.

Homeruns and Loops

This section describes the options available in the **Homeruns and Loops** group in the **Design Master Electrical Options** dialog box.

Hot or neutral first: Whether the hot or neutral ticks marks are displayed first in groups of tick marks. The type of tick mark that is displayed first will be inserted closer to the homerun arrowhead.

Ground location: Sets the location of the ground wire tick mark in groups of tick marks. The **Front** is located toward the homerun arrowhead. The **Back** is located away from the homerun arrowhead.

Draw loops: How loops are drawn between devices. See the **Draw Loops Between Circuited Devices** section for more information about the available options.

Loop to same circuit first on combined homeruns (light fixtures): How loops are drawn on combined homeruns containing light fixtures. See the **Set Looping Options** section for more information.

Loop to same circuit first on combined homeruns (other devices): How loops are drawn on combined homeruns containing devices other than light fixtures. See the **Set Looping Options** section for more information.

Homerun wire callouts and tick marks: Sets when wire callouts and/or tick marks are displayed on homeruns.
- **Tick marks for all homeruns:** Tick marks are displayed on all homeruns. Wire callouts are not displayed on homeruns.
- **Tick marks for #12, wire callouts if > #12:** Tick marks are displayed if the wires are #12. Wire callouts are displayed if the wires are #10 or larger.
- **Tick marks for #12, tick marks and wire callouts if > #12:** Tick marks are displayed if the wires are #12. Wire callouts and tick marks are displayed if the wires are #10 or larger.

- **Tick marks for #12 1-pole, wire callouts for all others:** Tick marks are displayed if the wires are #12 and the distribution equipment has one pole. Wire callouts are displayed if the wires are #10 or larger or if the wires are #12 and the distribution equipment has two or three poles.
- **Wire callouts for all homeruns:** Wire callouts are displayed on all homeruns. Tick marks are not displayed on homeruns.
- **No wire callouts or tick marks:** Wire callouts and tick marks are not displayed on homeruns.
- **Nothing for #12, wire callouts if > #12:** Wire callouts and tick marks are not displayed if the wires are #12. Wire callouts are displayed if the wires are #10 or larger.

Display tick marks on loops: Whether tick marks are displayed on loops between devices.

Multiple arrowheads on combined homeruns: Whether combined homeruns display a single arrowhead or one arrowhead for each circuit on the homerun.

Share neutrals on combined homeruns: Whether neutrals on combined homeruns are shared**.** The *2008 NEC* requires this to be set to **No**. To share neutrals in accordance with the *2008 NEC,* create a multiple pole breaker and use the **Circuit to Phase** button on the **Circuiting** dialog box.

Circuit break block: The block that is used when circuits are broken. Use the **None** setting if you do not want a block displayed on broken circuits.

Homerun arrow block: The block used for homerun arrowheads.

Homerun arrow length: The length of the homerun arrowhead.

Tick Mark Blocks

This section describes the options available in the **Tick Mark Blocks** group in the **Design Master Electrical Options** dialog box.
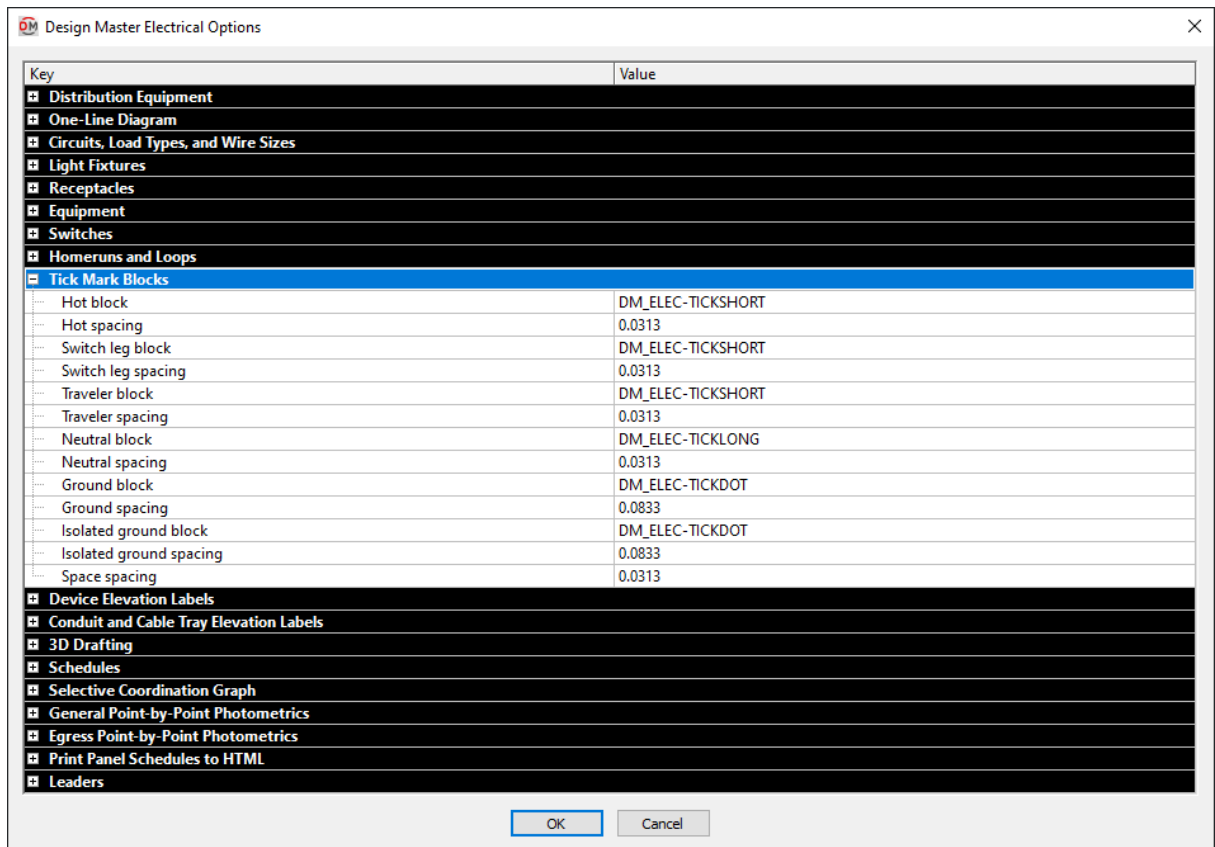
Hot block: The block used for tick marks representing the hot wires.

Hot spacing: The distance between hot tick marks and other tick marks. This value is multiplied by `DIMSCALE` to get the final tick mark spacing distance.

Switch leg block: The block used for tick marks representing the switch leg wires.

Switch leg spacing: The distance between switch leg tick marks and other tick marks. This value is multiplied by `DIMSCALE` to get the final tick mark spacing distance.

Traveler block: The block used for tick marks representing the traveler wires.

Traveler spacing: The distance between traveler tick marks and other tick marks. This value is multiplied by `DIMSCALE` to get the final tick mark spacing distance.

Neutral block: The block used for tick marks representing the neutral wires.

Neutral spacing: The distance between neutral tick marks and other tick marks. This value is multiplied by `DIMSCALE` to get the final tick mark spacing distance.

Ground block: The block used for tick marks representing the ground wires.

Ground spacing: The distance between ground tick marks and other tick marks. This value is multiplied by `DIMSCALE` to get the final tick mark spacing distance.

Isolated ground block: The block used for tick marks representing the isolated ground wires.

Isolated ground spacing: The distance between isolated ground tick marks and other tick marks. This value is multiplied by `DIMSCALE` to get the final tick mark spacing distance.

Space spacing: The extra space inserted between tick marks when a space is included. This value is multiplied by `DIMSCALE` to get the final tick mark spacing distance.

Device Elevation Labels

This section describes the options available in the **Device Elevation Labels** group in the **Design Master Electrical Options** dialog box.
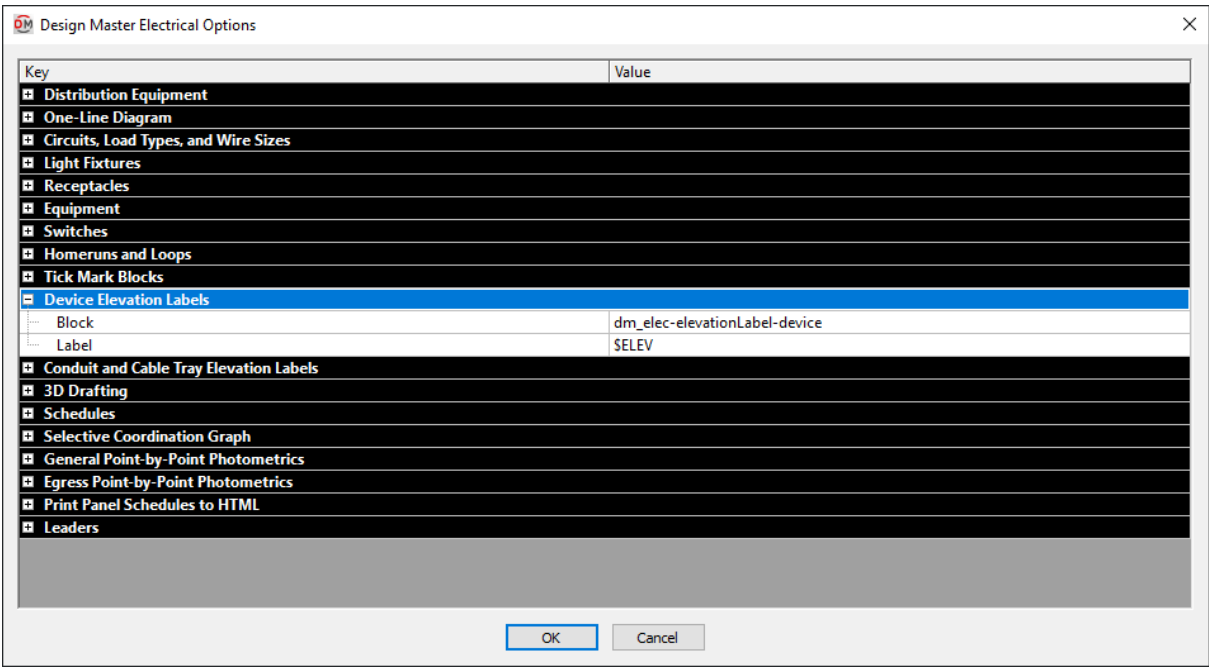
Block: The block used for the elevation label when the [Insert Device Elevation Label](#) command is used.

Label: The text that is used to display the elevation in the label. **$ELEV** is replaced with the elevation.

Conduit and Cable Tray Elevation Labels

This section describes the options available in the **Conduit and Cable Tray Elevation Labels** group in the **Design Master Electrical Options** dialog box.
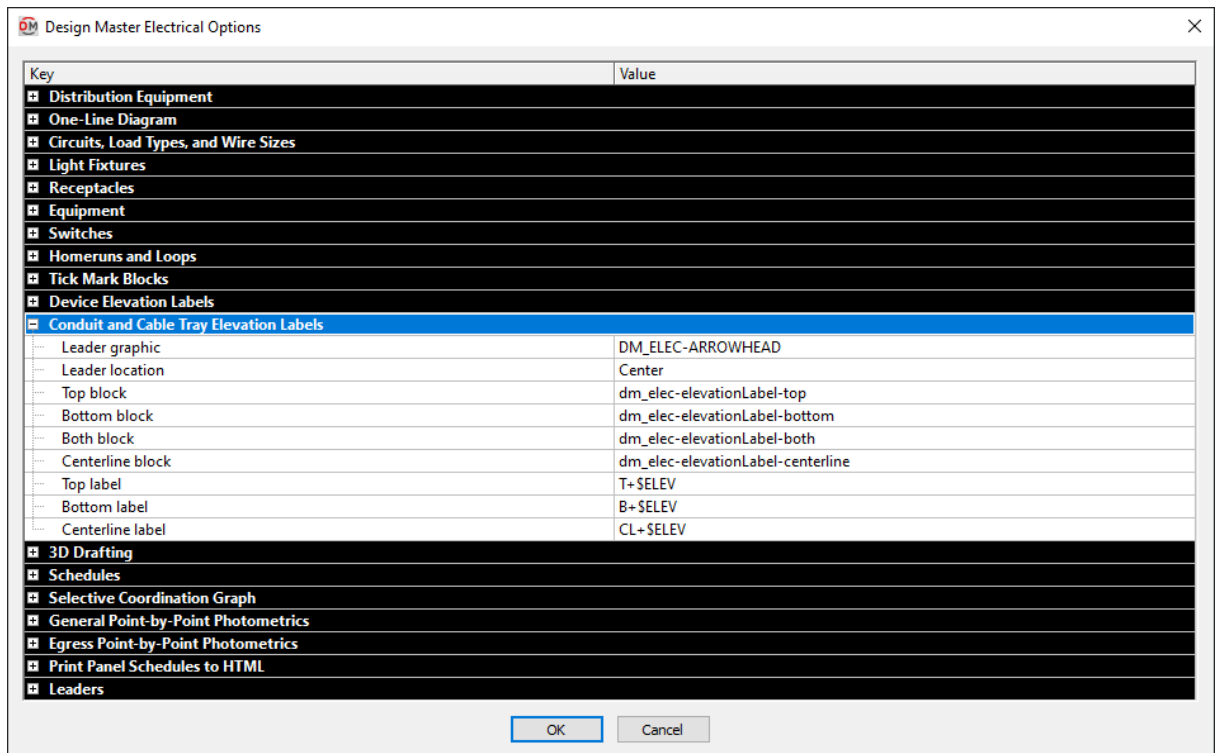
Leader graphic: The block that is used for elevation label leaders.

Leader location: Where the leader points on the conduit or cable tray.
- **Edge:** The leader points to the edge of the conduit or cable tray as shown below.
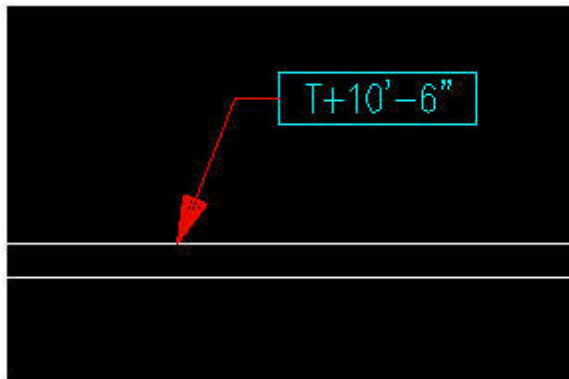
- **Center:** The leader points to the center of the conduit or cable tray as shown below.
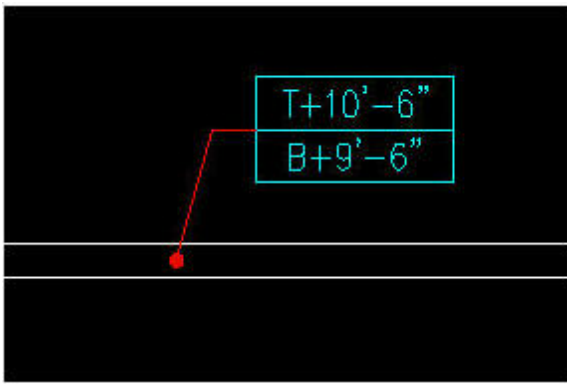
Top block: The block used for the elevation label when the **Insert Top Elevation Label** command is used.

Bottom block: The block used for the elevation label when the **Insert Bottom Elevation Label** command is used.

Both block: The block used for the elevation label when the **Insert Top and Bottom Elevation Label** command is used.

Centerline block: The block used for the elevation label when the **Insert Centerline Elevation Label** command is used.

Top label: The text that is used to display the top elevation in the label. **$ELEV** is replaced with the elevation.

Bottom label: The text that is used to display the top elevation in the label. **$ELEV** is replaced with the elevation.

Centerline label: The text that is used to display the top elevation in the label. **$ELEV** is replaced with the elevation.

3D Drafting

This section describes the options available in the **3D Drafting** group in the **Design Master Electrical Options** dialog box.
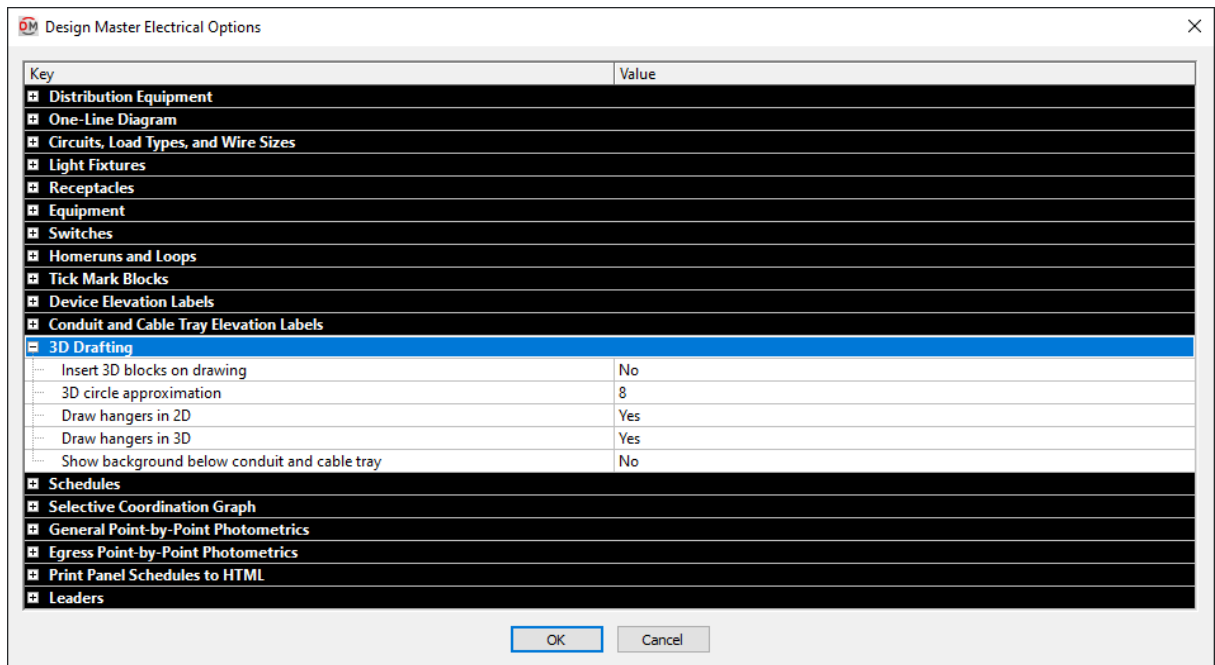
Insert 3D blocks on drawing: Whether a 3D block is inserted when a 2D block is inserted.

It is important to select **No** if you do not want to see the blocks. Do not freeze or turn off the layer using standard CAD commands. If the 3D layer is frozen or turned off, devices will no longer copy properly.

This setting has no impact on exporting devices to IFC files.

3D circle approximation: Circles are approximated in the 3D model as polygons. This option sets the number of sides of a polygon used to represent a circle. Higher numbers more closely approximate a circle, but greatly increase file size. The minimum number value for this option is 8.

Draw hangers in 2D: Whether hangers are drawn in 2D.

Draw hangers in 3D: Whether hangers are drawn in 3D. This setting has no impact on exporting hangers to an IFC file. It only affects whether they are visible in your CAD program.

Show background below conduit and cable tray: Whether the architectural background that appears behind conduits and cable trays is visible.

Schedules

This section describes the options available in the **Schedules** group in the **Design Master Electrical Options** dialog box.
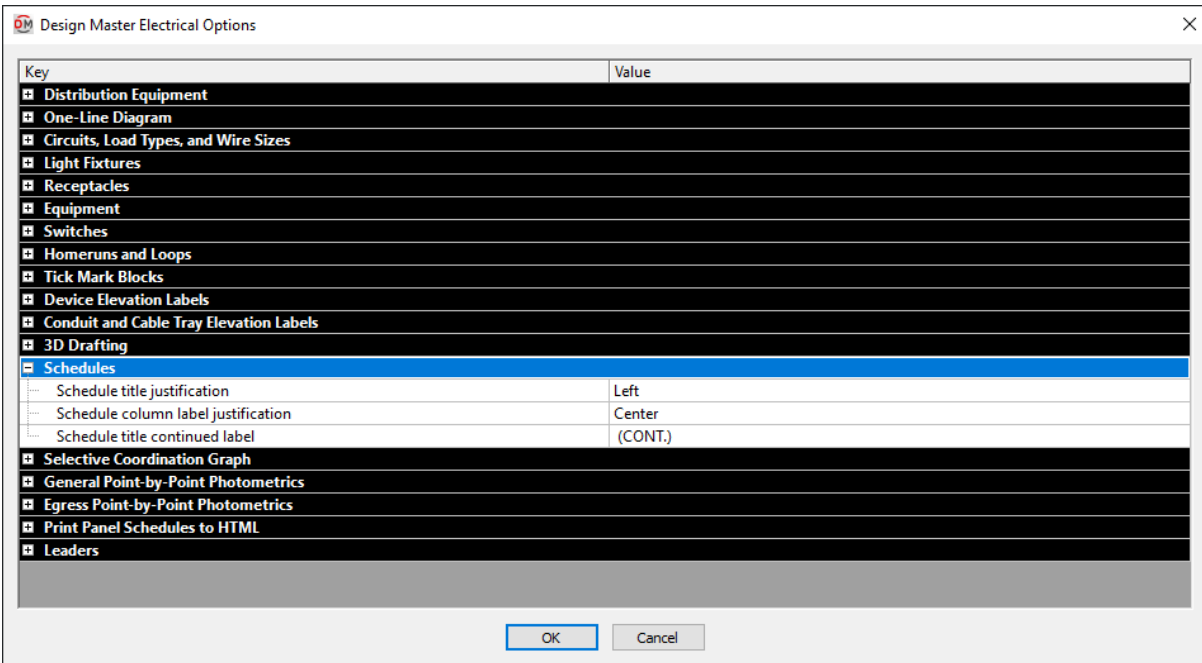
Schedule title justification: The justification for the title at the top of all of the schedules.
- **Left:** The schedule title is justified to the left.
- **Center:** The schedule title is centered.

Schedule column label justification: The justification for the label at the top of each column in all of the schedules.
- **Left:** The column labels are justified to the left.
- **Center:** The column labels are centered.

Schedule title continued label: The text in this option is added to the schedule title when the schedule is broken into multiple sections.

When inserting schedules, there is an option to set a maximum height for the schedule. If the schedule exceeds this height, the schedule is continued in a second section next to the first. The title of the second and following sections is the schedule title and the text in the *Schedule title continued label* option.

Selective Coordination Graph

This section describes the options available in the **Selective Coordination Graph** group in the **Design Master Electrical Options** dialog box.
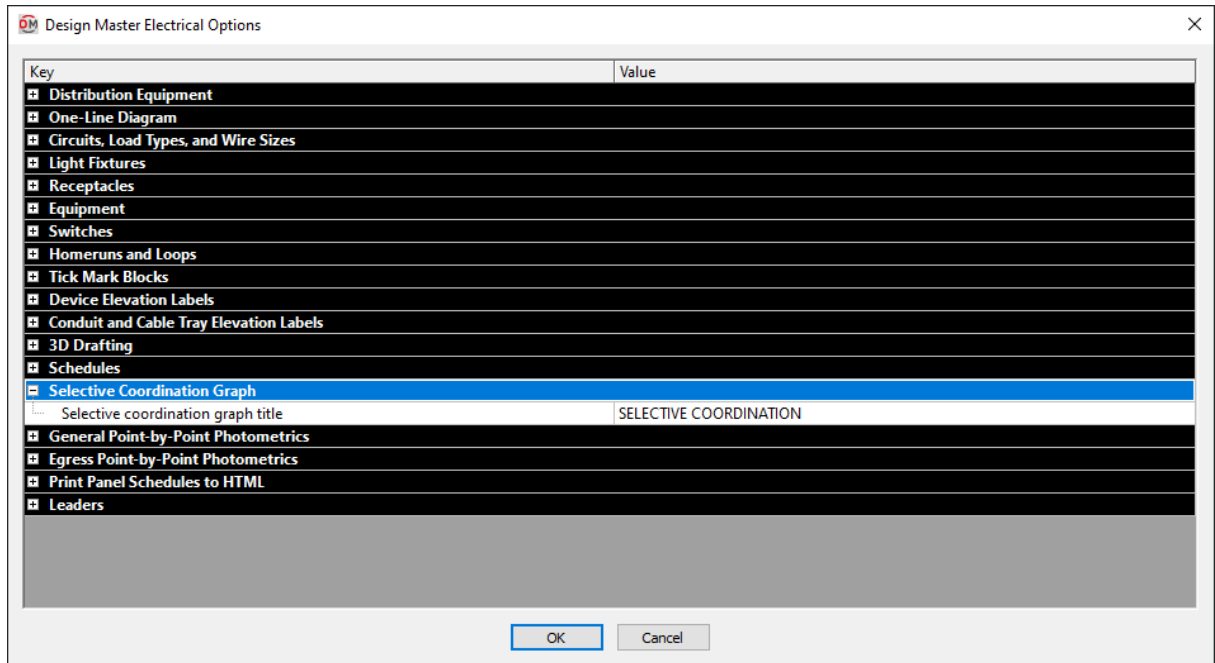
Selective coordination graph title: The default text that is entered in the *Title* field when running the **Insert Selective Coordination Graph** command.

General Point-by-Point Photometrics

This section describes the options available in the **General Point-by-Point Photometrics** group in the **Design Master Electrical Options** dialog box.

Most of these options set the default values that appear when running the **Calculate General Photometrics** command. These values can be changed in the **Calculate General Photometrics** dialog box.

Copy IES files to project folder: Whether IES files loaded into the **Light Fixture Project Schedule** are copied to the project folder.

Default grid spacing: The default *Grid Spacing* value that will be used when creating a new general photometric calculation area or line. This value can be changed when the area or line is created.

Default vertical grid spacing: The default *Vertical Calculation Spacing* value that will be used when creating a new general photometric calculation area or line. This value can be changed when the area or line is created.

Default vertical display spacing: The default *Vertical Display Spacing* value that will be used when creating a new general photometric calculation area or line. This value can be changed when the area or line is created.

Egress Point-by-Point Photometrics

This section describes the options available in the **Egress Point-by-Point Photometrics** group in the **Design Master Electrical Options** dialog box.

Most of these options set the default values that appear when running the **Calculate Egress Photometrics** command. These values can be changed in the **Calculate Egress Photometrics** dialog box.

Default grid spacing: The default *Grid Spacing* value that will be used when creating a new egress photometric calculation area or line. This value can be changed when the area or line is created.

Default vertical grid spacing: The default *Vertical Calculation Spacing* value that will be used when creating a new egress photometric calculation area or line. This value can be changed when the area or line is created.

Default vertical display spacing: The default *Vertical Display Spacing* value that will be used when creating a new egress photometric calculation area or line. This value can be changed when the area or line is created.

Print Panel Schedules to HTML

This section describes the options available in the **Print Panel Schedules to HTML** group in the **Design Master Electrical Options** dialog box.

These options set the default values and settings that appear when running the **Print Distribution Equipment Schedule** command. These values can be changed in the **Print Panel Schedule to HTML** dialog box.

Leaders

This section describes the options available in the **Leaders** group in the **Design Master Electrical Options** dialog box.
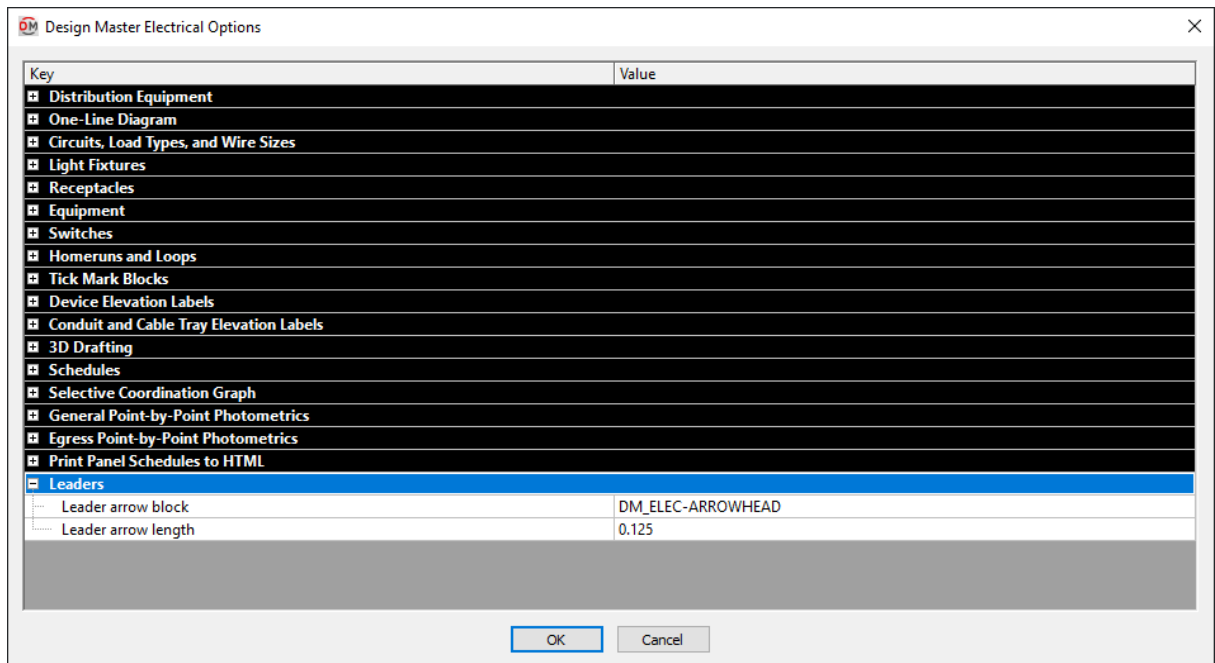
Leader arrow block: The block used for leader arrowheads.

Leader arrow length: The length of the leader arrowhead.

Block Creation

This section describes the commands used to create custom blocks.

Plan view blocks represent devices on your drawings. They are 2D representations of the devices.

3D blocks represent devices in your 3D model.

One-line blocks represent items on your one-line diagram.

Training Videos

- **One-Line Diagram Block Customization**
- **Panel Schedule Customization**
- **Creating Custom Blocks**

Common Block Information

Description: The name of the block that is displayed when you select the block in the schedule dialog box.

Block Name: The file name of the block displayed on the plan.

3D Block Name: The file name of the block displayed in 3D. You can enter a specific file name, or select **<2D-Block Name>-3d**. If you select this name, the 3D block name will be the same as the *Block Name* field, but with **-3D** appended to the end of the file name.

To Scale: Whether the block is to scale.

Blocks that are to scale are inserted on the drawing at their actual size. The scale factor for the block will be 1. Changing **DIMSCALE** will not change the size of the block. Light fixtures and panels are commonly inserted to scale.

Blocks that are not to scale will vary in size depending upon the scale of the drawing. The scale factor of the block will be equal to the **DIMSCALE** setting of the drawing. Changing **DIMSCALE** will change the size of the device. Receptacles and switches are commonly not inserted to scale.

Can Rotate: Whether the block can be rotated.

When a block that can be rotated is inserted, you are prompted to specify the rotation angle. They can be rotated using the standard CAD **ROTATE** command.

When a block that cannot be rotated is inserted, you are not prompted to specify the rotation angle. The block is inserted with a rotation angle of 0. If you attempt to rotate the block, the rotation angle will be reset to 0.

Create Plan View Block from Entities

The `Create Plan View Block from Entities` command is used to create a 2D block that can be used in your plans to represent a device. Specific entities on the current drawing are used to create the block. The entities selected will be removed from the drawing during this process.

To create the block, go to

Ribbon: `DME: Customization->Block Creation->Create Plan View Block from Entities`

Pulldown Menu: `DM Electrical->Customization->Block Creation->Create Plan View Block from Entities`

Create Plan View Block from Entities Dialog Box

Block Name: The name of the block. This will be the name of the block on the drawing and the name of the drawing file saved to your computer.

Folder: The folder where the block will be saved. The default location is your customization folder.

Set Folder: Press this button to change the folder where the block will be saved.

Block Settings

Move entities to layer 0: Whether the entities you select for the block are moved to layer 0 or left on the layer they are currently on. If you are not sure what this means, leave this box checked.

See the help for your CAD program for more information about setting layers in blocks.

Block has a highlight: Whether the block has a highlight associated with it. Highlights are used to include a second layer and color in blocks. The highlight block is inserted on a separate layer from the main device block. The two layers can have different colors and other properties set on them.

Block can rotate: Whether the block can be rotated. See the **Can Rotate** section for more information.

Block is "To Scale": Whether the block is to scale. See the **To Scale** section for more information.

Connect loops and leaders to: Where loops and leaders are connected to the block.
- **Specific points:** The loops and leaders are connected to specific points on the block. Choose this option for blocks that are not circles or rectangles.
- **Edge of a circle:** The loops and leaders are connected to the edge of a circle. Choose this option if the block has a circle in it that you want the loops and leaders to connect to.
- **Edge of a rectangle:** The loops and leaders are connected to the edge of a rectangle. Choose this option if the block has a rectangle in it that you want the loops and leaders to connect to.

Slide Settings

Create slide: Whether a slide for the block will be created automatically. The slide is used in dialog boxes to display the graphics.

If you do not create a slide for a block, an "X" will be shown in the dialog boxes. The block will still display properly on the drawing.

You can manually create the slide later using the `Create Slide` command. See the **Create Slide** section for more information.

Block Color: The color to use for the block in the slide.

Select Block Color: Press this button to set *Block Color* using the standard CAD **Select Color** dialog box.

Highlight Color: The color to use for the highlight block in the slide.

Select Highlight Color: Press this button to set *Highlight Color* using the standard CAD **Select Color** dialog box.

Database Settings

Create records in database: Whether records will be created in the master and/or project databases.

Block Type: The type of device that the plan view block is used to represent.

Master Schedule Records

Create record in master block list: Whether the block is added to the master block list. If this box is checked, you can specify a label for the block in the master list. If you leave the label field blank, the block name is used as the label in the master list.

Create record in master schedule: Whether a record using the block is created in the master schedule. If this box is checked, you can specify a name for the entry in the master schedule. If you leave the name field blank, the block name is used as the name of the entry in the master schedule.

Master Schedule Group: The group with which the record will be associated in the master schedule.

Project Schedule Records

Create record in project block list: Whether the block is added to the project block list. If this box is checked, you can specify a label for the block in the project list. If you leave the label field blank, the block name is used as the label in the project list.

Create record in project schedule: Whether a record using the block is created in the project schedule. If this box is checked, you can specify a name for the entry in the project schedule. If you leave the name field blank, the block name is used as the name of the entry in the project schedule.

Project Schedule Group: The group with which the record will be associated in the project schedule. This field is disabled when the *Block Type* is set to **Equipment** or **Distribution Equipment**.

Enter the information needed to create the plan view block and press the **OK** button.

You will be prompted to select entities on the drawing to include in the block.

Select entities to use in the block:

If the block has a highlight, you will then be prompted to specify entities for the highlight block.

Select entities to use in the highlight block:

You will then prompted to specify the insertion point for the block.

Specify insertion base point:

You will then be prompted to specify where loops and leaders will connect to the block. The prompts depend upon the option you chose in the dialog box.

- **Specific points:** Specify the points where loops and leaders can connect to the block. You may specify as many points as you need.

  Specify connection point:

- **Edge of a circle:** Specify two points on opposite sides of the circle to connect the loops and leaders to.

  Specify point on one side of circle:
  Specify point on opposite side of circle:

- **Edge of a rectangle:** Specify two corners of the rectangle to connect the loops and leaders to.

  Specify first corner of rectangle:
  Specify opposite corner of rectangle:

The block, slide, and database records will be created for you. The entities selected for the block will be removed

from the drawing.

If a slide is created, the screen will flash during the creation process. When it is finished, you should return to your original view of the drawing.

Create 3D Block from Entities

The `Create 3D Block from Entities` command is used to create a 3D block that can be used in your 3D model to represent a device. Select an outline on the drawing that represents the block when viewed from above. This outline will be extruded to create a 3D solid. The entities selected will be removed from the drawing during this process.

To create a more complex 3D model for your device, use standard CAD commands.

To create the block, go to

Ribbon: `DME: Customization->Block Creation->Create 3D Block from Entities`

Pulldown Menu: `DM Electrical->Customization->Block Creation->Create 3D Block from Entities`

Create 3D Block from Entities Dialog Box

Block Type: The type of device blocks shown in *Block List*. Change this field to display a different list of blocks.

Block List: A list of all of the blocks based upon the *Block Type* field. When you select a block from this list, the *File Name* field is filled in with the name of the block with "-3D" appended to the end.

Using the list is optional to set the *File Name* field. You can set the name of the file manually and ignore the list.

File Name: The file name of the 3D block. This field can be set by selecting a block from the *Block List*.

Insertion Elevation Location: Where the elevation is measured on the 3D block. When you insert a device, you are prompted for the elevation. Where that elevation is measured on the device depends upon how the 3D block representing the device is defined.

- **Top:** The elevation is measured from the top of the device. The device will extend below the specified elevation.
- **Center:** The elevation is measured from the center of the device. The device will extend above and below the specified elevation.
- **Bottom:** The elevation is measured from the bottom of the device. The device will extend above the specified elevation.

Block Height: The height at which the block outline will be extruded, in inches.

Enter the information needed to create the 3D block and press the **OK** button.

You will be prompted to select the entity to use as the outline for the block. You can select either a closed polyline or a circle.

Select polyline or circle to extrude:

You will then be prompted to select any other entities you want to include in the block. These entities will be included exactly as they are drawn. They will not be extruded or otherwise modified.

Select other entities to include in the block:

You will then be prompted to specify the insertion point for the block. It is important that the insertion point for the block match the insertion point used for the plan view block.

Specify insertion point:

The block will be created for you. The entities selected for the block will be removed from the drawing.

Create Plan View Block from This Drawing

The `Create Plan View Block from This Drawing` command is used to configure your current drawing to work as a custom electrical block.

You can manually create a slide for the block using the `Create Slide` command. See the **Create Slide** section for more information.

To configure the current drawing, go to

Ribbon: `DME: Customization->Block Creation->Create Plan View Block from This Drawing`

Pulldown Menu: `DM Electrical->Customization->Block Creation->Create Plan View Block from This Drawing`

You will be prompted to specify the insertion point for the block.

Specify insertion base point:

The **Design Master Electrical** dialog box shown below will appear.
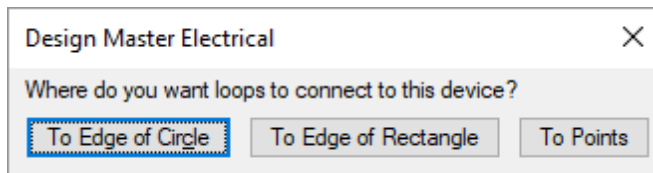
Press the **To Edge of Circle** button to connect loops to the edges of a circle on this block. Press the **To Edge of Rectangle** button to connect loops to the edges of a rectangle on this block. Press the **To Points** button to connect loops to points on this block.

Connecting Loops to the Edge of a Circle

If the **To Edge of Circle** button is pressed, you will be prompted to specify a point on one side of the circle.

Specify a point on one side of circle:

You will then be prompted to identify a point on the opposite side of the circle.

Specify a point on opposite side of circle:

Connecting Loops to the Edge of a Rectangle

If the **To Edge of Rectangle** button is pressed, you will be prompted to identify the location of the first corner of the rectangle.

Specify first corner of rectangle.

You will then be prompted to identify the location of the opposite corner.

Specify opposite corner of rectangle.

Connecting Loops to Points on the Device

If the **To Points** button is pressed, you will be prompted to identify the location where the loop is to connect to the device.

Specify connection point:

Specify as many connection points as desired for the block. Press **ENTER** to finish the command.

Create Slide

The `Create Slide` command is used to create a slide using an area of your drawing. A slide is an image file that is displayed in dialog boxes to represent the block that is selected.

The slide file must have the same name as the block that it represents. If a block is selected that does not have a corresponding slide, an "X" will be shown in the dialog box instead. The block will still insert properly if a corresponding slide does not exist.

To create a slide, go to

Ribbon: `DME: Customization->Block Creation->Create Slide`

Pulldown Menu: `DM Electrical->Customization->Block Creation->Create Slide`

Save Slide As Dialog Box
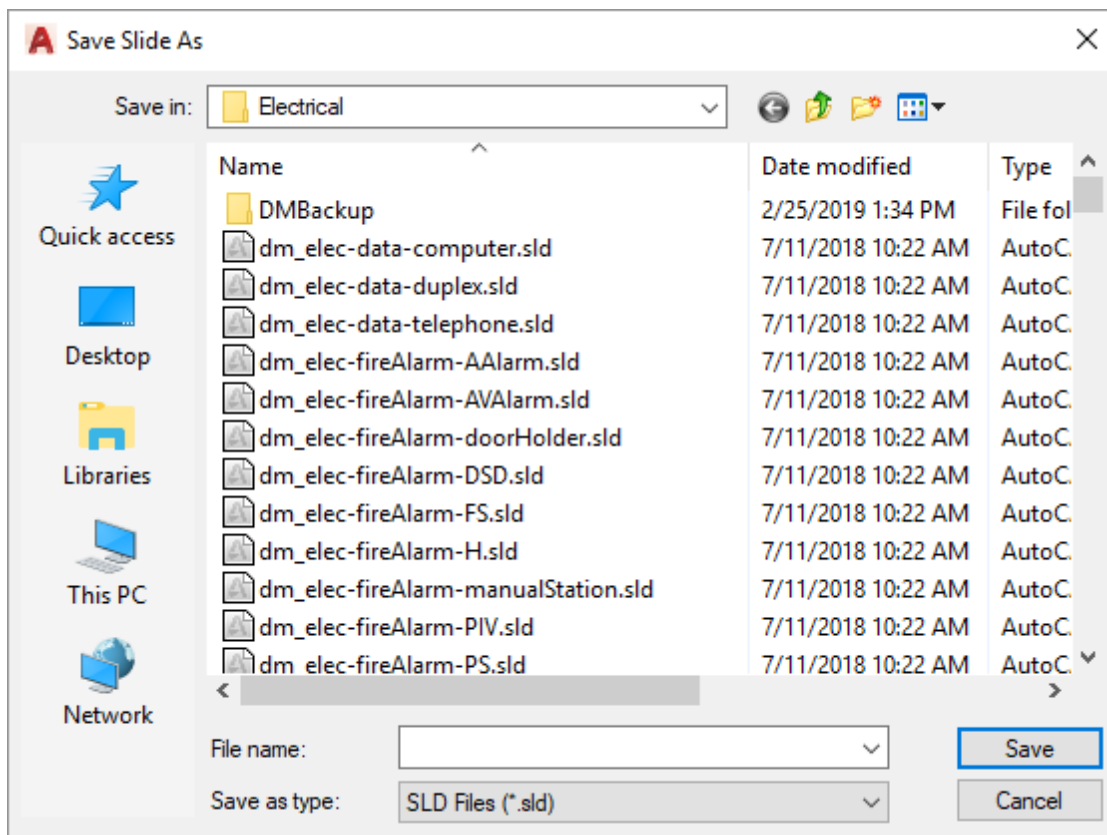
Specify the name of the slide file to create, then press the **Save** button to create the slide.

You will be prompted to draw a box on your drawing. A snapshot of everything inside the box will be saved in the slide.

Specify first corner of slide:

You will then be asked to specify the opposite corner of the box.

Specify second corner:

The screen will flash while the slide is being created.

Insert Non-Rotating Text

The `Insert Non-Rotating Text` command is used to create text that can be included in a custom block that will not rotate when the block is rotated.

Normal CAD text rotates when a block is rotated. If a block is rotated 180 degrees, the text will appear upside-down.

Text created using this command will always be displayed at angle 0. When the block is rotated, the text is rotated back to 0.

To insert non-rotating text on the current drawing, go to

Ribbon: `DME: Customization->Block Creation->Insert Non-Rotating Text`

Pulldown Menu: `DM Electrical->Customization->Block Creation->Insert Non-Rotating Text`

You will be prompted to identify where the non-rotating text is to be inserted on the drawing.

Insertion point for non-rotating text:

The point specified will be the center point for the inserted text. You will then be prompted to enter the text itself.

Text value:

You will then be prompted to specify the height of the text.

Text height:

You will then be prompted to specify whether the text will be used in a one-line diagram block.
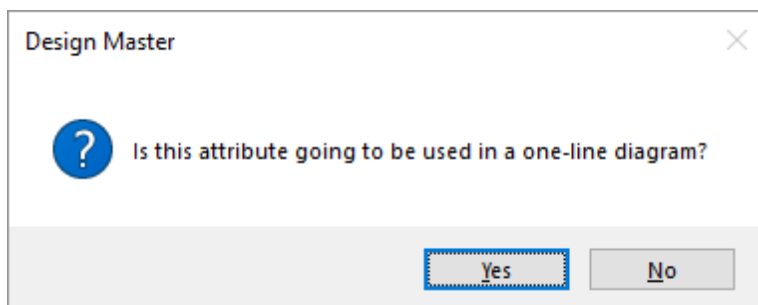
Select the appropriate response for your intended use of the text. The text will be inserted on the drawing.

Once the text is inserted, you can modify it using standard CAD commands.

Make sure you select the text when choosing entities to create a block.

Create One-Line Diagram Block from Entities

The `Create One-Line Diagram Block from Entities` command is used to create a block that can be used on your one-line diagram. Specific entities on the current drawing are used to create the block. The entities selected will be removed from the drawing during this process.

To create the block, go to

Ribbon: `DME: Customization->Block Creation->Create One-Line Diagram Block from Entities`

Pulldown Menu: `DM Electrical->Customization->Block Creation->Create One-Line Diagram Block from Entities`

Create One-Line Diagram Block Dialog Box

Block Drawing File Name: The name of the drawing file for the block.

Folder: The folder where the block will be saved. The default location is your customization folder.

Set Folder: Press this button to change the folder where the block will be saved.

Move entities to layer 0: Whether the entities you select for the block are moved to layer 0 or left on the layer they are currently on. If you are not sure what this means, leave this box checked.

See the help for your CAD program for more information about setting layers in blocks.

Block Type: The type of device that the one-line diagram block is used to represent.

If you select **Feeder** or **Meter**, the connection point settings below will be disabled.

Upstream Connection Points: Sets the number of upstream connection points for the block.

If **None** is selected and an upstream connection is required on the one-line diagram, the feeder will connect to the insertion point of the block.

Select **Two** if the block will be used to represent transfer switches or UPSs.

Upstream Feeder Connection Direction: The direction from which the upstream feeder connects to the block.

Downstream Connection Point: Whether the block has a downstream connection point.

If **No** is selected and a downstream connection is required on the one-line diagram, the feeder will connect to the insertion point of the block.

Downstream Feeder Connection Direction: The direction from which the downstream feeder connects to the block.

Slide Settings

Create slide: Whether a slide for the block will be created automatically. The slide is used in dialog boxes to display the graphics.

If you do not create a slide for a block, an "X" will be shown in the dialog boxes. The block will still display properly on the drawing.

You can manually create the slide later using the Create Slide command. See the **Create Slide** section for more information.

Block Color: The color to use for the block in the slide.

Select Block Color: Press this button to set *Block Color* using the standard CAD **Select Color** dialog box.

Database Settings

Create record in project database: Whether the block is added to the project database.

Create record in master database: Whether the block is added to the master database.

Display Name: The name of the block on the drawing and in the selected databases.

Enter the information needed to create the one-line diagram block and press the **OK** button.

If you did not set *Upstream Connection Points* to **None**, you will be prompted to specify the upstream connection point for the block.

Specify upstream connection point:

If you set *Upstream Connection Points* to **Two**, you will be prompted to specify a second upstream connection point.

If you set *Downstream Connection Point* to **Yes**, you will then be prompted to specify the downstream connection point for the block.

Specify downstream connection point:

If you set *Block Type* to **Feeder** or **Meter**, you will then be prompted to specify the mirroring point. This point is displayed as a grip when the block is inserted and is used to mirror the block.

Specify mirroring point:

The block, slide, and database records will be created for you. The entities selected for the block will be removed from the drawing.

If a slide is created, the screen will flash during the creation process. When it is finished, you should return to your original view of the drawing.

Create One-Line Diagram Block from this Drawing

The `Create One-Line Diagram Block from This Drawing` command is used to configure your current drawing to work as a custom block on your one-line diagram.

You can manually create a slide for the block using the `Create Slide` command. See the **Create Slide** section for more information.

To configure the current drawing, go to

Ribbon: `DME: Customization->Block Creation->Create One-Line Diagram Block from This Drawing`

Pulldown Menu: `DM Electrical->Customization->Block Creation->Create One-Line Diagram Block from This Drawing`

A **Design Master Electrical** dialog box will appear, asking if the block will be used as a feeder or meter block.

Yes: The block will be a feeder or meter block. Feeder blocks are able to be inserted at the ends or in the middle of feeders. The block can also be used to represent distribution equipment or equipment connections on the one-line diagram.

No: The block will not be a feeder or meter block. The block can only be used to represent distribution equipment or equipment connections on the one-line diagram. It cannot be used in a feeder.

Creating a Feeder Block

You will be prompted to specify the location of the upstream connection point. This is the point that the feeder from the upstream device will connect to by default.

Specify upstream connection point:

You will then be prompted to specify the location of the downstream connection point. This is the point that the feeder to the downstream device will connect to by default.

Specify downstream connection point:

You will then be prompted to specify the mirroring point. This point is displayed as a grip when the block is inserted. It is used to mirror the block over the feeder line.

Specify mirroring point:

Creating a Non-Feeder Block

A **Design Master Electrical** dialog box will appear, asking whether the block has an upstream connection point. If you press the **Yes** button, you will be prompted to specify the location of the upstream connection point. This is the point that the feeder from the upstream device will connect to by default.

Specify upstream connection point:

You will then be asked if the block has a second upstream connection point. If you press the **Yes** button, you will be prompted to specify its location. The second point is used for the second upstream connection on transfer switches and UPSs.

If an upstream connection point is not inserted but is required on the one-line diagram, the feeder will connect to the insertion point of the block.

A **Design Master Electrical** dialog box will appear, asking whether the block has a downstream connection point. If you press the **Yes** button, you will be prompted to specify the location of the downstream connection point. This is the point that the feeder to the downstream device will connect to by default.

Specify downstream connection point:

If a downstream connection point is not inserted but is required on the one-line diagram, the feeder will connect to the insertion point of the block.

Create One-Line Diagram Block from One-Line Diagram

The `Create One-Line Diagram Block from One-Line Diagram` command is used to configure entities on a one-line diagram to work as a custom block on other one-line diagrams. This command can be used to create blocks that will be inserted with labels.

You can manually create a slide for the block using the `Create Slide` command. See the **Create Slide** section for more information.

To configure a one-line diagram block, go to

Ribbon: `DME: Customization->Block Creation->Create One-Line Diagram Block from One-Line Diagram`

Pulldown Menu: `DM Electrical->Customization->Block Creation->Create One-Line Diagram Block from One-Line Diagram`

Create One-Line Diagram Block

Block Drawing File Name: The name of the drawing file for the block.

Folder: The folder where the block will be saved. The default location is your customization folder.

Set Folder: Press this button to change the folder where the block will be saved.

Move entities to layer 0: Whether the entities you select for the block are moved to layer 0 or left on the layer they are currently on. If you are not sure what this means, leave this box checked.

See the help for your CAD program for more information about setting layers in blocks.

Block Type: The type of device that the one-line diagram block is used to represent.

Database Settings

Create record in project database: Whether the block is added to the project database.

Create record in master database: Whether the block is added to the master database.

Display Name: The name of the block on the drawing and in the selected databases.

Enter the information needed to create the one-line diagram block and press the **OK** button.

You will be prompted to specify a distribution equipment or feeder block.

Select distribution equipment or feeder to create block from:

You must specify a block that corresponds to the selected *Block Type*.

You will then be prompted to select any other entities you want to add to the block.

Select other entities to add to block:

If you selected **Feeder** or **Meter** for the *Block Type*, you will then be prompted to specify a label to include in the block.

Select label to include in block:

The block will be created for you.

Insert Distribution Equipment Schedule Attribute

The `Insert Distribution Equipment Schedule Attribute` command is used when customizing distribution equipment schedule blocks. This command is used to insert attributes that will be filled in with a value by the **[Update Distribution Equipment Schedule](#)** command.

To insert a distribution equipment schedule attribute, go to

Ribbon: `DME: Customization->Block Creation->Insert Distribution Equipment Schedule Attribute`

Pulldown Menu: `DM Electrical->Customization->Block Creation->Insert Distribution Equipment Schedule Attribute`

Panel Schedule Attributes Dialog Box
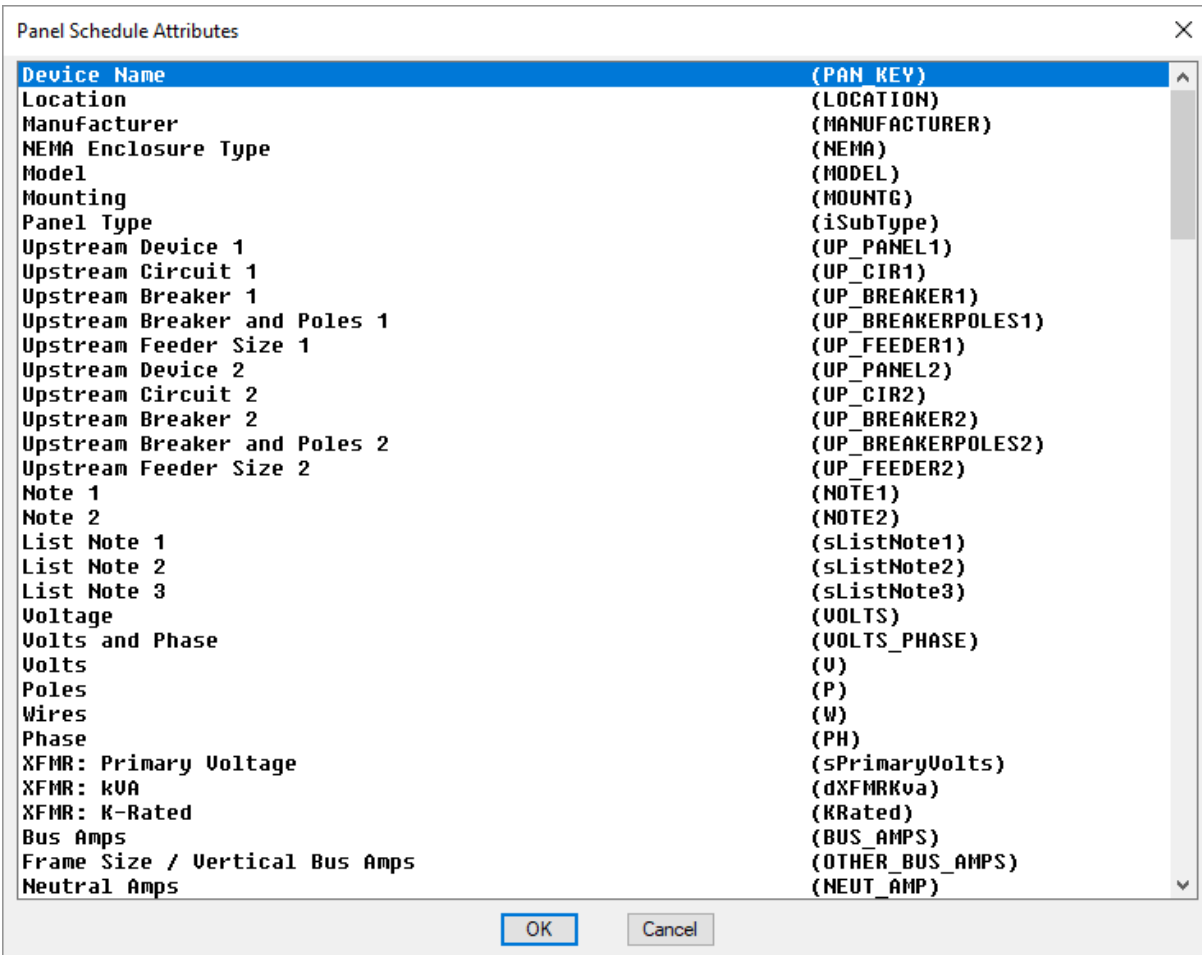
The dialog box is a list of attributes you can insert. Two values are listed in each row.

The first value is a description of the attribute.

The second value, in parenthesis, is the label of the attribute on the drawing. Use this value to compare what is already on your drawing with what is available to be inserted.

Select the attribute you want to insert and press the **OK** button. You will be prompted to specify the insertion point for the attribute.

The following fields are available to be inserted:

Device Name (PAN_KEY): Name of the distribution equipment. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Location (LOCATION): Location of the distribution equipment. Specified in the distribution equipment definition. See the **Notes Settings** section for more information.

Manufacturer (MANUFACTURER): Manufacturer of the distribution equipment. Specified in the distribution equipment definition. See the **Notes Settings** section for more information.

NEMA Enclosure Type (NEMA): NEMA rating of the distribution equipment. Specified in the distribution equipment definition. See the **Notes Settings** section for more information.

Model (MODEL): Model of the distribution equipment. Specified in the distribution equipment definition. See the **Notes Settings** section for more information.

Mounting (MOUNTG): Mounting of the distribution equipment. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Panel Type (iSubType): Lists the *Main Disconnect Type* for panels, switchboards, motor control centers, and meter centers. Lists the *Type* for enclosed breakers/disconnects and transfer switches. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Upstream Device 1 (UP_PANEL1): Name of the distribution equipment connected to source 1. Set using the the `Connect Distribution Equipment` command. See the **Connect Distribution Equipment** section for more information.

Upstream Circuit 1 (UP_CIR1): Circuit number connected to source 1. Set using the the `Connect Distribution Equipment` command. See the **Connect Distribution Equipment** section for more information.

Upstream Breaker 1 (UP_BREAKER1): Size of the breaker connected to source 1. Specified in the distribution equipment definition. See the **Feeder Specification Settings** section for more information.

Upstream Breaker and Poles 1 (UP_BREAKERPOLES1): The breaker and numbers of poles connected to source 1. Specified in the distribution equipment definition.See the **Feeder Specification Settings** section for more information.

Upstream Feeder Size 1 (UP_FEEDER1): Feeder callout connected to source 1. Specified in the distribution equipment definition. See the **Feeder Specification Settings** section for more information.

Upstream Device 2 (UP_PANEL2): Name of the distribution equipment connected to source 2. Set using the the `Connect Distribution Equipment` command. See the **Connect Distribution Equipment** section for more information. Source 2 is used for UPSs and transfer switches.

Upstream Circuit 2 (UP_CIR2): Circuit number connected to source 2. Set using the the `Connect Distribution Equipment` command. See the **Connect Distribution Equipment** section for more information. Source 2 is used for UPS's and transfer switches.

Upstream Breaker 2 (UP_BREAKER2): Size of the breaker connected to source 2. Specified in the distribution equipment definition. See the **Feeder Specification Settings** section for more information. Source 2 is used for UPS's and transfer switches.

Upstream Breaker and Poles 2 (UP_BREAKERPOLES2): The breaker and numbers of poles connected to source 2. Specified in the distribution equipment definition. See the **Feeder Specification Settings** section for more information. Source 2 is used for UPS's and transfer switches.

Upstream Feeder Size 2 (UP_FEEDER2): Feeder callout connected to source 2. Specified in the distribution equipment definition. See the **Feeder Specification Settings** section for more information. Source 2 is used for UPS's and transfer switches.

Note 1 (NOTE1): Note 1 for the distribution equipment. Specified as the *Header Note* in the distribution equipment definition. See the **Notes Settings** section for more information.

Note 2 (NOTE2): Note 2 for the distribution equipment. Specified as the *Footer Note* in the distribution equipment definition. See the **Notes Settings** section for more information.

List Note 1 (sListNote1): List Note 1 for the distribution equipment. Specified in the distribution equipment definition. See the **Notes Settings** section for more information.

List Note 2 (sListNote2): List Note 2 for the distribution equipment. Specified in the distribution equipment definition. See the **Notes Settings** section for more information.

List Note 3 (sListNote3): List Note 3 for the distribution equipment. Specified in the distribution equipment definition. See the **Notes Settings** section for more information.

Voltage (VOLTS): The full voltage designation for the distribution equipment. It includes the voltage, the number of poles, and the number of wires. For example, **120V 1P 2W** or **208V 3P 4W**. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Volts and Phase (VOLTS_PHASE): The voltage designation for the distribution equipment with the phase listed instead of the number of poles. For example, **120V 1-PH 2W** or **208V 3-PH 4W**. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Volts (V): The voltage designation for the distribution equipment. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Poles (P): The number of poles for the distribution equipment. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Wires (W): The number of wires for the distribution equipment. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Phase (PH): The number of phases for the distribution equipment. Specified in the distribution equipment definition. See the **General Settings** section for more information.

XFMR: Primary Voltage (sPrimaryVolts): Voltage of the transformer. Specified in the distribution equipment definition. See the **General Settings** section for more information. Only applies to transformers.

XFMR: kVA (dXFMRKva): kVA rating of the transformer. Specified in the distribution equipment definition. See the **General Settings** section for more information. Only applies to transformers.

XFMR: K-Rated (KRated): Whether the transformer is k-rated. Displays **K-RATED** if it is, nothing if it is not. Specified in the distribution equipment definition. See the **General Settings** section for more information. Only applies to transformers.

Bus Amps (BUS_AMPS): Bus amps of the distribution equipment. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Frame Size / Vertical Bus Amps (OTHER_BUS_AMPS): Frame size or vertical bus amps of the distribution equipment. Frame size is used for enclosed breakers / disconnects. Vertical bus amps is used for motor control centers. This value is not used for other distribution equipment types. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Neutral Amps (NEUT_AMP): Neutral amps of the distribution equipment. Specified in the distribution equipment definition. See the **General Settings** section for more information.

AIC Rating (AIC): AIC Rating of the distribution equipment. Specified in the distribution equipment definition. See the **Fault Analysis Settings** section for more information.

Total Fault (dFaultTotal): Total fault at the distribution equipment. See the **Insert Fault Schedule** section for information about how this is calculated.

Main Trip Amps (MAIN): Trip size of the main breaker. Displays **MLO** if the distribution equipment is main lugs only. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Main Frame Amps (MAIN_FRAME): Frame size of the main breaker. Displays **MLO** if the distribution equipment is main lugs only. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Lugs (LUGS): Lugs type of the distribution equipment. Specified in the distribution equipment definition. See the **General Settings** section for more information.

IG Bus (IG_BUS): Whether the distribution equipment has an IG bus. Displays **ISO GND BUS** if it does, nothing if it does not. Specified in the distribution equipment definition. See the **General Settings** section for more information.

UPS: Input kVA (INPUT_KVA): The kVA input of the distribution equipment. This value is only used for UPSs and phase inverters. Specified in the distribution equipment definition. See the **General Settings** section for more information.

UPS: Efficiency (EFFICIENCY): The efficiency of the distribution equipment. This value is only used for UPSs and phase inverters. Specified in the distribution equipment definition. See the **General Settings** section for more information.

UPS / Generator: kVA (KVA): The kVA output of the distribution equipment. This value is only used for UPSs, phase inverters, and generators. Specified in the distribution equipment definition. See the **General Settings** section for more information.

UPS / Generator: kW (KW): The kW output of the distribution equipment. This value is only used for UPSs, phase inverters, and generators. Specified in the distribution equipment definition. See the **General Settings** section for more information.

UPS / Generator: Power Factor (POWER_FACTOR): The power factor of the distribution equipment. This value is only used for UPSs, phase inverters, and generators. Specified in the distribution equipment definition. See the **General Settings** section for more information.

Circuit: Breaker Size / Poles (B1/B2): Breaker size and number of poles of the circuit.

Circuit: Breaker Size (BREAKERAMPS1/BREAKERAMPS2): Breaker size of the circuit.

Circuit: Poles (POLES1/POLES2): Number of poles of the circuit.

Circuit: Description (D1/D2): Circuit description of the circuit.

Circuit: Description End (DEND): The location where the circuit description should end. The circuit description width is calculated based upon the distance between the *D1* and the *DEND* attributes. Circuit descriptions that exceed this length are wrapped to create additional lines.

Circuit: Total Load (L1/L2): The total load of the circuit, in kVA. It will include the load of all phases if it is a two- or three-phase load.

Circuit: Total Load (Amps) (L-AMPS1/L-AMPS2): The total load of the circuit, in amps. It will include the load of all phases if it is a two- or three-phase load.

Circuit: Phase Load (PL1/PL2): The phase load of the circuit, in kVA.

Circuit: Phase Load (Amps) (PL-AMPS1/PL-AMPS2): The phase load of the circuit, in amps.

Circuit: Feeder Size (F1/F2): The wire callout of the specific circuit. This value includes the size of the conduit and all of the wires.

Circuit: Wire Size (WIRE1/WIRE2): The phase wire size of the circuit.

Circuit: Receptacle Count (REC_COUNT1/REC_COUNT2): The number of receptacles connected to the circuit.

Circuit: Light Fixture Count (LTG_COUNT1/LTG_COUNT2): The number of light fixtures connected to the circuit.

Circuit: Equipment Connection Count (EQU_COUNT1/EQU_COUNT2): The number of equipment connections on the circuit.

Circuit: Underground (UNDERGROUND1/UNDERGROUND2): Whether the circuit is underground.

Circuit: Conduit (CONDUIT1/CONDUIT2): The conduit wire size of the circuit.

Circuit: Ground (GROUND1/GROUND2): The ground wire size of the circuit.

Total Lug Load (LUG_LOAD): The total lug load connected to the distribution equipment in kVA.

Lug Load, Phase A (LUG_PHA): The lug load on phase A in kVA.

Lug Load, Phase B (LUG_PHB): The lug load on phase B in kVA.

Lug Load, Phase C (LUG_PHC): The lug load on phase C in kVA.

Phase A KVA (PHA_KVA): The connected load on phase A in kVA.

Phase B KVA (PHB_KVA): The connected load on phase B in kVA.

Phase C KVA (PHC_KVA): The connected load on phase C in kVA.

Phase A Amps (100%) (PHA_AMP): The connected load on phase A in amps.

Phase B Amps (100%) (PHB_AMP): The connected load on phase B in amps.

Phase C Amps (100%) (PHC_AMP): The connected load on phase C in amps.

Phase A Amp Percentage (PHA_AMPPERCENTAGE): The percentage load of phase A compared to the average load on all phases.

Phase B Amp Percentage (PHB_AMPERCENTAGE): The percentage load of phase B compared to the average load on all phases.

Phase C Amp Percentage (PHC_AMPERCENTAGE): The percentage load of phase C compared to the average load on all phases.

Deprecated Attributes

The following attributes are available for use to display different load types on your distribution equipment schedules. However, using dynamic footers instead is recommended. See the **New Electrical Features** article in the knowledge base for more information about dynamic footers.

Connected General Lighting Load, Total (GEN_CON): The total connected general lighting load in kVA.

Calculated General Lighting Load, Total (GEN_CAL): The total calculated general lighting load in kVA.

Connected General Lighting Load, Table 220.45 Demand (GEN_TABLE_CON): The connected general lighting load in kVA for *NEC Table 220.45* demand.

Calculated General Lighting Load, Table 220.45 Demand (GEN_TABLE_CAL): The calculated general lighting load in kVA for *NEC Table 220.45* demand.

Connected General Lighting Load, Table 100% Demand (GEN_100_CON): The connected general lighting load in kVA for 100% demand.

Calculated General Lighting Load, Table 100% Demand (GEN_100_CAL): The calculated general lighting load in kVA for 100% demand.

Connected Lighting Load (LTG_CON): The connected lighting load in kVA.

Calculated Lighting Load (LTG_CAL): The calculated lighting load in kVA. This value is 125% of the connected lighting load.

Connected Largest Motor Load (LAR_CON): The connected largest motor load in kVA.

Calculated Largest Motor Load (LAR_CAL): The calculated largest motor load in kVA. This value is 125% of the connected largest motor load.

Largest Motor Load Multiplier (LAR_DIV): The load multiplier for the largest motor load as a percentage. Equal to calculated largest motor load divided by connected largest motor load.

Connected Other Motor Load (MTR_CON): The connected load of all of the other motors in kVA.

Calculated Other Motor Load (MTR_CAL): The calculated load of all of the other motors in kVA. This value is 100% of the connected load of all of the other motors.

Connected Receptacle Load (REC_CON): The connected receptacle load in kVA.

Calculated Receptacle Load (REC_CAL): The calculated receptacle load in kVA. This value is 100% of the first 10

kVA of receptacle load and 50% of the rest of the receptacle load.

Connected Continuous Load (CON_CON): The connected continuous load in kVA.

Calculated Continuous Load (CON_CAL): The calculated continuous load in kVA. This value is 125% of the connected continuous load.

Connected Heating Load (HTG_CON): The connected heating load in kVA.

Calculated Heating Load (HTG_CAL): The calculated heating load in kVA. This value is 100% of the connected heating load.

Heating Load Multiplier (HTG_DIV): The load multiplier for the heating load as a percentage. Equal to calculated heating load divided by connected heating load.

Connected Cooling Load (COL_CON): The connected cooling load in kVA.

Calculated Cooling Load (COL_CAL): The calculated cooling load in kVA. This value is 100% of the connected heating load.

Cooling Load Multiplier (COL_DIV): The load multiplier for the cooling load as a percentage. Equal to calculated cooling load divided by connected heating load.

Connected Noncontinuous Load (NON_CON): The connected noncontinuous load in kVA.

Calculated Noncontinuous Load (NON_CAL): The calculated noncontinuous load in kVA. This value is 100% of the connected noncontinuous load.

Connected Kitchen Load (KIT_CON): The connected kitchen load in kVA.

Calculated Kitchen Load (KIT_CAL): The calculated kitchen load in kVA. The value is based upon *NEC Table 220.56*. The table determines the demand factor based upon the number of units of kitchen equipment.

Kitchen Load Multiplier (KIT_DIV): The demand factor for the calculated kitchen load. The value is based upon *NEC Table 220.56*. The table determines the demand factor based upon the number of units of kitchen equipment.

Connected Diverse Load (DIV_CON): The connected diverse load in kVA.

Calculated Diverse Load (DIV_CAL): The calculated diverse load in kVA.

Diverse Load Multiplier (DIV_DIV): The load multiplier for the diverse load as a percentage. Each diverse device has this value set individually. Equal to calculated diverse load divided by connected diverse load.

Connected Metered Demand Load (MD_CON): The connected metered demand load in kVA.

Calculated Metered Demand Load (MD_CAL): The calculated metered demand load in kVA.

Connected Future Demand Load (FUTURE_CON): The connected future demand load in kVA.

Calculated Future Demand Load (FUTURE_CAL): The calculated future demand load in kVA.

Future Demand Load Multiplier (FUTURE_DIV): The load multiplier for the future demand load as a percentage. Equal to calculated future demand load divided by connected future demand load.

Connected School Load (SCH_CON): The connected school load in kVA.

Calculated School Load (SCH_CAL): The calculated school load in kVA.

School Load Multiplier (SCH_DIV): The load multiplier for the school load as a percentage. Equal to calculated school load divided by connected school load.

Connected Residential Load (DGN_CON): The connected residential load in kVA.

Calculated Residential Load (DGN_CAL): The calculated residential load in kVA.

Connected Dwelling Unit General Load (DGT_CON): The connected general dwelling unit load in kVA.

Calculated Dwelling Unit General Load (DGT_CAL): The calculated general dwelling unit load in kVA.

Connected Multifamily Dwelling Unit General Lighting Load (DGL_CON): The connected general lighting load in kVA for a general multifamily dwelling unit.

Connected Dwelling Unit Small Appliance Load (DSA_CON): The connected dwelling unit small appliance load in kVA.

Calculated Dwelling Unit Small Appliance Load (DSA_CAL): The calculated dwelling unit small appliance load in kVA.

Connected Dwelling Unit Laundry Load (DLA_CON): The connected dwelling unit laundry load in kVA.

Calculated Dwelling Unit Laundry Load (DLA_CAL): The calculated dwelling unit laundry load in kVA.

Connected Dwelling Unit Other Appliance Load (DOA_CON): The connected dwelling unit other appliance load in kVA.

Connected Dwelling Unit Other Appliance Load, No Electric Co (DOA_NOKT_CON): The connected noncontinuous dwelling unit other appliance load in kVA.

Connected Dwelling Unit Appliance Load (DAP_CON): The connected dwelling unit appliance load in kVA.

Calculated Dwelling Unit Appliance Load (DAP_CAL): The calculated dwelling unit appliance load in kVA.

Dwelling Unit Appliance Load Multiplier (DAP_DIV): The load multiplier for the dwelling unit appliance load as a percentage. Equal to calculated dwelling unit appliance load divided by connected dwelling unit appliance load.

Connected Dwelling Unit Electric Dryer Load (DED_CON): The connected dwelling unit electric dryer load in kVA.

Calculated Dwelling Unit Electric Dryer Load (DED_CAL): The calculated dwelling unit electric dryer load in kVA.

Dwelling Unit Electric Dryer Load Multiplier (DED_DIV): The load multiplier for the dwelling unit electric dryer

load as a percentage. Equal to calculated dwelling unit electric dryer load divided by connected dwelling unit electric dryer load.

Connected Dwelling Unit Electric Cooking Load (DKT_CON): The connected dwelling unit electric cooking load in kVA.

Calculated Dwelling Unit Electric Cooking Load (DKT_CAL): The calculated dwelling unit electric cooking load in kVA.

Dwelling Unit Electric Cooking Load Multiplier (DAP_DIV): The load multiplier for the dwelling unit electric cooking load as a percentage. Equal to calculated dwelling unit electric cooking load divided by connected dwelling unit electric cooking load.

Connected Dwelling Unit Heating Load (DEH_CON): The connected dwelling unit heating load in kVA.

Calculated Dwelling Unit Heating Load (DEH_CAL): The calculated dwelling unit heating load in kVA.

Dwelling Unit Heating Load Multiplier (DEH_DIV): The load multiplier for the dwelling unit heating load as a percentage. Equal to calculated dwelling unit heating load divided by connected dwelling unit heating load.

Connected Dwelling Unit Heat Pump Load (DHP_CON): The connected dwelling unit heat pump load in kVA.

Calculated Dwelling Unit Heat Pump Load (DHP_CAL): The calculated dwelling unit heat pump load in kVA.

Dwelling Unit Heat Pump Load Multiplier (DHP_DIV): The load multiplier for the dwelling unit heat pump load as a percentage. Equal to calculated dwelling unit heat pump load divided by connected dwelling unit heat pump load.

Connected Dwelling Unit Cooling Load (DCL_CON): The connected dwelling unit cooling load in kVA.

Calculated Dwelling Unit Cooling Load (DCL_CAL): The calculated dwelling unit cooling load in kVA.

Dwelling Unit Cooling Load Multiplier (DCL_DIV): The load multiplier for the dwelling unit cooling load as a percentage. Equal to calculated dwelling unit cooling load divided by connected dwelling unit cooling load.

Dwelling Unit Heating and AC Load Maximum--NEC 220.82 (C) (DHM_CON): The dwelling unit heating and AC load maximum, per *NEC 220.82 (C)*.

Dwelling Unit Heating and AC Load Maximum Basis (DHM_BASIS): The dwelling unit heating and AC load maximum basis.

Connected Multifamily Dwelling Unit Load (DWL_CON): The connected multifamily dwelling unit load in kVA.

Calculated Multifamily Dwelling Unit Load (DWL_CAL): The calculated multifamily dwelling unit load in kVA.

Multifamily Dwelling Unit Load Multiplier (DWL_DIV): The load multiplier for the multifamily dwelling unit load as a percentage. Equal to calculated multifamily dwelling unit load divided by connected multifamily dwelling unit load.

Connected Multifamily Dwelling House Load (DHS_CON): The connected multifamily dwelling house load in kVA.

Calculated Multifamily Dwelling House Load (DHS_CAL): The calculated multifamily dwelling house load in kVA.

Dwelling Units (DWELLING_UNITS): The number of dwelling units.

Total Area (TOTAL_AREA): The total area in square feet.

Total 220.45 Demand Area (TABLE_AREA): The total area for *NEC Table 220.45* demand in square feet.

Total 100% Demand Area (100_AREA): The total area for 100% demand in square feet.

0.25 VA / sf Area, 220.45 Demand (AREA_025): The 0.25 VA/sf area for *NEC Table 220.45* demand.

0.25 VA / sf Area, 100% Demand (AREA_025_100): The 0.25 VA/sf area for 100% demand.

0.5 VA / sf Area, 220.45 Demand (AREA_05): The 0.5 VA/sf area for *NEC Table 220.45* demand.

0.5 VA / sf Area, 100% Demand (AREA_5_100): The 0.5 VA/sf area for 100% demand.

1 VA / sf Area, 220.45 Demand (AREA_1): The 1 VA/sf area for *NEC Table 220.45* demand.

1VA / sf Area, 100% Demand (AREA_1_100): The 1 VA/sf area for 100% demand.

2 VA / sf Area, 220.45 Demand (AREA_2): The 2 VA/sf area for *NEC Table 220.45* demand.

2 VA / sf Area, 100% Demand (AREA_2_100): The 2 VA/sf area for 100% demand.

3 VA / sf Area, 220.45 Demand (AREA_3): The 3 VA/sf area for *NEC Table 220.45* demand.

3 VA / sf Area, 100% Demand (AREA_3_100): The 3 VA/sf area for 100% demand.

3.5 VA / sf Area, 220.45 Demand (AREA_35): The 3.5 VA/sf area for *NEC Table 220.45* demand.

3.5 VA / sf Area, 100% Demand (AREA_35_100): The 3.5 VA/sf area for 100% demand.

Custom Load 1: Name (LT1_DESC): The name of custom load 1.

Custom Load 1: Connected Load (LT1_CON): The connected custom load 1 in kVA.

Custom Load 1: Calculated (LT1_CAL): The calculated custom load 1 in kVA.

Custom Load 1: Demand Factor (LT1_DIV): The demand factor of custom load 1. Equal to calculated custom load 1 divided by connected custom load 1.

Custom Load 2: Name (LT2_DESC): The name of custom load 2.

Custom Load 2: Connected Load (LT2_CON): The connected custom load 2 in kVA.

Custom Load 2: Calculated (LT2_CAL): The calculated custom load 2 in kVA.

Custom Load 2: Demand Factor (LT2_DIV): The demand factor of custom load 2. Equal to calculated custom load 2 divided by connected custom load 2.

Connected KVA (KVA_CON): The total connected load in kVA.

Calculated KVA (KVA_CAL): The total calculated load in kVA.

Connected Amps (CON_AMPS): The total connected load in amps. Calculated using the total connected load in kVA and assuming a balanced load. Does not take into account any phase imbalance.

Calculated Amps (TOTAL_AMPS): The total calculated load in amps. Calculated using the total calculated load in kVA and assuming a balanced load. Does not take into account any phase imbalance.

Open Block from Drawing

The `Open Block from Drawing` command is used to select a block on the drawing and open it in your CAD program. This command is useful when making changes to your customization. To make a change to a block that appears on your drawing, you can select it using this command.

To open a block from the current drawing, go to

Ribbon: `DME: Customization->Block Creation->Open Block from Drawing`

Pulldown Menu: `DM Electrical->Customization->Open Block from Drawing`

You will be prompted to select the block to be opened.

Select block to open:

The drawing that the selected block is based upon will be opened. Use standard CAD commands to make changes to the block.

After changes are made to the block, use the **Redefine Block in this Drawing** command to update the block definition on the current drawing. Otherwise, none of the changes you made will appear on your drawing.

Redefine Block in this Drawing

The `Redefine Block in this Drawing` command is used to update a block definition on a drawing after changes have been made to the drawing the block is based upon.

When a block is inserted on the drawing, the definition for that block is stored on the drawing. If the drawing that the block is based upon is later changed, the definition on the drawings where it has been inserted is not updated. You must update the block definition manually. It is possible to do this using the standard CAD **INSERT** command, but the `Redefine Block in this Drawing` command simplifies the process.

To update a block on your drawing, go to

Ribbon: `DME: Customization->Block Creation->Redefine Block in this Drawing`

Pulldown Menu: `DM Electrical->Customization->Redefine Block in this Drawing`

You will be prompted to select the block to be updated.

Select block to redefine:

The definition for the selected block will be updated to match the drawing that it is based upon.

Light Fixture Blocks

The `Light Fixture Blocks` commands allow you to customize the list of blocks available to be used for light fixtures in the project and master schedules.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Standards List`, `Copy Project List to Standards`, and `Copy Standards List to Project` commands work.

To edit the light fixture blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` ^{LTG} P `Light Fixture Blocks`

Pulldown Menu: `DM Electrical->Customization->Light Fixture Blocks->Edit Project List`

To edit the light fixture blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ^{LTG} M `Light Fixture Blocks`

Pulldown Menu: `DM Electrical->Customization->Light Fixture Blocks->Edit Master List`

Light Fixture Blocks Dialog Box

Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

For most light fixtures, *To Scale* should be checked and *Can Rotate* should be checked. See the **Common Block Information** section for more information about these values and others that can be specified for light fixture blocks.

Receptacle Blocks

The `Receptacle Blocks` commands allow you to customize the list of blocks available to be used for receptacles in the project and master schedules.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To edit the receptacle blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` ^{REC}`P` `Receptacle Blocks`
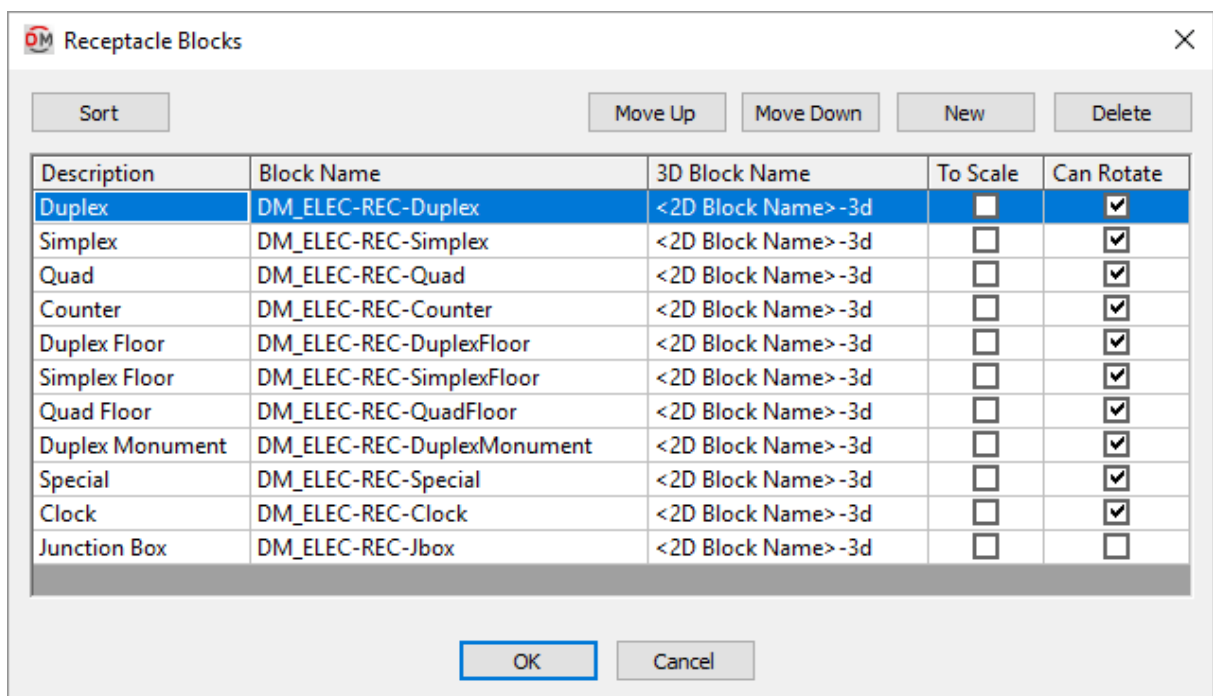
Pulldown Menu: `DM Electrical->Customization->Receptacle Blocks->Edit Project List`

To edit the receptacle blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ^{REC}`M` `Receptacle Blocks`

Pulldown Menu: `DM Electrical->Customization->Receptacle Blocks->Edit Master List`

Receptacle Blocks Dialog Box

Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

For most receptacles, *To Scale* should not be checked and *Can Rotate* should be checked. See the **Common Block Information** section for more information about these values and others that can be specified for receptacle blocks.

Equipment Blocks

The `Equipment Blocks` commands allow you to customize the list of blocks available to be used for equipment connections listed in the project and master schedules.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To edit the equipment blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` ^{EQU} `P` `Equipment Blocks`

Pulldown Menu: `DM Electrical->Customization->Equipment Blocks->Edit Project List`

To edit the equipment blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ^{EQU} `M` `Equipment Blocks`

Pulldown Menu: `DM Electrical->Customization->Equipment Blocks->Edit Master List`

Equipment Connection Blocks Dialog Box

Equipment Connection Blocks dialog

Description	2D Block Name	3D Block Name	Disconnect Block Name	3D Disconnect Block Name	NEMA	To Scale	Can Rotate	Disconnect Can Rotate
Hardwired Connection	dm_elec-equ-Connection	<2D Block Name>-3d		<2D Block Name>-3d		☐	☐	☐
Non-Fused	dm_elec-equ-Connection	<2D Block Name>-3d	dm_elec-disc-NonFused	<2D Block Name>-3d		☐	☐	☑
Fused	dm_elec-equ-Connection	<2D Block Name>-3d	dm_elec-disc-Fused	<2D Block Name>-3d		☐	☐	☑
Toggle Switch	dm_elec-equ-Connection	<2D Block Name>-3d	dm_elec-disc-ToggleSwitch	<2D Block Name>-3d		☐	☐	☑
Manual Motor Starter	dm_elec-equ-Motor	<2D Block Name>-3d	dm_elec-disc-ManualMotorStarter	<2D Block Name>-3d		☐	☐	☑
Motor Starter	dm_elec-equ-Motor	<2D Block Name>-3d	dm_elec-disc-MotorStarter	<2D Block Name>-3d		☐	☐	☑
Combination Motor Starter	dm_elec-equ-Motor	<2D Block Name>-3d	dm_elec-disc-CombinationMotorStarter	<2D Block Name>-3d		☐	☐	☑
Disconnect via Motor Starter	dm_elec-equ-Motor	<2D Block Name>-3d	dm_elec-disc-NonFused	<2D Block Name>-3d	VIA MS	☐	☐	☑
Disconnect via Combinatio...	dm_elec-equ-Motor	<2D Block Name>-3d	dm_elec-disc-NonFused	<2D Block Name>-3d	VIA COMB MS	☐	☐	☑
Fused Disconnect via Motor...	dm_elec-equ-Motor	<2D Block Name>-3d	dm_elec-disc-Fused	<2D Block Name>-3d	VIA MS	☐	☐	☑
Fused Disconnect via Comb...	dm_elec-equ-Motor	<2D Block Name>-3d	dm_elec-disc-Fused	<2D Block Name>-3d	VIA COMB MS	☐	☐	☑
Circuit Breaker	dm_elec-equ-Connection	<2D Block Name>-3d	dm_elec-disc-CircuitBreaker	<2D Block Name>-3d		☐	☐	☑
Duplex Receptacle	dm_elec-equ-DuplexRec	<2D Block Name>-3d		<2D Block Name>-3d		☐	☑	☐
Simplex Receptacle	dm_elec-equ-SimplexRec	<2D Block Name>-3d		<2D Block Name>-3d		☐	☑	☐
Special Receptacle	dm_elec-equ-SpecialRec	<2D Block Name>-3d		<2D Block Name>-3d		☐	☑	☐
Junction Box	dm_elec-equ-JboxConnection	<2D Block Name>-3d		<2D Block Name>-3d		☐	☐	☐

Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

For most equipment connections, *To Scale* should not be checked and *Can Rotate* should not be checked. See the **Common Block Information** section for more information about these values and others that can be specified for equipment connection blocks.

Disconnect Block Name: Each equipment connection can be composed of two blocks, one for the equipment and one for the disconnect. This field stores the name of the file containing the block for the disconnect. If this field is left blank, the equipment will be inserted without a disconnect.

3D Disconnect Block Name: The name of the file used to draw the disconnect in 3D. If **<2D Block Name>-3d** is chosen, the file name will be the *Disconnect Block Name* file name with "-3d" added to the end. For example, if the *Disconnect Block Name* is "disc", the 3D block name will be "disc-3d".

NEMA: This value is included as a note below the disconnect when the equipment is inserted on the drawing. If the equipment connection does not have a disconnect, this field is disabled. See the **Common Equipment Connection Information** section for more information about *NEMA* classifications.

Disconnect Can Rotate: Whether the disconnect can be rotated on the drawing.

Junction Box Blocks

The `Junction Box Blocks` commands allow you to customize the list of blocks available to be used for junction boxes in the project and master schedules.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To edit the junction box blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` ^{JBOX} `P` `Junction Box Blocks`

Pulldown Menu: `DM Electrical->Customization->Junction Box Blocks->Edit Project List`

To edit the junction box blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ^{JBOX} `M` `Junction Box Blocks`

Pulldown Menu: `DM Electrical->Customization->Junction Box Blocks->Edit Master List`

Junction Box Blocks Dialog Box
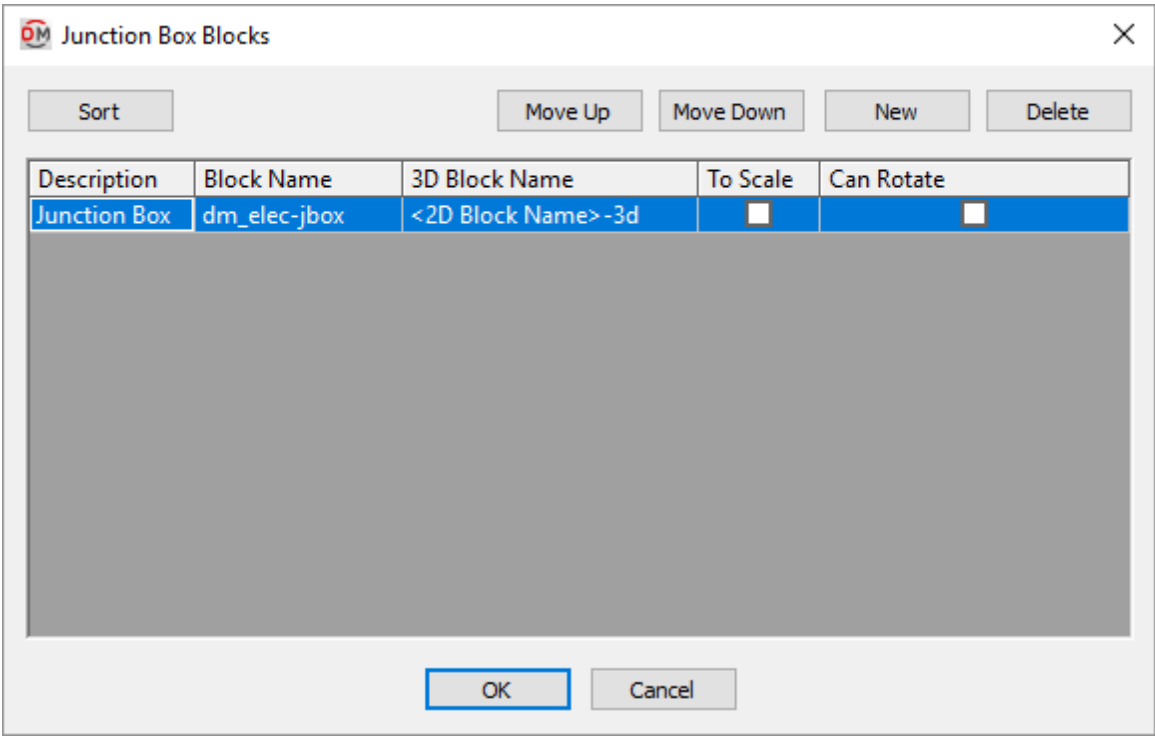
Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule,

you will be asked if you want to replace the block in the schedule with another block.

For most junction boxes, *To Scale* should not be checked and *Can Rotate* should not be checked. See the **Common Block Information** section for more information about these values and others that can be specified for junction box blocks.

Switch Blocks

The `Switch Blocks` commands allow you to customize the list of blocks available to be used for switches in the project and master schedules.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To edit the switch blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` ^{SW}P `Switch Blocks`

Pulldown Menu: `DM Electrical->Customization->Switch Blocks->Edit Project List`

To edit the switch blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ^{SW}M `Switch Blocks`

Pulldown Menu: `DM Electrical->Customization->Switch Blocks->Edit Master List`

Switch Blocks Dialog Box
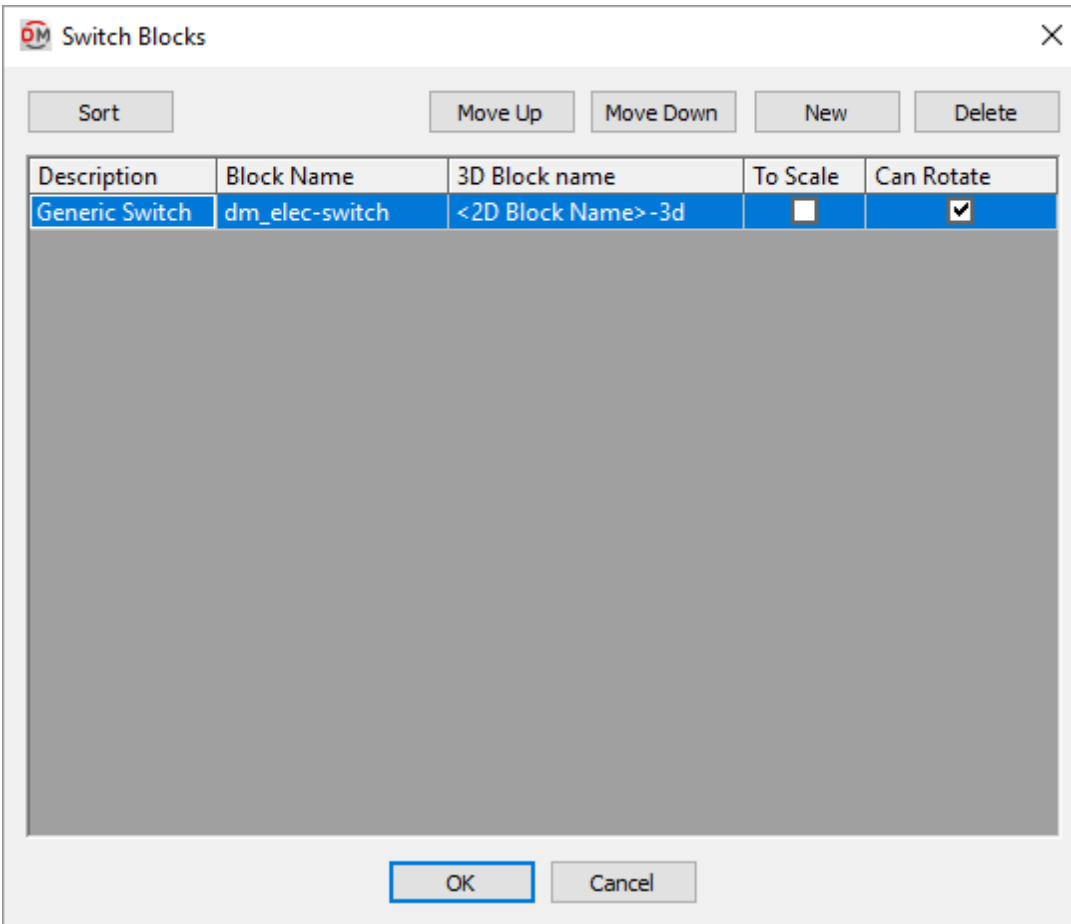
Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

For most switches, *To Scale* should not be checked and *Can Rotate* should be checked. See the **Common Block Information** section for more information about these values and others that can be specified for switch blocks.

Low Voltage Device Blocks

The `Low Voltage Device Blocks` commands allow you to customize the list of blocks available to be used for low voltage devices in the project and master schedules.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List,`

`Edit Master List,` `Copy Project List to Master,` and `Copy Master List to Project` commands
work.

To edit the low voltage device blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` ^{LVD} `P` `Low Voltage Device Blocks`
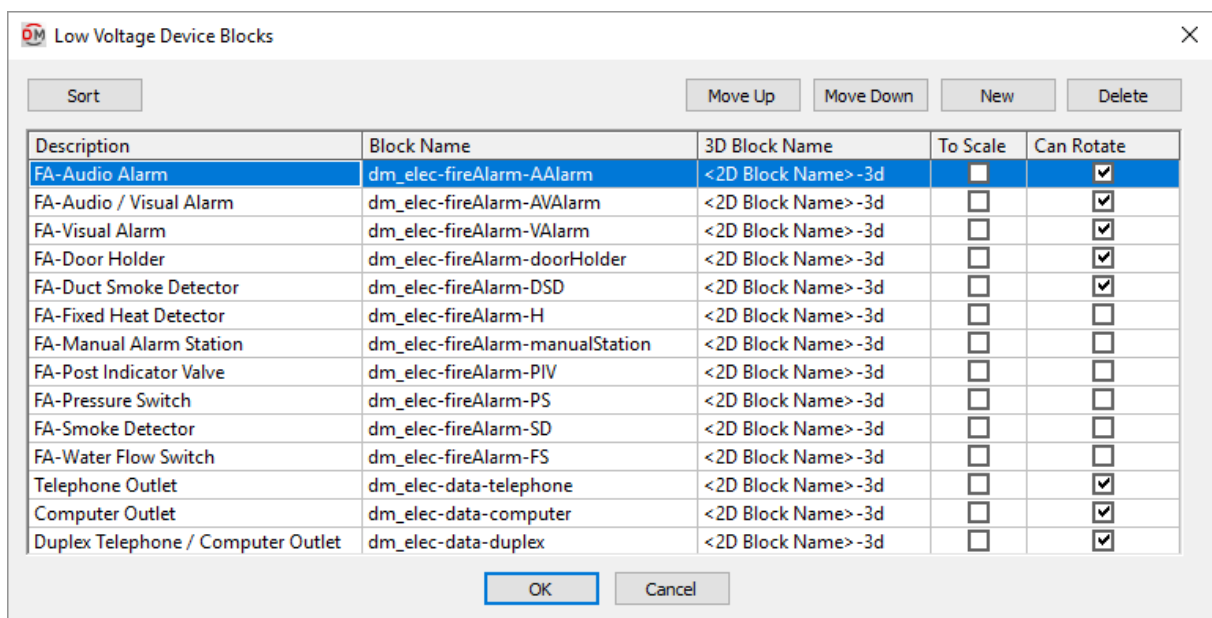
Pulldown Menu: `DM Electrical->Customization->Low Voltage Devie Blocks->Edit Project List`

To edit the low voltage device blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ^{LVD} `M` `Low Voltage Device Blocks`

Pulldown Menu: `DM Electrical->Customization->Low Voltage Device Blocks->Edit Master List`

Low Voltage Device Blocks Dialog Box

Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

For most low voltage devices, *To Scale* should not be checked and *Can Rotate* should be checked. See the **Common Block Information** section for more information about these values and others that can be specified for low voltage device blocks.

Print Blocks

The `Print Blocks` command is used to print all of the blocks in your database on the drawing. You can use this command to review all of the blocks that are defined in your database.

To insert a table of all available blocks on your drawing, go to

Ribbon: `DME: Customization->Block Creation->Print Blocks`

Pulldown Menu: `DM Electrical->Customization->Print Blocks`

Select Drawing List Dialog Box
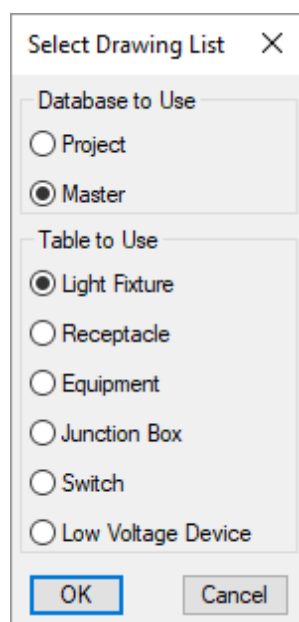
Database to Use: Whether to use the block list from the project or master database.

Table to Use: Which type of blocks to display.

Press the **OK** button. You will be prompted you to select an insertion point for the table.

Insertion point:

A table of blocks will be inserted on the drawing, similar to the light fixture block table shown below. The appearance of this table cannot be customized. It is intended for internal review purposes and not to be published in your final plans.

8"x48" Pendant or Surface	1'x4' Pendant or Surface	1'x8' Pendant or Surface	2'x2' Pendant or Surface	2'x4' Pendant or Surface	1'x2' Pendant or Surface Strip	1'x3' Pendant or Surface Strip	1'x4' Pendant or Surface Strip	1'x6' Pendant or Surface Strip	1'x8' Pendant or Surface Strip	8"x48" Recessed
DM_ELEC-LF1	DM_ELEC-LF3	DM_ELEC-LF27	DM_ELEC-LF6	DM_ELEC-LF7	DM_ELEC-LF28	DM_ELEC-LF34	DM_ELEC-LF9	DM_ELEC-LF35	DM_ELEC-LF10	DM_ELEC-LF2
1'x4' Recessed	2'x2' Recessed	2'x4' Recessed	1'x4' Recessed Nightlight	2'x2' Recessed Nightlight	2'x4' Recessed Nightlight	1'x2' Recessed Strip	1'x3' Recessed Strip	1'x4' Recessed Strip	1'x6' Recessed Strip	1'x8' Recessed Strip
DM_ELEC-LF4	DM_ELEC-LF5	DM_ELEC-LF8	DM_ELEC-LF44	DM_ELEC-LF42	DM_ELEC-LF43	DM_ELEC-LF36	DM_ELEC-LF37	DM_ELEC-LF11	DM_ELEC-LF12	DM_ELEC-LF38
6" Round Ceiling Down	12" Round Ceiling Down	18" Round Ceiling Down	24" Round Ceiling Down	6" Square Ceiling Down	12" Square Ceiling Down	Round Wall Mounted (Small)	Round Wall Mounted (Large)	Square Wall Mounted	2x Wall Mounted Fluorescent	3x Wall Mounted Fluorescent
DM_ELEC-LF32	DM_ELEC-LF23	DM_ELEC-LF24	DM_ELEC-LF25	DM_ELEC-LF33	DM_ELEC-LF29	DM_ELEC-LF41	DM_ELEC-LF15	DM_ELEC-LF14	DM_ELEC-LF13	DM_ELEC-LF22
4x Wall Mounted Fluorescent	Track or Wall Wash (Small)	Track or Wall Wash (Medium)	Track or Wall Wash (Large)	Eye Ball Aimed at Wall	2' Track Section	3' Track Section	4' Track Section	6' Track Section	8' Track Section	Ceiling Mounted Exit
DM_ELEC-LF21	DM_ELEC-LF30	DM_ELEC-LF39	DM_ELEC-LF26	DM_ELEC-LF40	DM_ELEC-LF49	DM_ELEC-LF46	DM_ELEC-LF47	DM_ELEC-LF48	DM_ELEC-LF45	DM_ELEC-LF19
Wall Mounted Exit	Emergency	Single Pole Mounted	Dual Pole Mounted	Dot	test-lf					
DM_ELEC-LF17	DM_ELEC-LF18	DM_ELEC-LF51	DM_ELEC-LF50	dm_elec-ltDot	test-lf					

Distribution Equipment Plan View Blocks

The `Distribution Equipment Plan View Blocks` commands allow you to customize the list of blocks available to be used for distribution equipment plan views in the project and master schedules.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To edit the distribution equipment plan view blocks project list, go to

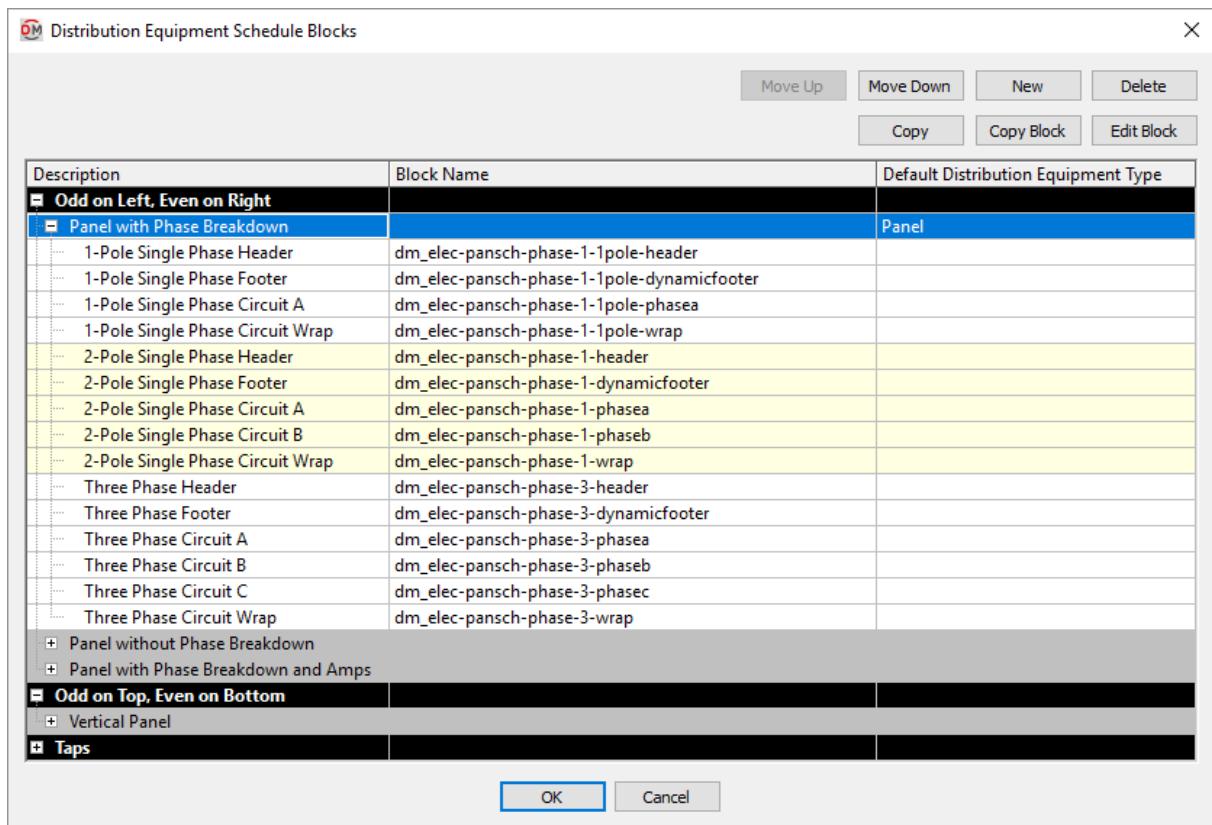**Ribbon:** `DME: Customization->Project Customization->` DEPV P `Distribution Equipment Plan View Blocks`

Pulldown Menu: `DM Electrical->Customization->Distribution Equipment Plan View Blocks->Edit Project List`

To edit the distribution equipment plan view blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` DEPV M `Distribution Equipment Plan View Blocks`

Pulldown Menu: `DM Electrical->Customization->Distribution Equipment Plan View Blocks->Edit Master List`

Distribution Equipment Plan View Blocks Dialog Box

Description	Block Name	3D Block Name	To Scale	Can Rotate	Include Access	Distribution Equipment Type	Default	Custom Size	Maximum Transformer KVA
4" x 16" Panel	dm_elec-panel16x4v2	<2D Block Name>-3d	☑	☑	☑	Panel	☑	No	
4" x 20" Panel	dm_elec-panel20x4v2	<2D Block Name>-3d	☑	☑	☑	Panel	☐	No	
5" x 20" Panel	dm_elec-panel20x5v2	<2D Block Name>-3d	☑	☑	☑	Panel	☐	No	
6" x 20" Panel	dm_elec-panel20x6v2	<2D Block Name>-3d	☑	☑	☑	Panel	☐	No	
6" x 30" Panel	dm_elec-panel30x6v2	<2D Block Name>-3d	☑	☑	☑	Panel	☐	No	
8" x 42" Panel	dm_elec-panel42x8v2	<2D Block Name>-3d	☑	☑	☑	Panel	☐	No	
Custom Panel (Elevation is Top of Panel)	dm_elec-panelCustom	<2D Block Name>-3d	☑	☑	☑	Any	☐	Yes, Elevation at Top	
Custom Panel (Elevation is Bottom of Panel)	dm_elec-panelCustom	<2D Block Name>-3d	☑	☑	☑	Panel	☐	Yes, Elevation at Bottom	
12" x 12" Transformer	dm_elec-xfmr12x12	<2D Block Name>-3d	☑	☑	☐	Transformer	☑	No	30
20" x 20" Transformer	dm_elec-xfmr20x20	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	45
30" x 20" Transformer	dm_elec-xfmr30x20	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	100
30" x 24" Transformer	dm_elec-xfmr30x24	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	150
32" x 27" Transformer	dm_elec-xfmr32x27	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	225
35" x 29" Transformer	dm_elec-xfmr35x29	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	300
48" x 36" Transformer	dm_elec-xfmr48x36	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	500
56" x 41" Transformer	dm_elec-xfmr56x41	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	750
72" x 54" Transformer	dm_elec-xfmr72x54	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	1000
84" x 60" Transformer	dm_elec-xfmr84x60	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	1500
96" x 72" Transformer	dm_elec-xfmr96x72	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	2000
102" x 78" Transformer	dm_elec-xfmr102x78	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	2500
108" x 84" Transformer	dm_elec-xfmr108x84	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	3000
120" x 96" Transformer	dm_elec-xfmr120x96	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	No	5000
Custom Transformer (Elevation is Top of Tr...	dm_elec-xfmrCustom	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	Yes, Elevation at Top	
Custom Transformer (Elevation is Bottom ...	dm_elec-xfmrCustom	<2D Block Name>-3d	☑	☑	☐	Transformer	☐	Yes, Elevation at Bottom	
Custom Rectangle (Elevation is Top of Rect...	dm_elec-oneLineRect...	<2D Block Name>-3d	☑	☑	☐	Any	☐	Yes, Elevation at Top	
Custom Rectangle (Elevation is Bottom of ...	dm_elec-oneLineRect...	<2D Block Name>-3d	☑	☑	☐	Any	☐	Yes, Elevation at Bottom	

Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

For most distribution equipment plan view blocks, *To Scale* should be checked and *Can Rotate* should be checked.See the **Common Block Information** section for more information about these values and others that can be specified for distribution equipment plan view blocks.

Include Access: Whether an access area is drawn in front of the distribution equipment plan view block. The size of the access is specified in the **Insert Plan View Block** command in the *Clearance* section of the dialog box.

Distribution Equipment Type: Which type of distribution equipment can use this block.

Default: Whether the block is used as the default for the selected *Distribution Equipment Type*. Each type listed in the *Distribution Equipment Type* column can have one block selected.

Custom Size: Whether the block can be inserted as a custom size. If this is checked, the size of the device is specified when creating or inserting the distribution equipment. The block will be scaled by the specified size.

Maximum Transformer KVA: Transformer block sizes can be selected based upon the kVA of the transformer. The block with the *Maximum Transformer KVA* value closest to, but not exceeding, the kVA of the transformer will be used.

Distribution Equipment Schedule Blocks

The `Distribution Equipment Schedule Blocks` commands allow you to customize the list of blocks

available to create distribution equipment schedules on the drawing.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To edit the distribution equipment schedule blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` DESCH `P` `Distribution Equipment Schedule Blocks`

Pulldown Menu: `DM Electrical->Customization->Distribution Equipment Schedule Blocks->Edit Project List`

To edit the distribution equipment schedule blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` DESCH `M` `Distribution Equipment Schedule Blocks`

Pulldown Menu: `DM Electrical->Customization->Distribution Equipment Schedule Blocks->Edit Master List`

Distribution Equipment Schedule Blocks Dialog Box

Move Up: Press this button to move the selected block up in the list of blocks.

468

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

Copy: Press this button to copy the selected schedule row and blocks. You will be prompted to enter a name for the copy.

Copy Block: Press this button to copy the selected block. You will be prompted to enter a name for the copy.

Edit Block: Press this button to open the drawing for the selected block.

Distribution Equipment Schedule Types

There are three different types of distribution equipment schedule available. Each type consists of a header block, a footer block, and one or more phase blocks. When a schedule is inserted, these blocks are inserted on the drawing to create the schedule.

The header block is inserted at the top of the schedule. It typically lists the equipment name and the relevant attributes of the equipment, such as the voltage and bus amps.

The footer block is inserted at the bottom of the schedule. It typically includes all of the load totals. These can include totals by phase and the connected and calculated loads by load type.

All schedules have different blocks that are used for single-phase and three-phase distribution equipment. Typically, the only difference is that the single-phase schedules do not list any C phase loads.

Odd on Left, Even on Right: This schedule type is to be used with panels. The odd circuits are listed on the left side of the schedule and the even circuits are listed on the right. This schedule type has a phase block for each phase of the schedule. Each phase block should include attributes for two circuits on it, one from the odd side of the panel and one from the even side.

Odd on Top, Even on Bottom: This schedule type is to be used with panels. The odd circuits are listed above the even circuits. For this schedule type there is a phase block for each phase of the schedule. There is also a block that is inserted between the last odd block and the first even block. Each phase block should include attributes for only one circuit. The phase block will be used both above and below the divider for the odd and even circuits.

Taps: This schedule type is to be used for all distribution equipment other than panels. This schedule type only has one phase block. One block is inserted for each tap.

Distribution Equipment Schedule Block Settings

Description: A tree that lists each block group.

The black rows separate the blocks into the three schedule types listed above.

The dark grey rows list each schedule. The value entered in this column will be displayed in the distribution equipment dialog boxes in the *Schedule Block* field.

The white rows and yellow rows list the various blocks that make up each schedule. The value displayed in this column describes what the block will be used for in the schedule.

Block Name: The block used for the corresponding *Description* in the schedule. Values can be set for the white rows and yellow rows.

Default Distribution Equipment Type: The schedule block that is used as the default for each type of distribution equipment. Values can be set for the dark grey rows. Only one schedule block can be chosen for each type of distribution equipment.

One-Line Diagram Distribution Equipment Blocks

The `One-Line Diagram Distribution Equipment Blocks` commands allow you to customize the list of blocks available to be used for distribution equipment on the one-line diagram.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To edit the one-line diagram distribution equipment blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` 1LDE `P` `One-Line Diagram Distribution Equipment Blocks`

Pulldown Menu: `DM Electrical->Customization->One-Line Diagram Distribution Equipment Blocks->Edit Project List`

To edit the one-line diagram distribution equipment blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` 1LDE `M` `One-Line Diagram Distribution Equipment Blocks`

Pulldown Menu: `DM Electrical->Customization->One-Line Diagram Distribution Equipment Blocks->Edit Master List`

One-line Diagram Distribution Equipment Blocks Dialog Box

Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

See the **Common Block Information** section for more information about the *Description* and *Block Name* values that can be specified for one-line diagram distribution equipment blocks.

One-Line Diagram Branch Circuit Device Blocks

The `One-Line Diagram Branch Circuit Device Blocks` commands allow you to customize the list of blocks available to be used for branch circuit devices on the one-line diagram.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To edit the one-line diagram branch circuit blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` 🅿 One-Line Diagram Branch Circuit Device Blocks

Pulldown Menu: `DM Electrical->Customization->One-Line Diagram Branch Circuit Device Blocks->Edit Project List`

To edit the one-line diagram branch circuit blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` 🅼 One-Line Diagram Branch Circuit Device Blocks

Pulldown Menu: `DM Electrical->Customization->One-Line Diagram Branch Circuit Device Blocks->Edit Master List`

One-line Diagram Branch Circuit Blocks Dialog Box

Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

See the **Common Block Information** section for more information about the *Description* and *Block Name* values that can be specified for one-line diagram branch circuit blocks.

One-Line Diagram Feeder Blocks

The `One-Line Diagram Feeder Blocks` commands allow you to customize the list of blocks available to be used for feeders on the one-line diagram.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List,` `Edit Master List,` `Copy Project List to Master,` and `Copy Master List to Project` commands work.

To edit the one-line diagram feeder blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` `1LFD` `P` `One-Line Diagram Feeder Blocks`

Pulldown Menu: `DM Electrical->Customization->One-Line Diagram Feeder Blocks->Edit Project List`

To edit the one-line diagram feeder blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` `1LFD` `M` `One-Line Diagram Feeder Blocks`

Pulldown Menu: `DM Electrical->Customization->One-Line Diagram Feeder Blocks->Edit Master List`

One-line Diagram Wire Blocks Dialog Box

Description	Block Name
Circuit Breaker	dm_elec-oneline-breaker
Circuit Breaker (Amps)	dm_elec-oneline-breaker-amps
Circuit Breaker (Amps and Poles)	dm_elec-oneline-breaker-poles
Circuit Breaker, Vertical (Amps)	dm_elec-oneline-breaker-vertical-amps
Circuit Breaker, Vertical (Amps and Poles)	dm_elec-oneline-breaker-vertical-amps-poles
Circuit Breaker, Single Drawout	dm_elec-oneline-breaker-drawout-single
Circuit Breaker, Double Drawout	dm_elec-oneline-breaker-drawout-double
Circuit Breaker with Dots	dm_elec-oneline-breaker-dots
Circuit Breaker with Dots (Amps)	dm_elec-oneline-breaker-dots-amps
Circuit Breaker with Dots (Amps and Poles)	dm_elec-oneline-breaker-dots-poles
Circuit Breaker with Dots, Vertical (Amps)	dm_elec-oneline-breaker-dots-vertical-amps-poles
Circuit Breaker with Dots, Vertical (Amps and Poles)	dm_elec-oneline-breaker-dots-vertical-amps-poles
Circuit Breaker with Dots, Single Drawout	dm_elec-oneline-breaker-dots-drawout-single
Circuit Breaker with Dots, Double Drawout	dm_elec-oneline-breaker-dots-drawout-double
Non-Fused Disconnect	dm_elec-oneline-nonFusedDisconnect
Fused Disconnect	dm_elec-oneline-fusedDisconnect
Fused Switch	dm_elec-oneline-fusedSwitch
Non-Fused Switch	dm_elec-oneline-nonFusedSwitch

Sort | Move Up | Move Down | New | Delete

OK | Cancel

Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

See the **Common Block Information** section for more information about the *Description* and *Block Name* values that can be specified for one-line diagram feeder blocks.

One-Line Diagram Meter Blocks

The `One-Line Diagram Meter Blocks` commands allow you to customize the list of blocks available to be used for meters on the one-line diagram.

See the **Block Creation** section for more information about creating new blocks.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To edit the one-line diagram meter blocks project list, go to

Ribbon: `DME: Customization->Project Customization->` **1LM P** `One-Line Diagram Meter Blocks`

Pulldown Menu: `DM Electrical->Customization->One-Line Diagram Meter Blocks->Edit Project List`

To edit the one-line diagram meter blocks master list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` **1LM M** `One-Line Diagram Meter Blocks`

Pulldown Menu: `DM Electrical->Customization->One-Line Diagram Meter Blocks->Edit Master List`

One-line Diagram Meter Blocks Dialog Box

Sort: Press this button to sort the list of blocks alphabetically by the *Description* column.

Move Up: Press this button to move the selected block up in the list of blocks.

Move Down: Press this button to move the selected block down in the list of blocks.

New: Press this button to create a new block.

Delete: Press this button to delete the selected block. If the block is currently used by a record in the schedule, you will be asked if you want to replace the block in the schedule with another block.

See the **Common Block Information** section for more information about the *Description* and *Block Name* values that can be specified for one-line diagram meter blocks.

One-Line Diagram Default Labels

The `One-Line Diagram Default Labels` commands allow you to customize the default settings for labels available to be inserted on the one-line diagram.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands

work.

To edit the default one-line diagram labels project list, go to

Ribbon: `DME: Customization->Project Customization->` ^{**1LDL**} `P` `One-Line Diagram Default Labels`

Pulldown Menu: `DM Electrical->Customization->One-Line Diagram Default Labels->Edit Project List`

To edit the default one-line diagram labels standards list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ^{**1LDL**} `M` `One-Line Diagram Default Labels`

Pulldown Menu: `DM Electrical->Customization->One-Line Diagram Default Labels->Edit Standards List`

Default One-Line Diagram Labels Dialog Box

Copy Labels From Drawing: Press this button to copy the current label settings from an entity on the one-line diagram.

You will be prompted to specify the entity from which the label settings will be copied.

Select one-line entity to copy:

The label settings for the selected entity will be copied.

Label: The name of the label.

Prefix: Text that is added before the value associated with the label in the database.

Suffix: Text that is added after the value associated with the label in the database.

Alignment: The alignment of the text within the text boundary.

Scale: The scale factor of the text.

Feeder IDs

The `Feeder IDs` commands allow you to customize the feeder IDs available to be used on the one-line diagram.

See the **General Customization Commands** section for more information about how the `Edit Project List` and `Edit Master List` commands work.

To edit the feeder ID project schedule, go to

Ribbon: `DME: Customization->Project Customization->` FID P `Feeder IDs`

Pulldown Menu: `DM Electrical->Customization->Feeder IDs->Edit Project List`

To edit the feeder ID master schedule, go to

Ribbon: `DME: Customization->Master & Standards Customization->` FID M `Feeder IDs`
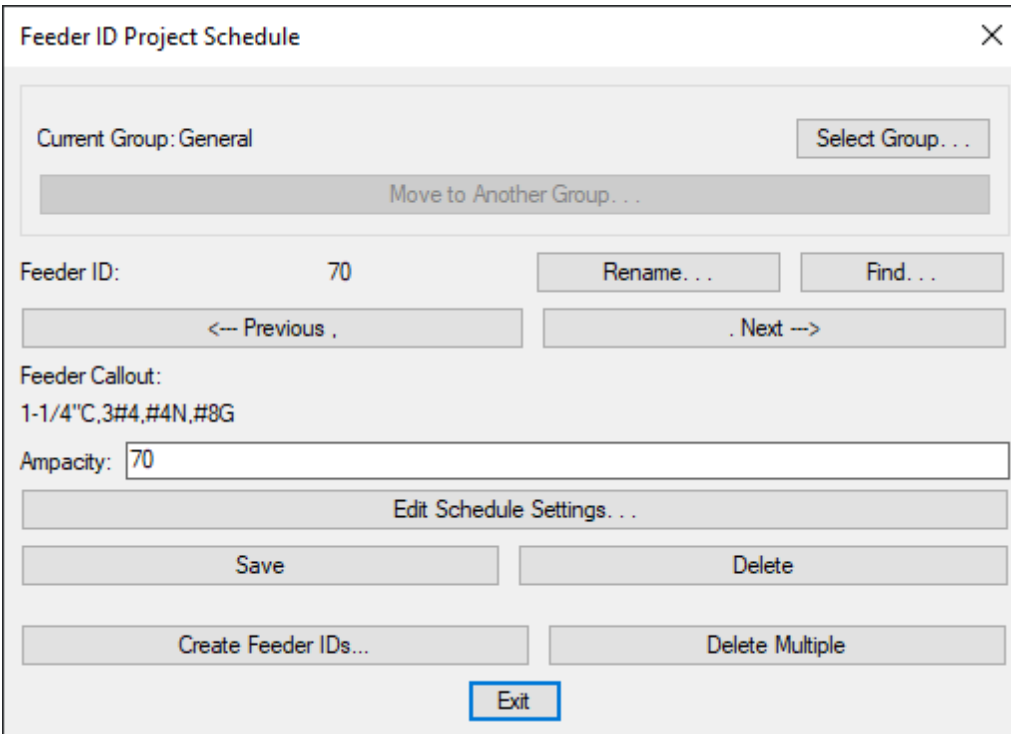
Pulldown Menu: `DM Electrical->Customization->Feeder IDs->Edit Master List`

The `Edit Project List` command opens the **Feeder ID Project Schedule** dialog box. The `Edit Master List` command opens the **Feeder ID Master Schedule** dialog box.

Training Videos

- **Feeder IDs Overview**

Feeder ID Project Schedule Dialog Box

See the **Common Schedule Dialog Box Features** section for more information about how this dialog box works. This section describes the **Rename**, **Find**, **Previous**, **Next, Save**, and **Delete** buttons.

Select Group: Press this button to open the **Schedule Groups** dialog box.

Move to Another Group: Press this button to open a list of available groups to which the feeder ID can be moved.

Feeder ID: The name of the feeder ID. This will be displayed when the feeder ID is inserted on a drawing.

Feeder Callout: The full callout of the feeder.

Ampacity: The ampacity of the feeder.

Edit Schedule Settings: Press this button to open the **Feeder ID Schedule Settings** dialog box, shown below.
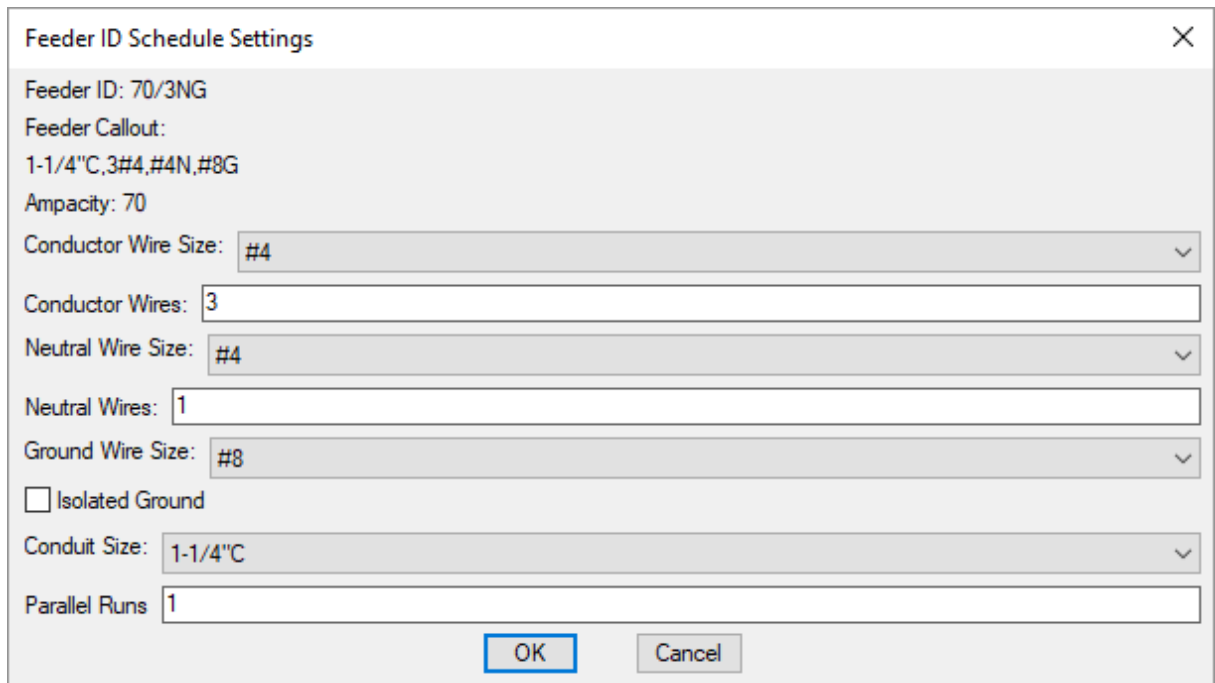
If the feeder schedule is not currently set to display any of the information set in the the **Feeder ID Schedule Settings** dialog box, a warning dialog box will appear with instructions on how to change the display settings. See the **Insert Feeder Schedule** section for more information.

Delete Multiple: Press this button to remove multiple feeder IDs from the schedule. A dialog box will appear showing the list of feeder IDs.

Select the feeder IDs to delete. You can select multiple items in the list using the SHIFT or CTRL keys. Press the **OK** button to remove them from the schedule.

Create Feeder IDs: Press this button to open the **Create Feeder ID** dialog box, shown below. Available only in the **Feeder ID Project Schedule** dialog box.

Feeder ID Schedule Settings Dialog Box

The **Feeder ID Schedule Settings** dialog box is used to change the wire size, number of wires, and other wire properties listed for the selected Feeder ID. These values are for informational purposes only and are not otherwise used.

Conductor Wire Size: The size of the conductor wire.

Conductor Wires: The number of conductor wires.

Neutral Wire Size: The size of the neutral wire.

Neutral Wires: The number of neutral wires.

Ground Wire Size: The size of the ground wire.

Isolated Ground: Whether the ground is isolated.

Conduit Size: The size of the conduit.

Parallel Runs: The number of parallel runs.

Press the **OK** button to save your changes.

Create Feeder ID Dialog Box

The **Create Feeder ID** dialog box is used to manually create one or more feeder IDs. Multiple settings can be selected to create feeder IDs for each combination of the selected settings.

Ampacity: The ampacity of the feeder.

Number of Conductors: The number of conductor wires.

Neutral Size: The size of the neutral wires.

Ground Size: The size of the ground wires.

Include IG: Whether the ground will be isolated.

Automatically create feeder IDs: Whether the feeder IDs are created automatically.

If this box is not checked, the **New Feeder Callout ID** dialog box will appear when you press the **Create Feeder IDs** button. For each feeder ID created, you can set the feeder ID, feeder ID group, whether feeder IDs should be automatically generated in the future, and whether the feeder ID should be added to the master schedule.

Create feeder ID in master schedule: Whether the feeder IDs will be inserted in the feeder ID master schedule.

Press the **Create Feeder IDs** button to create the feeder IDs.

Layers

The `Layers` command is used to customize the layers used when inserting items on the drawing.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Standards List`, `Copy Project List to Standards`, and `Copy Standards List to Project` commands work.

To customize the layers project list, go to

Ribbon: `DME: Customization->Project Customization->` `LAYER` `P` `Layers`

Pulldown Menu: `DM Electrical->Customization->Layers->Edit Project List`

To customize the layers standards list, go to

Ribbon: `DME: Customization->Master & Standards Customization->` `LAYER` `S` `Layers`

Pulldown Menu: `DM Electrical->Customization->Layers->Edit Standards List`

Layers Dialog Box

Layer Systems

Move Up: Press this button to move the selected row up in the *Layer System* list.

Move Down: Press this button to move the selected row down in the *Layer System* list.

New System: Press this button to create a new layer system.

Edit System: Press this button to open the **Layer System Options** dialog box for the selected layer system.

> **Layer System:** The name of the layer system.

> **Conduits and Cable Trays are solid:** Whether conduits and cable trays on the layer system are drawn with solid lines. If you want conduits and cable trays on the layer system to be drawn with dashed lines, uncheck this. See the **How to Make Dashed Conduits and Cable Trays** article in the knowledge base for more information.

<div align="center">483</div>

Delete System: Press this button to delete the selected layer system.

Layer System: The list of layer systems currently defined.

General: This layer system is used to store all of the layers not associated with devices, such as photometrics and schedules. Devices cannot be assigned to this layer system. It cannot be deleted or moved.

Layer Keys

Undo Layer Key Changes: Press this button to revert the layer key settings to their values when the dialog box was opened.

Layer Key: This column lists all of the different items that can be inserted using the software. The names in this column are fixed. Each layer key can be mapped to a layer that will be used when that type of item is inserted on the drawing. Multiple layer keys can use the same layer.

Layer Name: This column lists the layer that will be used when inserting items of the corresponding *Layer Key* type on the drawing. The layers you can choose from are defined in the lower section of the dialog box.

Layers

The layer list displays all of the layers that can be assigned to layer keys. The settings for each layer can be set in the list.

Layer Name: The name of the layer.

Color: The color number of the layer.

Linetype: The linetype of the layer.

Plot: Whether the layer will be plotted.

Lineweight: The lineweight of the layer. Only used when plotting with STBs.

Plot Style: The plot style of the layer. Only used when plotting with STBs.

Managing Layers

The following buttons in the **Layers** dialog box can be used to manage the layers in the dialog box.

Delete All Unused Layers: Press this button to delete all layers that are not associated with a layer key.

New Layer: Press this button to add a new layer.

Delete Layer: Press this button to delete the selected layer.

Undo Layer Changes: Press this button to revert the layer settings to their values when the dialog box was opened.

Using Excel to Modify Layer Settings

The **Import Layers from Excel** and **Export Layers to Excel** buttons are used to import and export the layer settings to Excel. Using Excel is recommended when modifying large numbers of layer settings.

Import Layers from Excel: Press this button to load the layer setting information entered in Excel into the dialog box. The layer settings will be applied to the selected layer system.

Export Layers to Excel: Press this button to export the layer settings to a CSV file that can be opened in Excel. The **Save Layer File As** dialog box will appear.
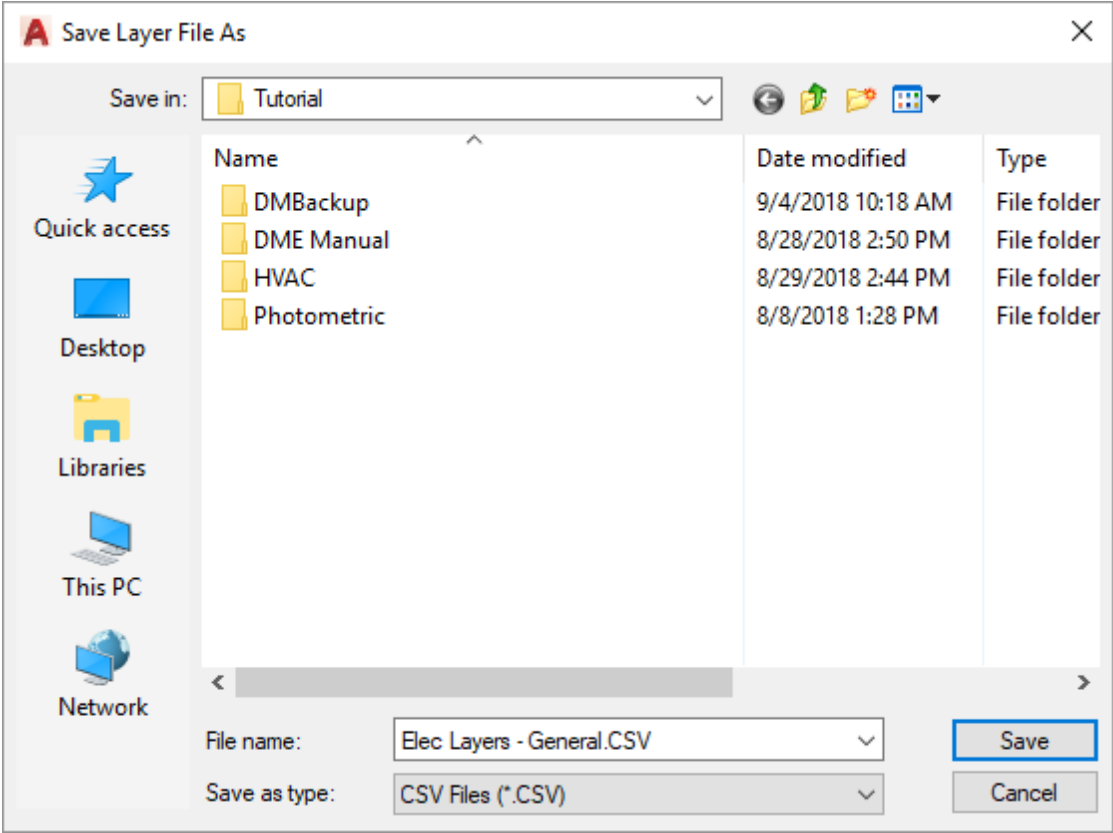
The selected layer system will be saved to the file you specify. The file will be opened for immediate editing. When modifying the file, do not change the header rows, the first column, or the last column. Restrict your changes to the *Layer Name*, *Color*, *Linetype*, *Plot*, *Lineweight*, and *Plot Style* columns.

Update Drawing Layers to Match Project List

The `Update Drawing Layers to Match Project List` command is used to update the layer definitions on the drawing. Once a layer is created on a drawing, it is not modified. Changes you make to the color, lineweight, or other attributes are not overwritten. This command is the only way to modify layers that already exist.

To update existing layers on a drawing to match new definitions in the project list, go to

Ribbon: `DME: Customization->Project Customization->` ⟳ `Update Drawing Layers to Match Project List`

Pulldown Menu: `DM Electrical->Customization->Layers->Update Drawing Layers to Match Project List`

The layer definitions on the drawing will be updated to match the definitions in the project list.

Labels and Schedules

This section describes how to customize the layout of labels, schedules, and lists on the drawing.

Common Schedule Dialog Box Information

Buttons

Move Up: Move the selected row up in the list. For schedule dialog boxes, this moves the corresponding column to the left. For labels, this moves the corresponding row up.

Move Down: Move the selected row down in the list. For schedule dialog boxes, this moves the corresponding column to the right. For labels, this moves the corresponding row down.

Grid Rows

For schedules, the first row is the schedule title. It cannot be moved. The schedule title always appears at the top of the schedule. The other rows correspond to a column in the schedule. The rows at the top of the list will be displayed on the left of the schedule.

For labels, there is no schedule title. The rows correspond to the rows of information in the label. The order of the rows in the list is used as the order of the rows on the drawing.

Grid Columns

Key: The identifier for the column, describing what value it will display. The value in this column is fixed and cannot be changed.

The **Schedule Title** key is a special key that exists for schedules. The *Label* of this key is the title of the schedule. The *Width* and *Display* values of this key are ignored. The width of the title is based upon the width of the columns in the schedule. The title is always displayed with the schedule.

Label: The name that will be displayed. For schedules, this is the column label. For labels, this is the row label.

You can define a second header for a column by separating the two headers with a | character. The first header will be merged across multiple columns with the same first header. See the **Column Subheadings in Schedules** article in the knowledge base for more information.

Width: The width in inches on the printed drawing. For schedules, each column has a separate width value that controls the value of the specific column. For labels, all of the rows share a common width.

Justification: The justification of the text values in the column. The justification of the title for the schedule and of the column headers are set using the *Schedule title justification* options and the *Schedule column label justification* option in the **Options** command.
- **Left:** The text is left-justified.
- **Center:** The text is center-justified.
- **Right:** The text is right-justified.

Display: Whether the column is shown on the schedule or the row is shown on the label. If this is not checked, the value will not be displayed.

Related Options

Schedule title justification: Sets the justification for the schedule title.

Schedule column label justification: Sets the justification for column headings.

Schedule title continued label: Sets the label added to the schedule title for continued sections.

Edit Default Schedule Settings

The `Edit Default Schedule Settings` command is used to modify the default schedule settings that are used when a project is first created. The schedule note that is displayed before the schedule and the maximum schedule height can be set using this command.

To change these settings in a project, insert the schedule using the corresponding command and make the changes in the dialog box.

To edit the schedule notes in the master database, go to

Ribbon: `DME: Customization->Master & Standards Customization->` SCH NOTE `Default Schedule Notes`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Default Schedule Settings`

Default Schedule Settings Dialog Box
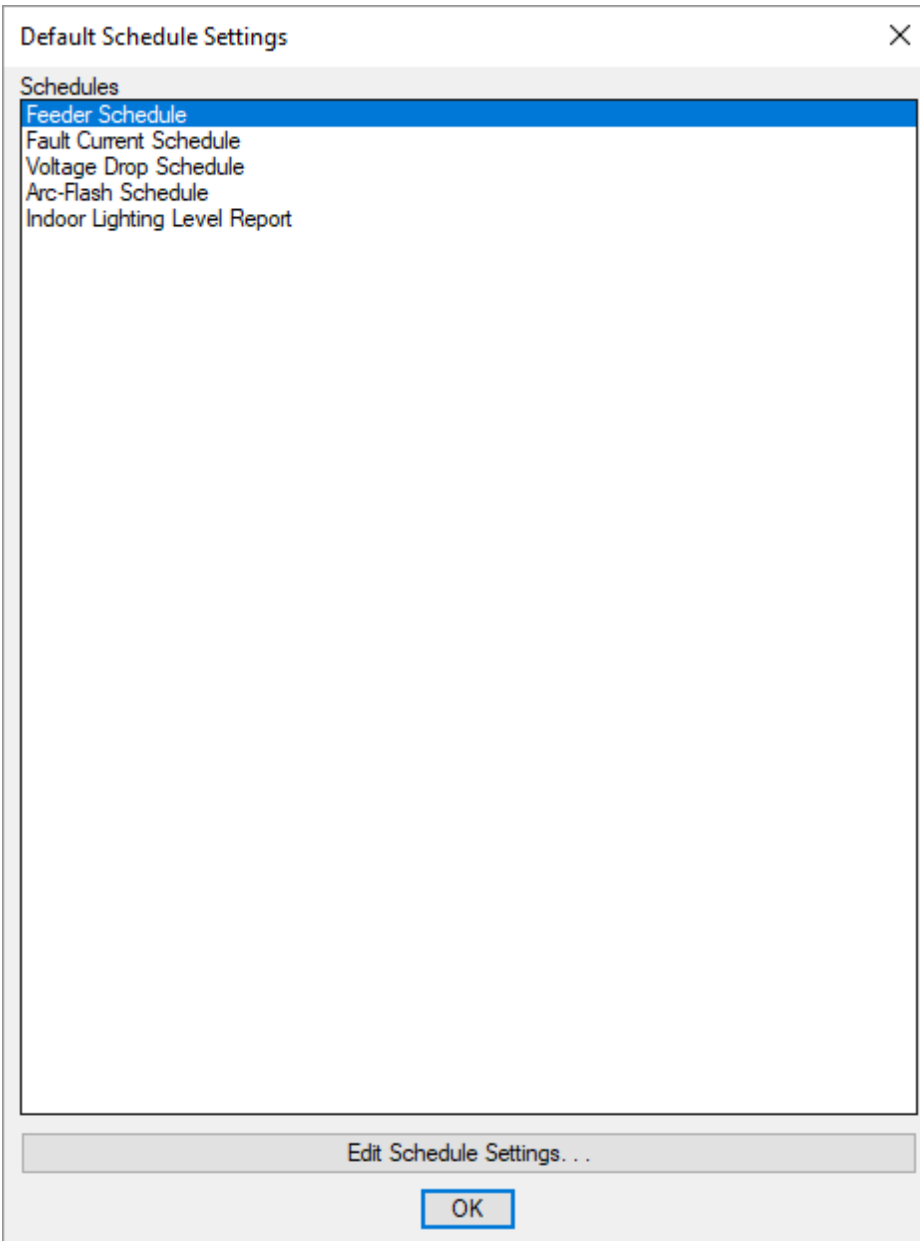
Schedules: The list of all of the device schedule types. Select the schedule for which you want to modify the note.

Edit Schedule Settings: Press this button to modify the settings for the selected schedule. The **Edit Default Schedule Settings** dialog box will appear.
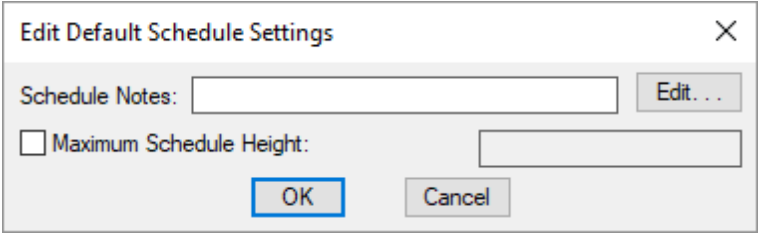
Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height is inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Edit Room Label List

The `Edit Room Label` commands are used to modify the labels that appear on rooms on the drawing. Each row that is displayed is included in the label in the order listed in the table.

To edit room label project settings, go to

Ribbon: `DME: Customization->Project Customization->` ROOM LABEL `Room Label Group`
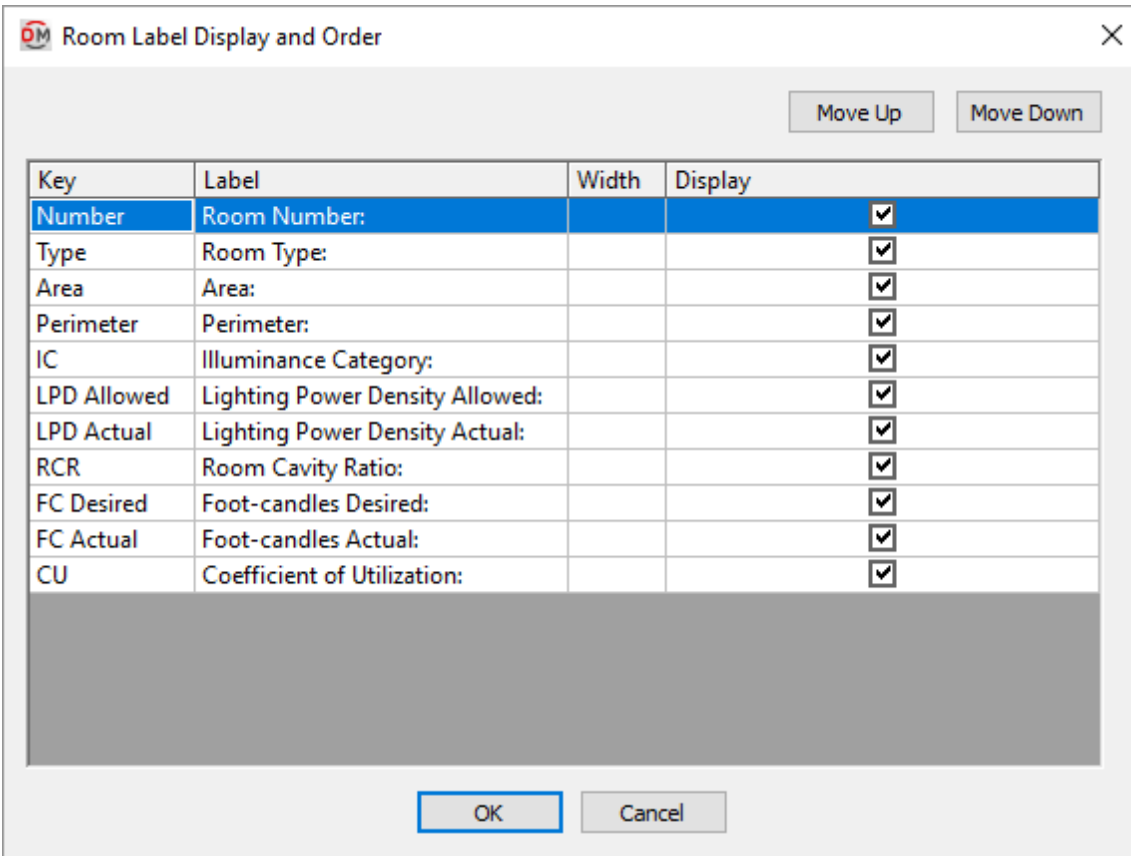
Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Room Label Project List`

To edit room label standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ROOM LABEL `Room Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Room Label Standards List`

Room Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Number*, *Label*, *Width*, and *Display* fields.

Run the **Update Room Labels** command after changing the room label settings to update room labels already inserted on the drawing.

Values in the *Key* column are listed alphabetically below.

Area: The area of the room based upon the room outline inserted on the drawing, in square feet.

CU: The coefficient of utilization of the room. This value includes the coefficient of utilization for the light fixtures and all depreciation factors.

CU = Coefficient of Utilization * Lamp Depreciation * Dirt Depreciation

FC Actual: The actual lighting level in the room, measured in foot-candles.

FC Actual = Total Lumens * Coefficient of Utilization * Lamp Depreciation * Dirt Depreciation / Area

> *Total Lumens:* The total lumens for all of the light fixtures in the room. The light fixtures in the room are based upon the light fixtures inserted on the drawing. The lumens for each light fixture is based upon the *Number of Lamps* and *Lumens / Lamp* values set in the **Light Fixture Project Schedule**.

> *Coefficient of Utilization:* If the IES file associated with the light fixture callout is well-formed, it can be used

491

to calculate the coefficient of utilization. It is based upon the IES file (see the **Light Fixture Project Schedule** section for more information), the room cavity ratio based upon the room outline inserted on the drawing, and the *Wall Cavity Reflectance*, *Effective Ceiling Cavity Reflectance*, and *Effective Floor Cavity Reflectance* (see the **Edit Rooms** and **Room Project Schedule** sections for more information).

If the IES file is not well-formed and cannot be used to calculate the coefficient of utilization, or there is no IES file, then a default value of 0.7 is used.

Lamp Depreciation: See the **Light Fixture Project Schedule** section for more information.

Dirt Depreciation: See the **Edit Rooms** and **Room Project Schedule** sections for more information.

Area: The area of the room based upon the room outline inserted on the drawing.

FC Desired: The desired lighting level for the room, measured in foot-candles. This value is specified in the room itself or in the room type. See the **Edit Rooms** section for more information.

IC: The illuminance category specified in the room project schedule. See the **Room Project Schedule** section for more information.

LPD Actual: The actual watts per square foot in the room. This value is based upon the area of the room and the light fixtures inserted in it.

LPD Allowed: The watts per square foot allowed in the room specified in the room project schedule. See the **Room Project Schedule** section for more information.

Perimeter: The perimeter of the room based upon the room outline inserted on the drawing, in feet.

RCR: The room cavity ratio of the room.

RCR = (Ceiling Height - Workspace Height) * Perimeter * 2.5 / Area

Ceiling Height: See the **Edit Rooms** and **Floor List** sections for more information.

Workspace Height: See the **Edit Rooms** and **Room Project Schedule** sections for more information.

Perimeter: The perimeter of the room based upon the room outline inserted on the drawing.

Area: The area of the room based upon the room outline inserted on the drawing.

Type: The type of the room, taken from the room project schedule. See the **Edit Rooms** and **Room Project Schedule** sections for more information.

Edit Lighting Group Label List

The `Edit Lighting Group Label` commands are used to modify the labels that appear on lighting groups on the drawing. Each row that is displayed is included in the label in the order listed in the table.

To edit lighting group label project settings, go to

Ribbon: `DME: Customization->Project Customization->` `Lighting Group Label Group`
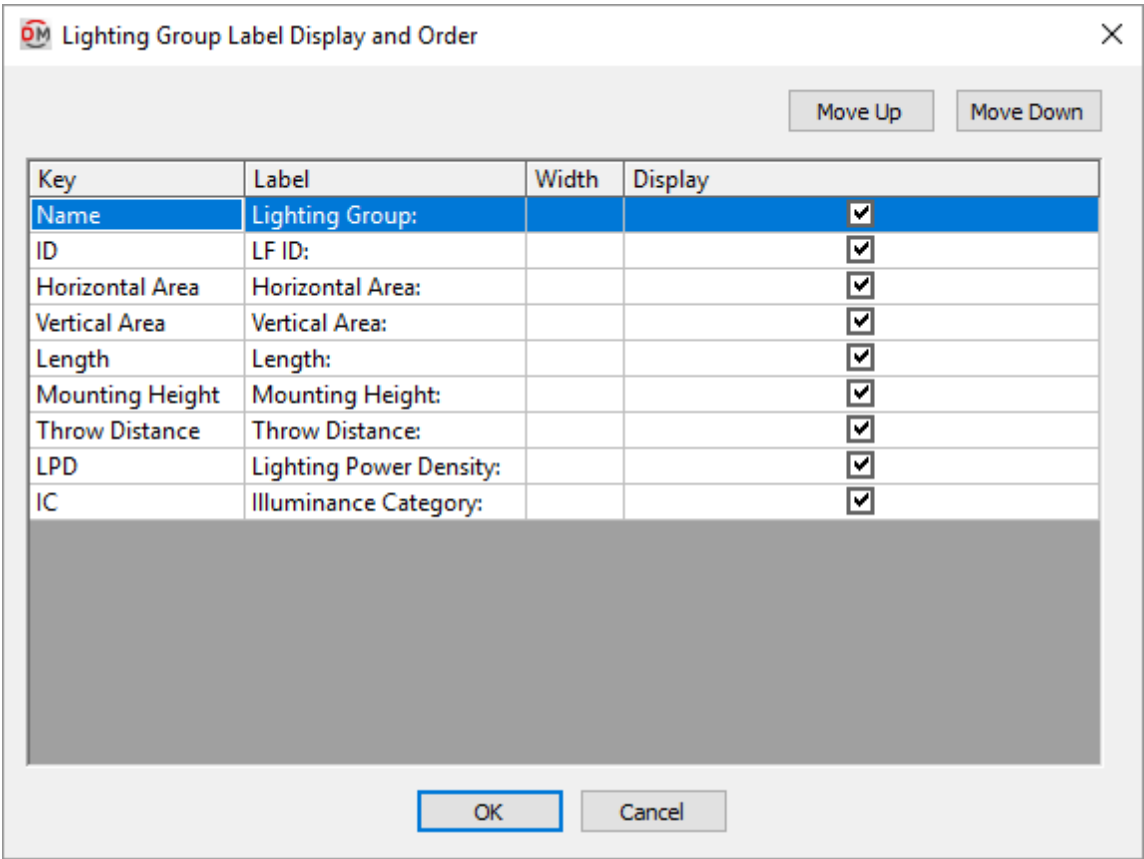
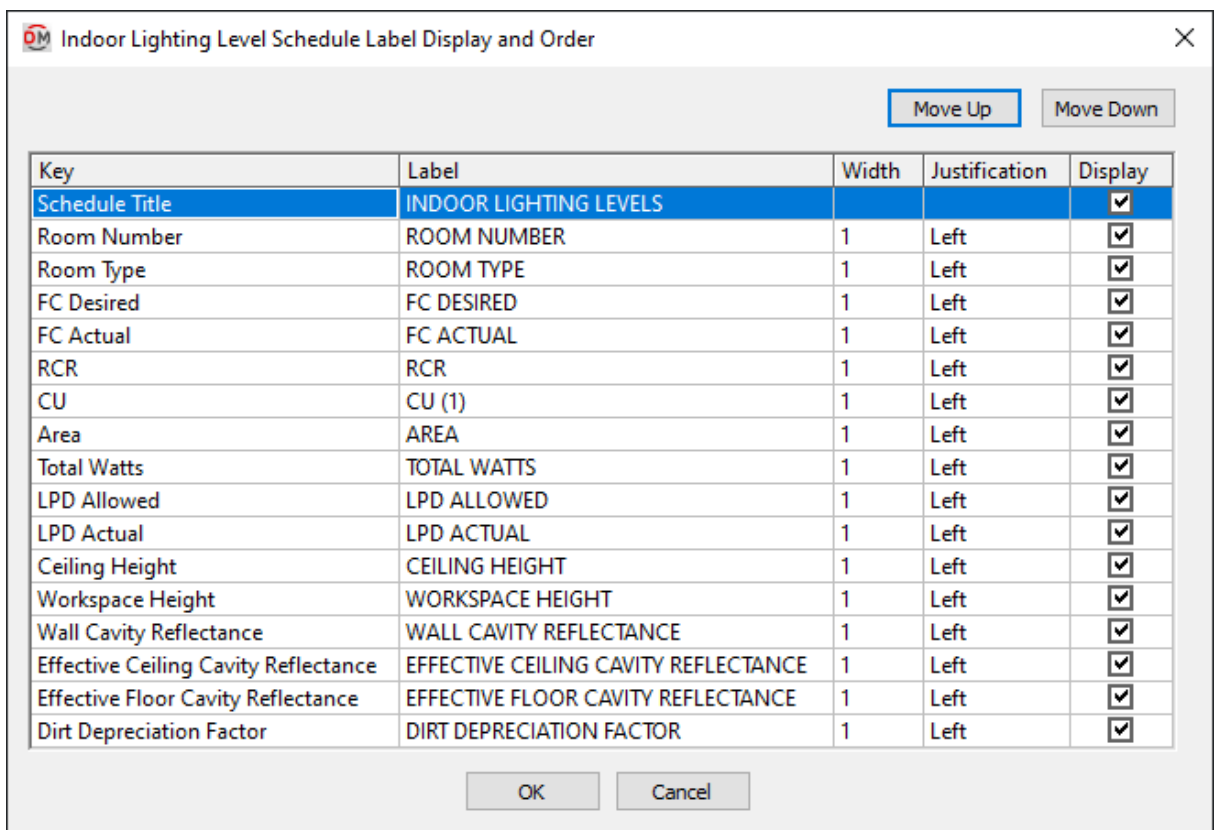**Pulldown Menu:** `DM Electrical->Customization->Labels and Schedules->Project List->Edit Lighting Group Label Project List`

To edit lighting group label standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` LG LABEL `Lighting Group Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Lighting Group Label Standards List`

Lighting Group Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, and *Display* fields.

Run the **Update Lighting Group Labels** command after changing the lighting group label settings to update lighting group labels already inserted on the drawing.

Values in the *Key* column are listed alphabetically below.

Horizontal Area: The area of the room in square feet if the group is horizontal lighting.

IC: The illuminance category specified when the group was created. See the **Common Lighting Group**

section for more information.

ID: The ID number assigned to the group.

Length: The length of the lighting group in feet. If the the group is a vertical area, the length is determined by the length of the line specifying the path of the area.

LPD: The lighting power density of the group. See the **Common Lighting Group Information** section for more information.

Mounting Height: The height at which the light fixtures in the group are mounted, in feet.

Name: The name of the lighting group specified when the group was created.

Throw Distance: The throw distance from the light fixtures to the group, in feet.

Vertical Area: The area of the room in square feet if the group is vertical lighting.

Edit Lighting Group Schedule List

To edit lighting group schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` LGS LABEL `Lighting Group Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Lighting Group Schedule Project List`

To edit lighting group schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` LGS LABEL `Lighting Group Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Lighting Group Schedule Standards List`

Lighting Group Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Fixture Type: The fixture type specified when the group was created.

Fixture Quantity: The number of the specific fixture type included in the group.

Fixture Watts: The actual wattage of the individual fixture.

Group: The name of the lighting group specified when the group was created.

ID: The ID number assigned to the group.

Total Watts: The total watts for all of the fixtures in the group.

Edit Indoor Lighting Level Schedule List

To edit indoor lighting level schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` ᴵᴸᴸ ᴸᴬᴮᴱᴸ `Indoor Lighting Level Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Indoor Lighting Level Schedule Project List`

To edit indoor lighting level schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ![ILL LABEL] `Indoor Lighting Level Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Indoor Lighting Level Schedule Standards List`

Indoor Lighting Level Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, and *Display* fields.

Values in the *Key* column are listed alphabetically below.

Area: The area of the room in square feet.

Ceiling Height: The height of the ceiling for the room in feet.

CU: The coefficient of utilization of the room. This value includes the coefficient of utilization for the light fixtures

496

and all depreciation factors.

CU = Coefficient of Utilization * Lamp Depreciation * Dirt Depreciation

Dirt Depreciation Factor: The factor by which the photometric calculations are reduced due to dirt in the room.

Effective Ceiling Cavity Reflectance: The reflective value of the ceiling in the room type. See the **Edit Rooms** and **Room Project Schedule** sections for more information.

Effective Floor Cavity Reflectance: The reflective value of the floor in the room type. See the **Edit Rooms** and **Room Project Schedule** sections for more information.

FC Actual: The actual lighting level in the room, measured in foot-candles.

FC Actual = Total Lumens * Coefficient of Utilization * Lamp Depreciation * Dirt Depreciation / Area

Total Lumens: The total lumens for all of the light fixtures in the room. The light fixtures in the room are based upon the light fixtures inserted on the drawing. The lumens for each light fixture is based upon the *Number of Lamps* and *Lumens / Lamp* values set in the **Light Fixture Project Schedule**.

Coefficient of Utilization: If the IES file associated with the light fixture callout is well-formed, it can be used to calculate the coefficient of utilization. It is based upon the IES file (see the **Light Fixture Project Schedule** section for more information), the room cavity ratio based upon the room outline inserted on the drawing, and the *Wall Cavity Reflectance*, *Effective Ceiling Cavity Reflectance*, and *Effective Floor Cavity Reflectance* (see the **Edit Rooms** and **Room Project Schedule** sections for more information).

If the IES file is not well-formed and cannot be used to calculate the coefficient of utilization, or there is no IES file, then a default value of 0.7 is used.

Lamp Depreciation: See the **Light Fixture Project Schedule** sections for more information.

Dirt Depreciation: See the **Edit Rooms** and **Room Project Schedule** sections for more information.

Area: The area of the room based upon the room outline inserted on the drawing.

FC Desired: The desired lighting level for the room, measured in foot-candles. The value is specified in the room itself or in the room type. See the **Edit Rooms** section for more information.

LPD Actual: The actual watts per square foot in the room. Based upon the total watts for all of the fixtures in the room divided by the room area.

LPD Allowed: The watts per square foot allowed in the room specified in the room project schedule. See the **Edit Rooms** and **Room Project Schedule** sections for more information.

RCR: The room cavity ratio of the room.

RCR = (Ceiling Height - Workspace Height) * Perimeter * 2.5 / Area

Ceiling Height: See the **Edit Rooms** and **Floor List** sections for more information.

Workspace Height: See the **Edit Rooms** and **Room Project Schedule** sections for more information.

Perimeter: The perimeter of the room based upon the room outline inserted on the drawing.

Area: The area of the room based upon the room outline inserted on the drawing.

Room Number: The number of the room.

Room Type: The type of room, taken from the room project schedule. See the **Edit Rooms** and **Room Project Schedule** sections for more information.

Total Watts: The total watts for all of the fixtures in the room.

Wall Cavity Reflectance: The reflective value of the walls in the room type. See the **Edit Rooms** and **Room Project Schedule** sections for more information.

Workspace Height: The elevation of the workspace above the floor level in feet.

Edit Light Fixture Schedule List

To edit light fixture schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` LTG LABEL `Light Fixture Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Light Fixture Schedule Project List`

To edit light fixture schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` LTG LABEL `Light Fixture Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Light Fixture Schedule Standards List`

Light Fixture Schedule Label Display and Order Dialog Box

Light Fixture Schedule (Group General) Label Display and Order

Key	Label	Width	Justification	Display
Schedule Title	LUMINAIRE SCHEDULE			☑
Callout	CALLOUT	1	Left	☑
Symbol	SYMBOL	1.5	Left	☑
Lamp	LAMP	2	Left	☑
Description	DESCRIPTION	3.5	Left	☑
Ballast	BALLAST	1.5	Left	☑
Mounting	MOUNTING	1.5	Left	☑
Model	MODEL	2	Left	☑
Model 1	MODEL 1	2	Left	☐
Model 2	MODEL 2	2	Left	☐
Model 3	MODEL 3	2	Left	☐
Input VA	INPUT WATTS	1	Left	☑
Volts	VOLTS	1.25	Left	☑
Note 1	NOTE 1	2	Left	☑
Note 2	NOTE 2	2	Left	☑
Note 3	NOTE 3	2	Left	☑
Quantity	QUANTITY	1	Left	☑
Volts Only	VOLTS	1	Left	☐
Total Input VA	INPUT WATTS	1	Left	☐
Default Elevation	DEFAULT ELEVATION	1	Left	☐
Lumens / Lamp	LUMENS / LAMP	1	Left	☐
Total Lumens	TOTAL LUMENS	1	Left	☐
Egress Mode Lumens / Lamp	EGRESS MODE LUMENS / LAMP	1	Left	☐
Egress Mode Total Lumens	EGRESS MODE TOTAL LUMENS	1	Left	☐
Lamp Depreciation	LAMP DEPRECIATION	1	Left	☐
IES File Name	IES FILE	2	Left	☐
IES File Name and Path	IES PATH	3	Left	☐
Arm Length	ARM LENGTH	1	Left	☐
Energy Code Watts	FIXTURE WATTS	1	Left	☐
Energy Code Fixture Type	FIXTURE TYPE	2	Left	☐
Short Voltage	VOLTS	1	Left	☐
Number of Heads	NUM OF HEADS	1	Left	☐

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Arm Length: The arm length taken from the schedule.

499

Ballast: The ballast taken from the schedule. Each of the *Ballast* values from the schedule will be listed on a separate line.

Callout: The name of the light fixture as defined in the light fixture schedule.

Default Elevation: The default elevation taken from the schedule.

Description: The description taken from the schedule.

Egress Mode Lumens / Lamp: The lumens per lamp for the fixture in egress mode. Each of the *Egress Mode Lumens / Lamp* values from the schedule will be listed on a separate line.

Egress Mode Total Lumens: The total lumens for the fixture in egress mode.

Energy Code Fixture Type: The energy code fixture type taken from the schedule.

Energy Code Watts: The energy code watts taken from the schedule.

IES File Name: The name of the IES file for the light fixture.

IES File Name and Path: The file path for the IES file for the light fixture.

Input VA: The total input watts for the fixture. Each of the *Input VA* values from the schedule will be listed on a separate line. This value includes the load on the ballast. If you only have one load in a light fixture, the *Input VA* and *Total Input VA* columns will be the same.

Lamp: The number of lamps, watts per lamp, and description taken from the schedule.

Lamp Depreciation: The lamp depreciation taken from the schedule. Each of the *Lamp Depreciation* values from the schedule will be listed on a separate line.

Lumens / Lamp: The lumens per lamp for the fixture. Each of the *Lumens / Lamp* values from the schedule will be listed on a separate line.

Model: All three models taken from the schedule. Each model will be listed on a separate line.

Model 1: The model taken from the schedule. Each model is listed in a separate column.

Model 2: The model taken from the schedule. Each model is listed in a separate column.

Model 3: The model taken from the schedule. Each model is listed in a separate column.

Mounting: The mounting taken from the schedule.

Note 1: Note taken from the schedule.

Note 2: Note taken from the schedule.

Note 3: Note taken from the schedule.

Number of Heads: The number of heads taken from the schedule.

Quantity: The number of light fixtures inserted in the project. If a light fixture type is not inserted, "NOT USED" will be displayed.

Examples: 120V 1P, 208V 2P

Short Voltage: The voltage value and poles of the light fixture. If *Volts* is set to **Universal**, all of the voltages that the light fixture type is connected to will be listed.

Examples: 120V 1P, 208V 2P

Symbol: The block used to represent the light fixture. The block is rotated and scaled so that it will fit into the schedule.

Total Input VA: The total input watts for the fixture. This value is the sum of all *Input VA* values from the schedule and the load on the ballast. If you only have one load in a light fixture, the *Input Watts* and *Total Input Watts* columns will be the same.

Total Lumens: The total lumens for the fixture.

Volts: The full voltage value of the light fixture. The volts, poles, and wires will be displayed. If *Volts* is set to **Universal**, all of the voltages that the light fixture type is connected to will be listed.

Examples: 120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the light fixture. Only the volts will be displayed. The poles and wires will not be displayed. If *Volts* is set to **Universal**, all of the voltages that the light fixture type is connected to will be listed.

Examples: 120V, 208V

Edit Receptacle Schedule List

To edit receptacle schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` **REC LABEL** `Receptacles Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Receptacle Schedule Project List`

To edit receptacle schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` **REC LABEL** `Receptacles Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Receptacle Schedule Standards List`

Receptacle Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Callout: The callout of the receptacle as defined in the receptacle project schedule.

Features: A list of the different features of the receptacle (e.g., whether the receptacle is explosion proof or waterproof, and if the GFCI has a ground or an isolated ground).

NEMA: The NEMA rating value taken from the schedule.

Note 1: Note taken from the schedule.

Note 2: Note taken from the schedule.

Note 3: Note taken from the schedule.

Quantity: The number of receptacles inserted in the project. If a receptacle type is not inserted, "NOT USED" will be displayed.

Short Voltage: The voltage value and poles of the receptacle.

Examples: 120V 1P, 208V 2P

502

Symbol: The block used to represent the receptacle. The subscript for the receptacle can be displayed in this column.See the **Insert Receptacle Project Schedule** section for more information.

Volts: The full voltage value of the receptacle. The volts, poles, and wires will be displayed.

Examples: 120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the receptacle. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 120V, 208V

Edit Equipment List

To edit the equipment list project settings, go to

Ribbon: `DME: Customization->Project Customization->` ![EQU LABEL] `Equipment List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Equipment List Project List`

To edit the equipment list standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ![EQU LABEL] `Equipment List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Equipment List Standards List`

Select Equipment Group Dialog Box
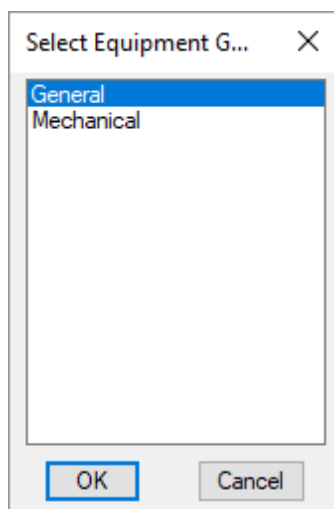
Select the equipment group to be edited and press the **OK** button.

Equipment Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Breaker: The size of the breaker and number of poles for the piece of equipment.

Examples: 20/1, 30/3

Calculated Load: The total calculated load in kVA.

If the equipment has a **Kitchen**, **Appliance, Continuous**, **Appliance, Noncontinuous**, **Electric Dryer**, **Electric Cooking**, or **Dwelling Unit Heating** load, the appropriate NEC reference for the calculated load will be displayed instead. See the **Load Information** section for more information.

Callout: The name of the piece of equipment as specified in the equipment definition.

Callout Tag: The callout tag block for the equipment. See the **Equipment Options** section for more information.

Circuit: The circuit to which the equipment is connected.

Control Wiring By: This value is taken from the equipment definition.

Custom Panel Description: The panel description for the equipment. The default value is the callout.

Demand Factor: The demand factor for each of the loads.

Description: This value is taken from the equipment definition.

Disconnect: This value is taken from the equipment definition.

Disconnect Description: This value is taken from the equipment definition.

Disconnect Fuse Size: This value is taken from the equipment definition.

Disconnect Fuse Type: This value is taken from the equipment definition.

Disconnect Installed By: This value is taken from the equipment definition.

Disconnect Mounting Height: The elevation at which the disconnect is mounted.

Disconnect Provided By: This value is taken from the equipment definition.

Disconnect Schedule Note 1: This value is taken from the equipment definition.

Disconnect Schedule Note 2: This value is taken from the equipment definition.

Disconnect Schedule Note 3: This value is taken from the equipment definition.

Equipment Mounting Height: The elevation at which the equipment is mounted.

FLA / BCSC: This value is taken from the equipment definition.

Frame Size: This value is taken from the equipment definition.

Full Volts: The full voltage designation for the equipment. It includes the voltage, the number of poles, and the number of wires.

Examples: 120V 1P 2W, 208V 2P 2W

Horsepower: If the piece of equipment has a motor specified in horsepower, the size will be displayed. If it does not have a motor specified in horsepower, this field will be left blank.

Installed By: This value is taken from the equipment definition.

KVA: The total connected load in kVA.

Load: All of the loads on the equipment. The load type and load size in kVA or horsepower are listed.

Load Type: All of the load types on the equipment.

Location: This value is taken from the equipment definition.

MCA: This value is taken from the equipment definition.

MOCP: This value is taken from the equipment definition.

NEMA: This value is taken from the equipment definition.

Poles: The number of poles on the equipment.

Provided By: This value is taken from the equipment definition.

Room: The room the equipment is inserted in.

Schedule Note 1: This value is taken from the equipment definition.

Schedule Note 2: This value is taken from the equipment definition.

Schedule Note 3: This value is taken from the equipment definition.

Separate Amps: The connected amps for each of the different load types for a piece of equipment. Each load is listed separately.

Sheet: This value is taken from the equipment definition.

Short Voltage: The voltage designation for the equipment without the wires listed. It includes the voltage and the number of poles.

Examples: 120V 1P, 208V 3P

Starter Description: This value is taken from the equipment definition.

Starter Installed By: This value is taken from the equipment definition.

Starter NEMA Enclosure: This value is taken from the equipment definition.

Starter NEMA Size: This value is taken from the equipment definition.

Starter Provided By: This value is taken from the equipment definition.

Starter Schedule Note 1: This value is taken from the equipment definition.

Starter Schedule Note 2: This value is taken from the equipment definition.

Starter Schedule Note 3: This value is taken from the equipment definition.

Starter Thermal Overload: This value is taken from the equipment definition.

Symbol: The blocks used to represent the piece of equipment, including the disconnect if the equipment has one.

Total Amps: The total calculated load in amps.

Trip Size: The size of the breaker for the circuit to which the equipment is connected.

Volts: The voltage only for the equipment. It does not include the number of poles or wires.

    Examples: 120, 208

Volts and Phase: The voltage designation for the equipment with the phase listed instead of the number of poles.

    Examples: 120V 1-PH 2W, 208V 3-PH 4W

Volts Only: The voltage value of the equipment. Only the volts will be displayed. The poles and wires will not be displayed.

    Examples: 120V, 208V

Wire Callout: The full wire callout for the equipment.

Wire Length: The length of the phase wire connected to the equipment.

Wire Size: The size of the phase wire connected to the equipment.

Edit Low Voltage Device Schedule List

To edit low voltage device schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` `LVD LABEL` `Low Voltage Device Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Low Voltage Device Schedule Project List`

To edit low voltage device schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` `LVD LABEL` `Low Voltage Device Schedule Label Group`

Select Low Voltage Device Group Dialog Box
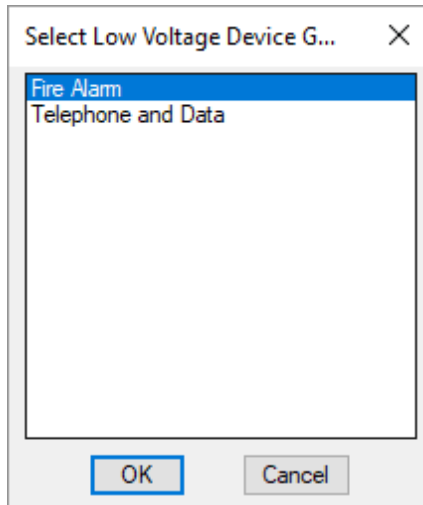
Select the low voltage device group to be edited and press the **OK** button.

<Low Voltage Device Group> Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key, Label, Width, Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Callout: The callout of the low voltage device as defined in the low voltage device project schedule.

Quantity: The number of low voltage devices inserted in the project. If a low voltage device type is not inserted, "NOT USED" will be displayed.

Schedule Note 1: Note taken from the schedule.

Schedule Note 2: Note taken from the schedule.

Schedule Note 3: Note taken from the schedule.

Symbol: The block used to represent the low voltage device.

Edit Junction Box Schedule List

To edit junction box schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` ^{JBOX LABEL} `Junction Box Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Junction Box Schedule Project List`

To edit junction box schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` JBOX LABEL `Junction Box Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Junction Box Schedule Standards List`

Junction Box Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display*, and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Callout: The callout of the junction box as defined in the junction box project schedule.

Quantity: The number of junction boxes inserted in the project. If a junction box type is not inserted, "NOT USED" will be displayed.

Schedule Note 1: Note taken from the schedule.

510

Schedule Note 2: Note taken from the schedule.

Schedule Note 3: Note taken from the schedule.

Symbol: The block used to represent the the junction box.

Edit Switch Schedule List

To edit switch schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` `Switch Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Switch Schedule Project List`

To edit switch schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` `Switch Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Switch Schedule Standards List`

Switch Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Callout: The callout of the switch as defined in the switch project schedule.

Quantity: The number of switches inserted in the project. If a switch type is not inserted, "NOT USED" will be displayed.

Schedule Note 1: Note taken from the schedule.

Schedule Note 2: Note taken from the schedule.

Schedule Note 3: Note taken from the schedule.

Symbol: The block used to represent the the switch.

Edit Feeder Schedule List

To edit feeder schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` FDR LABEL `Feeder Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Feeder Schedule Project List`

To edit feeder schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` FDR LABEL `Feeder Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Feeder Schedule Standards List`

Feeder Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Amps: The ampacity of the feeder.

Conductor Wire Count: The number of conductor wires for the feeder.

Conductor Wire Size: The conductor wire size for the feeder.

Conduit and Feeder: The size of the feeder, including the conduit size and all wire sizes.

Conduit Size: The conduit size for the feeder.

Devices: The distribution equipment fed by this size feeder.

Feeder ID: The feeder ID for this size feeder.

Ground Wire Size: The ground wire size for the feeder.

Neutral Wire Count: The number of neutral wires for the feeder.

Neutral Wire Size: The neutral wire size for the feeder.

Parallel Runs: The number of parallel runs for the feeder.

Total Wire Count: The total number of conductor and neutral wires for the feeder.

Edit Fault Schedule List

To edit the fault schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` **FAULT LABEL** `Fault Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Fault Schedule Project List`

To edit the fault schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` **FAULT LABEL** `Fault Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Fault Schedule Standards List`

Fault Current Schedule Label Display and Order Dialog Box

Fault Current Schedule Label Display and Order

Key	Label	Width	Justification	Display
Schedule Title	FAULT CURRENT SCHEDULE			☑
Device	DEVICE	1	Left	☑
Fault	FAULT	1	Left	☑
AIC Rating	AIC RATING	1	Left	☑
Line-to-Line Volts	L-L VOLTS	1	Left	☑
Line-to-Neutral Volts	L-N VOLTS	1	Left	☐
Utility: Fault	UTILITY|FAULT	1	Left	☐
Utility: X	UTILITY|X	1	Left	☐
Utility: R	UTILITY|R	1	Left	☐
Fed From: Device	FED FROM|DEVICE	1	Left	☐
Fed From: Fault	FED FROM|FAULT	1	Left	☐
Fed From: X	FED FROM|X	1	Left	☐
Fed From: R	FED FROM|R	1	Left	☐
Feeder: Size	FEEDER|SIZE	1	Left	☑
Feeder: X / 1000'	FEEDER|X / 1000'	1	Left	☐
Feeder: R / 1000'	FEEDER|R / 1000'	1	Left	☐
Feeder: Length	FEEDER|LENGTH	1	Left	☑
Feeder: X	FEEDER|X	1	Left	☐
Feeder: R	FEEDER|R	1	Left	☐
Transformer: KVA	TRANSFORMER|KVA	1	Left	☑
Transformer: Z%	TRANSFORMER|Z%	1	Left	☑
Transformer: XR Ratio	TRANSFORMER|XR RATIO	1	Left	☐
Transformer: Fault at Primary	TRANSFORMER|FAULT AT PRIMARY	1	Left	☑
Transformer: X	TRANSFORMER|X	1	Left	☐
Transformer: R	TRANSFORMER|R	1	Left	☐
Total Motor: Fault	MOTOR FAULT	1	Left	☑
Direct Motor: KVA	DIRECTLY CONNECTED MOTOR LOAD|KVA	1	Left	☐
Direct Motor: Fault	DIRECTLY CONNECTED MOTOR LOAD|FAULT	1	Left	☐
Direct Motor: X	DIRECTLY CONNECTED MOTOR LOAD|X	1	Left	☐
Direct Motor: R	DIRECTLY CONNECTED MOTOR LOAD|R	1	Left	☐

OK   Cancel

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC Rating: The AIC rating for the device. This value is set in the device definition.

Device: The distribution equipment to which the row of fault information corresponds. Devices are listed in the order they are connected, starting at the utility. Devices that are connected to other devices are indented to indicate their relationship. If the names of the devices extend beyond the edge of the column because of the indentation, increase the width of the column.

Direct Motor: Fault: The fault in amps at the device for all of the motors directly connected to the device. The fault is 4 times the load of the motors. The XR ratio for motors is assumed to be 4. The motor is assumed to be directly connected to the device. There is no reduction in the motor fault for the branch circuit length.

Direct Motor: KVA: The total load in kVA of all of the motors directly connected to the device. The fault from motors connected to other devices is reduced based upon the feeders connecting the devices.

Direct Motor: R: The resistance of all of the motors directly connected to the device.

Direct Motor: X: The reactance of all of the motors directly connected to the device.

Fault: The total fault in amps at the device, including the fault from the utility and the fault from any motors in the system.

Fed From: Device: The upstream device to which the device is connected.

Fed From: Fault: The fault in amps at the upstream device to which the device is connected.

Fed From: R: The resistance at the upstream device to which the device is connected.

Fed From: X: The reactance at the upstream device to which the device is connected.

Feeder: Length: The length of the feeder to the device.

Feeder: R: The resistance of the feeder to the device, based upon the resistance per 1,000 feet of the feeder and the actual length of the feeder.

Feeder: R / 1000': The resistance of the feeder to the device per 1,000 feet.

Feeder: Size: The wire size of the feeder connected to the device. This column lists the wire size of the phase conductors only.

Feeder: X: The reactance of the feeder to the device based upon the reactance per 1,000 feet of the feeder and the actual length of the feeder.

Feeder: X / 1000': The reactance of the feeder to the device per 1,000 feet.

Line-to-Line Volts: The line-to-line voltage for the device.

Line-to-Neutral Volts: The line-to-neutral voltage for the device.

Total Motor: Fault: The fault in amps at the device for all of the motors in the system.

Transformer: Fault at Primary: The fault in amps at the primary of the transformer. This fault is based upon the primary voltage.

Transformer: KVA: The size of the transformer in kVA. If the device is not a transformer, this field is left blank.

Transformer: R: The resistance at the primary of the transformer.

Transformer: X: The reactance at the primary of the transformer.

Transformer: XR Ratio: The transformer XR ratio. If the device is not a transformer, this field is left blank.

Transformer: Z% : The transformer impedance percentage. If the device is not a transformer, this field is left blank.

Utility: Fault: The utility fault in amps at the device. Use this column instead of the *Fault* column if you do not want to include motor contributions in your fault calculations.

Utility: X: The reactance at the device from the utility current.

Utility: R: The resistance at the device from the utility current.

Edit Voltage Drop Schedule List

To edit the voltage drop schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` UDROP LABEL `Voltage Drop Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Voltage Drop Schedule Project List`

To edit the voltage drop schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` UDROP LABEL `Voltage Drop Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Voltage Drop Schedule Standards List`

Voltage Drop Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Branch Circuit Farthest Device Distance: The distance to the farthest device on the branch circuit.

Branch Circuit Wire Size: The size of the phase conductors on the branch circuit.

Device: The distribution equipment to which the row of fault information corresponds. Devices are listed in the order they are connected, starting at the utility. Devices that are connected to other devices are indented to indicate their relationship. If the names of the devices extend beyond the edge of the column because of the indentation, increase the width of the column.

Feeder Length: The length of the feeder to the device.

Feeder Voltage Drop: The voltage drop on the feeder from the utility to the device.

Feeder Wire Size: The size of the phase conductors on the feeder to the device.

Maximum Branch Circuit Voltage Drop: The highest voltage drop on all of the branch circuits on the device.

Specific Feeder Voltage Drop: The voltage drop on the feeder from the upstream device to the device.

Total Voltage Drop: The *Feeder Voltage Drop* plus the *Maximum Branch Circuit Voltage Drop*.

Edit Arc-Flash Schedule List

To edit the arc-flash schedule project settings, go to

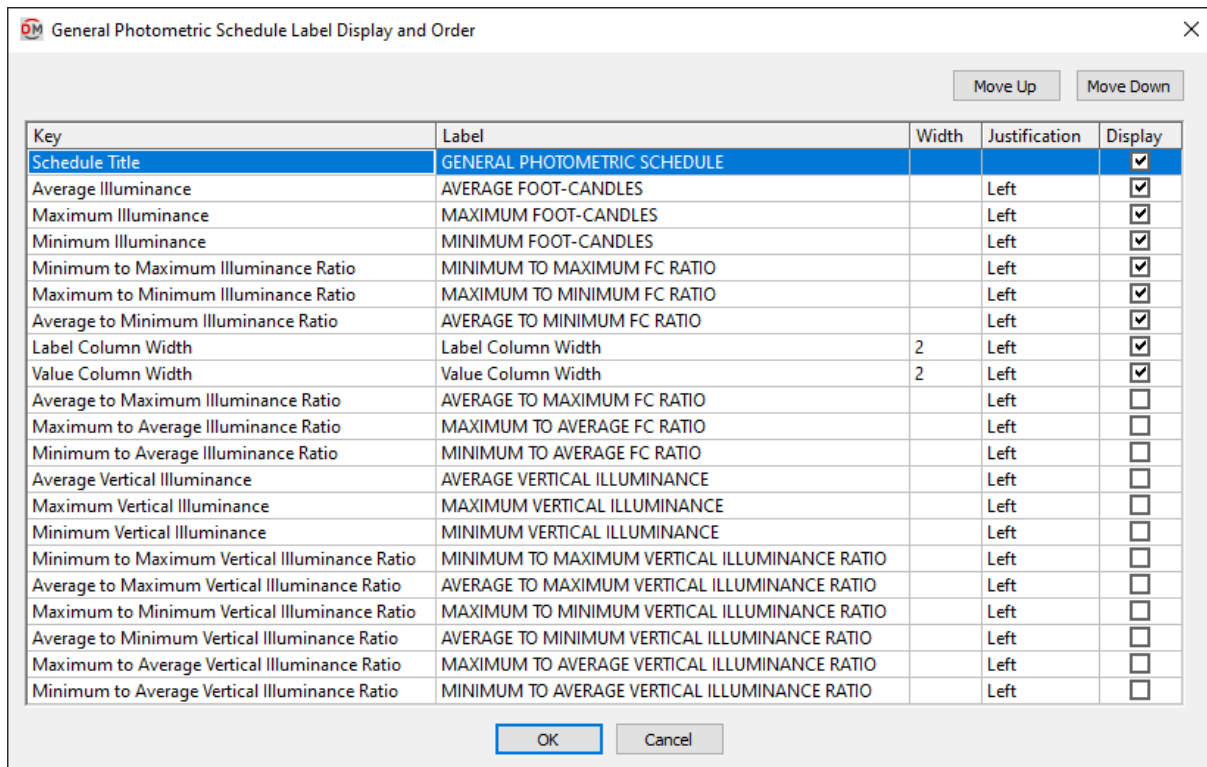**Ribbon:** `DME: Customization->Project Customization->` ᴬᴿᶜ `Arc-Flash Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Arc-Flash Schedule Project List`

To edit the arc-flash schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ᴬᴿᶜ `Arc-Flash Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Arc-Flash Schedule Standards List`

Arc-Flash Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and

Move Down buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below. Energy measurements can be displayed in joules or calories. Distance measurements can be displayed in metric or Imperial units.

Arc-Flash Boundary Distance: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Arcing Current, Maximum: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Arcing Current, Reduced: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Arcing Time, Maximum: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Arcing Time, Reduced: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Arcing Time Basis: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Bolted Fault Current: The most recently calculated fault at the device, taking into account the fault from the utility and any motors in the project.

Device: The distribution equipment to which the row of arc-flash information corresponds. Devices are listed in the order they are connected, starting at the utility. Devices that are connected to other devices are indented to indicate their relationship. If the names of the devices extend beyond the edge of the column because of the indentation, increase the width of the column.

Electrode Configuration: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Enclosure Depth: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Enclosure Height: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Enclosure Width: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Gap Between Conductors: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Incident Energy, Maximum: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Incident Energy, Reduced: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Maximum Incident Energy: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Voltage: This value is taken from the device definition.

Working Distance: This value is taken from the device definition in the **Arc-Flash Settings** dialog box.

Edit General Photometric Area Schedule List

The `Edit General Photometric Area Schedule List` commands are used to modify the labels that appear

on general photometric calculation area schedules on the drawing. Each row that is displayed is included as a row in the schedule. They do not correspond to columns like in other schedules. To set the width of the columns, use the two special values, *Label Column Width* and *Value Column Width*.

To edit the general photometric area schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` [LABEL] `General Photometric Area Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit General Photometric Area Schedule Project List`

To edit the general photometric area schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` [LABEL] `General Photometric Area Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit General Photometric Area Schedule Standards List`

General Photometric Schedule Label Display and Order Dialog Box

Key	Label	Width	Justification	Display
Schedule Title	GENERAL PHOTOMETRIC SCHEDULE			☑
Average Illuminance	AVERAGE FOOT-CANDLES		Left	☑
Maximum Illuminance	MAXIMUM FOOT-CANDLES		Left	☑
Minimum Illuminance	MINIMUM FOOT-CANDLES		Left	☑
Minimum to Maximum Illuminance Ratio	MINIMUM TO MAXIMUM FC RATIO		Left	☑
Maximum to Minimum Illuminance Ratio	MAXIMUM TO MINIMUM FC RATIO		Left	☑
Average to Minimum Illuminance Ratio	AVERAGE TO MINIMUM FC RATIO		Left	☑
Label Column Width	Label Column Width	2	Left	☑
Value Column Width	Value Column Width	2	Left	☑
Average to Maximum Illuminance Ratio	AVERAGE TO MAXIMUM FC RATIO		Left	☐
Maximum to Average Illuminance Ratio	MAXIMUM TO AVERAGE FC RATIO		Left	☐
Minimum to Average Illuminance Ratio	MINIMUM TO AVERAGE FC RATIO		Left	☐
Average Vertical Illuminance	AVERAGE VERTICAL ILLUMINANCE		Left	☐
Maximum Vertical Illuminance	MAXIMUM VERTICAL ILLUMINANCE		Left	☐
Minimum Vertical Illuminance	MINIMUM VERTICAL ILLUMINANCE		Left	☐
Minimum to Maximum Vertical Illuminance Ratio	MINIMUM TO MAXIMUM VERTICAL ILLUMINANCE RATIO		Left	☐
Average to Maximum Vertical Illuminance Ratio	AVERAGE TO MAXIMUM VERTICAL ILLUMINANCE RATIO		Left	☐
Maximum to Minimum Vertical Illuminance Ratio	MAXIMUM TO MINIMUM VERTICAL ILLUMINANCE RATIO		Left	☐
Average to Minimum Vertical Illuminance Ratio	AVERAGE TO MINIMUM VERTICAL ILLUMINANCE RATIO		Left	☐
Maximum to Average Vertical Illuminance Ratio	MAXIMUM TO AVERAGE VERTICAL ILLUMINANCE RATIO		Left	☐
Minimum to Average Vertical Illuminance Ratio	MINIMUM TO AVERAGE VERTICAL ILLUMINANCE RATIO		Left	☐

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Average Illuminance: The average illuminance value in the photometric calculation area.

Average to Maximum Illuminance Ratio: The *Average Illuminance* divided by the *Maximum Illuminance*.

Average to Maximum Vertical Illuminance Ratio: The *Average Vertical Illuminance* divided by the *Maximum Vertical Illuminance*.
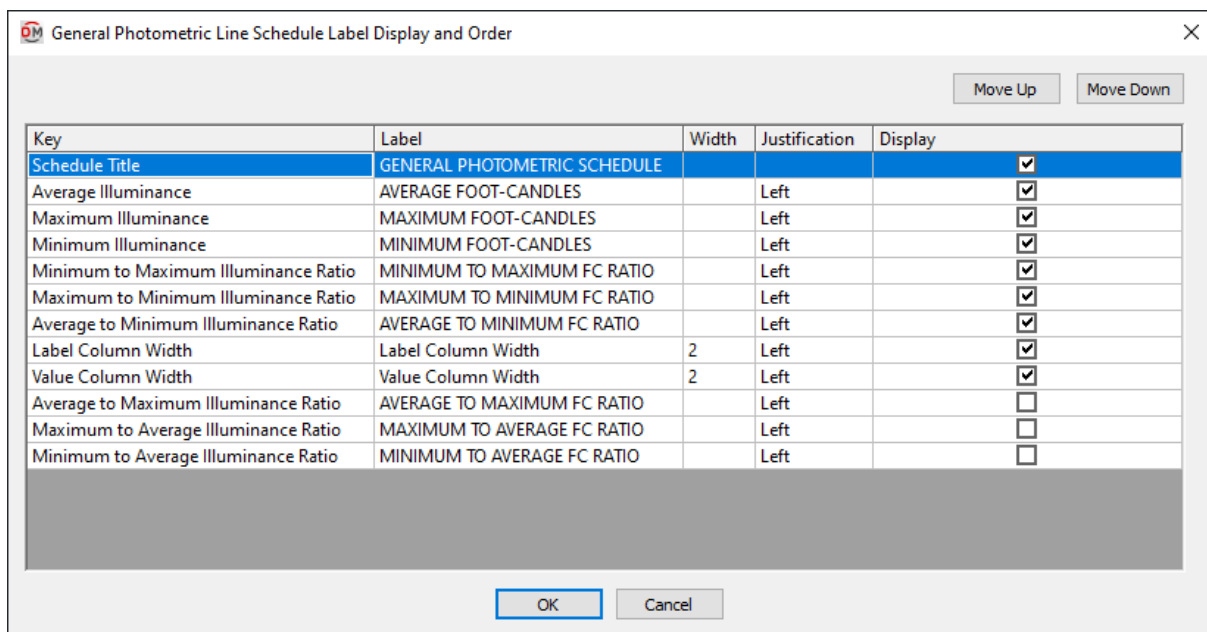
Average to Minimum Illuminance Ratio: The *Average Illuminance* divided by the *Minimum Illuminance*.

Average to Minimum Vertical Illuminance Ratio: The *Average Vertical Illuminance* divided by the *Minimum Vertical Illuminance*.

Average Vertical Illuminance: The average vertical illuminance value in the photometric calculation area.

Label Column Width: The width of the label column in the schedule.

Maximum Illuminance: The maximum illuminance value in the photometric calculation area.

Maximum to Average Illuminance Ratio: The *Maximum Illuminance* divided by the *Average Illuminance*.

Maximum to Average Vertical Illuminance Ratio: The *Maximum Vertical Illuminance* divided by the *Average Vertical Illuminance*.

Maximum to Minimum Illuminance Ratio: The *Maximum Illuminance* divided by the *Minimum Illuminance*.

Maximum to Minimum Vertical Illuminance Ratio: The *Maximum Vertical Illuminance* divided by the *Minimum Vertical Illuminance*.

Maximum Vertical Illuminance: The maximum vertical illuminance value in the photometric calculation area.

Minimum Illuminance: The minimum illuminance value in the photometric calculation area.

Minimum to Average Illuminance Ratio: The *Minimum Illuminance* divided by the *Average Illuminance*.

Minimum to Average Vertical Illuminance Ratio: The *Minimum Vertical Illuminance* divided by the *Average Vertical Illuminance*.

Minimum to Maximum Illuminance Ratio: The *Minimum Illuminance* divided by the *Maximum Illuminance*.

Minimum to Maximum Vertical Illuminance Ratio: The *Minimum Vertical Illuminance* divided by the *Maximum Vertical Illuminance*.

Minimum Vertical Illuminance: The minimum vertical illuminance value in the photometric calculation area.

Value Column Width: The width of the value column in the schedule.

Edit General Photometric Line Schedule List

The `Edit General Photometric Line Schedule List` commands are used to modify the labels that appear on general photometric calculation line schedules on the drawing. Each row that is displayed is included as a row in the schedule. They do not correspond to columns like in other schedules. To set the width of the columns, use the two special values, *Label Column Width* and *Value Column Width*.

To edit the general photometric line schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` ⬚ `General Photometric Line Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit General Photometric Line Schedule Project List`

To edit the general photometric line schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ⬚ `General Photometric Line Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit General Photometric Line Schedule Standards List`

General Photometric Line Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Average Illuminance: The average illuminance value along the photometric calculation line, including vertical illuminance levels if they are enabled.

Average to Maximum Illuminance Ratio: The *Average Illuminance* divided by the *Maximum Illuminance*.

Average to Minimum Illuminance Ratio: The *Average Illuminance* divided by the *Minimum Illuminance*.

Label Column Width: The width of the label column in the schedule.

Maximum Illuminance: The maximum illuminance value along the photometric calculation line, including vertical illuminance levels if they are enabled.

Maximum to Average Illuminance Ratio: The *Maximum Illuminance* divided by the *Average Illuminance*.

Maximum to Minimum Illuminance Ratio: The *Maximum Illuminance* divided by the *Minimum Illuminance*.

Minimum Illuminance: The minimum illuminance value along the photometric calculation line, including vertical illuminance levels if they are enabled.

Minimum to Average Illuminance Ratio: The *Minimum Illuminance* divided by the *Average Illuminance*.

Minimum to Maximum Illuminance Ratio: The *Minimum Illuminance* divided by the *Maximum Illuminance*.

Value Column Width: The width of the value column in the schedule.

Edit Egress Photometric Area Schedule List

The `Edit Egress Photometric Area Schedule List` commands are used to modify the labels that appear on egress photometric calculation area schedules on the drawing. Each row that is displayed is included as a row in the schedule. They do not correspond to columns like in other schedules. To set the width of the columns, use the two special values, *Label Column Width* and *Value Column Width*.

To edit the egress photometric area schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` 🏷️ `Egress Photometric Area Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Egress Photometric Area Schedule Project List`

To edit the egress photometric area schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` 🏷️ `Egress Photometric Area Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Egress Photometric Area Schedule Standards List`

Egress Photometric Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Average Illuminance: The average illuminance value in the photometric calculation area.

Average to Maximum Illuminance Ratio: The *Average Illuminance* divided by the *Maximum Illuminance*.

Average to Maximum Vertical Illuminance Ratio: The *Average Vertical Illuminance* divided by the *Maximum Vertical Illuminance*.

Average to Minimum Illuminance Ratio: The *Average Illuminance* divided by the *Minimum Illuminance*.

Average to Minimum Vertical Illuminance Ratio: The *Average Vertical Illuminance* divided by the *Minimum Vertical Illuminance*.

Average Vertical Illuminance: The average vertical illuminance value in the photometric calculation area.

Label Column Width: The width of the label column in the schedule.

Maximum Illuminance: The maximum illuminance value in the photometric calculation area.

Maximum to Average Illuminance Ratio: The *Maximum Illuminance* divided by the *Average Illuminance*.

Maximum to Average Vertical Illuminance Ratio: The *Maximum Vertical Illuminance* divided by the *Average Vertical Illuminance*.

Maximum to Minimum Illuminance Ratio: The *Maximum Illuminance* divided by the *Minimum Illuminance*.

Maximum to Minimum Vertical Illuminance Ratio: The *Maximum Vertical Illuminance* divided by the *Minimum Vertical Illuminance*.

Maximum Vertical Illuminance: The maximum vertical illuminance value in the photometric calculation area.

Minimum Illuminance: The minimum illuminance value in the photometric calculation area.

Minimum to Average Illuminance Ratio: The *Minimum Illuminance* divided by the *Average Illuminance*.

Minimum to Average Vertical Illuminance Ratio: The *Minimum Vertical Illuminance* divided by the *Average Vertical Illuminance*.

Minimum to Maximum Illuminance Ratio: The *Minimum Illuminance* divided by the *Maximum Illuminance*.

Minimum to Maximum Vertical Illuminance Ratio: The *Minimum Vertical Illuminance* divided by the *Maximum Vertical Illuminance*.

Minimum Vertical Illuminance: The minimum vertical illuminance value in the photometric calculation area.

Value Column Width: The width of the value column in the schedule.

Edit Egress Photometric Line Schedule List

The `Edit Egress Photometric Line Schedule List` commands are used to modify the labels that appear on egress photometric calculation line schedules on the drawing. Each row that is displayed is included as a row in the schedule. They do not correspond to columns like in other schedules. To set the width of the columns, use the two special values, *Label Column Width* and *Value Column Width*.

To edit the egress photometric line schedule project settings, go to

Ribbon: `DME: Customization->Project Customization->` LABEL `Egress Photometric Line Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Egress Photometric Line Schedule Project List`

To edit the egress photometric line schedule standards settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` LABEL `Egress Photometric Line Schedule Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master`

Egress Photometric Line Schedule Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Average Illuminance: The average illuminance value along the photometric calculation line, including vertical illuminance levels if they are enabled.

Average to Maximum Illuminance Ratio: The *Average Illuminance* divided by the *Maximum Illuminance*.

Average to Minimum Illuminance Ratio: The *Average Illuminance* divided by the *Minimum Illuminance*.

Label Column Width: The width of the label column in the schedule.

Maximum Illuminance: The maximum illuminance value along the photometric calculation line, including vertical illuminance levels if they are enabled.

Maximum to Average Illuminance Ratio: The *Maximum Illuminance* divided by the *Average Illuminance*.

Maximum to Minimum Illuminance Ratio: The *Maximum Illuminance* divided by the *Minimum Illuminance*.

Minimum Illuminance: The minimum illuminance value along the photometric calculation line, including vertical illuminance levels if they are enabled.

Minimum to Average Illuminance Ratio: The *Minimum Illuminance* divided by the *Average Illuminance*.

527

Minimum to Maximum Illuminance Ratio: The *Minimum Illuminance* divided by the *Maximum Illuminance*.

Value Column Width: The width of the value column in the schedule.

Edit Panel List

To edit panel list project settings, go to

Ribbon: `DME: Customization->Project Customization->` PAN LABEL `Panel List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Panel List`

To edit panel list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Distribution Equipment Master Groups`

Panel List Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display*, and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the panel definition.

Bus Amps: This value is taken from the panel definition.

Callout: The name of the panel.

529

Fault: The total fault in amps at the panel, including the fault from the utility and the fault from any motors in the system.

Feeder Size: This value is taken from the panel definition.

IG Bus: This value is taken from the panel definition.

List Note 1: This value is taken from the panel definition.

List Note 2: This value is taken from the panel definition.

List Note 3: This value is taken from the panel definition.

Location: This value is taken from the panel definition.

Lugs: This value is taken from the panel definition.

Main Disconnect Amps: This value is taken from the panel definition.

Main Disconnect Type: This value is taken from the panel definition.

Manufacturer: This value is taken from the panel definition.

Model: This value is taken from the panel definition.

Mounting: This value is taken from the panel definition.

NEMA: This value is taken from the panel definition.

Note 1: This value is taken from the panel definition.

Note 2: This value is taken from the panel definition.

Panel Neutral: This value is taken from the panel definition.

Poles: This value is taken from the panel definition.

Short Voltage: The voltage value and poles of the panel.

    Examples: 120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the panel.

Total Calculated KVA: The total calculated load in kVA of the panel.

Total Connected Amps: The total connected load in amps of the panel.

Total Connected KVA: The total connected load in kVA of the panel.

Upstream Breaker and Poles: The size of the breaker and number of poles for the panel feeder.

Examples: 20/1, 30/3

Upstream Circuit: The circuit to which the panel is connected on the upstream device.

Upstream Device: The upstream device to which the panel is connected.

Upstream OCP: This value is taken from the panel definition.

Utility Fault: The utility fault in amps at the panel. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the panel. The volts, poles, and wires will be displayed.

Examples: 120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the panel. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 120V, 208V

Edit Switchboard List

To edit switchboard list project settings, go to

Ribbon: DME: Customization->Project Customization-> SWBD LABEL Switchboard List Label Group

Pulldown Menu: DM Electrical->Customization->Labels and Schedules->Project List->Edit Switchboard List

To edit switchboard list master settings, go to

Ribbon: DME: Customization->Master & Standards Customization-> PAN LABEL Edit Distribution Equipment Master Groups

Pulldown Menu: DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Distribution Equipment Master Groups

Switchboard List Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the switchboard definition.

Bus Amps: This value is taken from the switchboard definition.

Callout: The name of the switchboard.

Fault: The total fault in amps at the switchboard, including the fault from the utility and the fault from any motors in the system.

Feeder Size: This value is taken from the switchboard definition.

IG Bus: This value is taken from the switchboard definition.

List Note 1: This value is taken from the switchboard definition.

List Note 2: This value is taken from the switchboard definition.

List Note 3: This value is taken from the switchboard definition.

Location: This value is taken from the switchboard definition.

Lugs: This value is taken from the switchboard definition.

Main Disconnect Amps: This value is taken from the switchboard definition.

Main Disconnect Type: This value is taken from the switchboard definition.

Manufacturer: This value is taken from the switchboard definition.

Model: This value is taken from the switchboard definition.

Mounting: This value is taken from the switchboard definition.

Neutral: This value is taken from the switchboard definition.

NEMA: This value is taken from the switchboard definition.

Note 1: This value is taken from the switchboard definition.

Note 2: This value is taken from the switchboard definition.

Number of Taps: This value is taken from the switchboard definition.

Short Voltage: The voltage value and poles of the switchboard.

Examples: 208Y/120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the switchboard.

Total Calculated KVA: The total calculated load in kVA of the switchboard.

Total Connected Amps: The total connected load in amps of the switchboard.

Total Connected KVA: The total connected load in kVA of the switchboard.

Upstream Breaker and Poles: The size of the breaker and number of poles for the switchboard feeder.

Examples: 20/1, 30/3

Upstream Circuit: The circuit to which the switchboard is connected on the upstream device.

Upstream Device: The upstream device to which the switchboard is connected.

Upstream OCP: This value is taken from the switchboard definition.

Utility Fault: The utility fault in amps at the switchboard. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the switchboard. The volts, poles, and wires will be displayed.

Examples: 208Y/120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the switchboard. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 208Y/120V, 208V

Edit Transformer List

To edit transformer list project settings, go to

Ribbon: `DME: Customization->Project Customization->` XFMR LABEL `Transformer List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Transformer List`

To edit transformer list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->`Edit Distribution Equipment Master Groups

Transformer List Label Display and Order Dialog Box

Key	Label	Width	Justification	Display
Schedule Title	TRANSFORMER SCHEDULE			☑
Callout	CALLOUT	1	Left	☑
Total Connected KVA	TOTAL CONNECTED KVA	1.5	Left	☑
Total Calculated KVA	TOTAL CALCULATED KVA	1.5	Left	☑
Total Connected Amps	TOTAL CONNECTED AMPS	1.5	Left	☑
Total Calculated Amps	TOTAL CALCULATED AMPS	1.5	Left	☑
XFMR KVA	KVA	1	Left	☑
KRated	K-RATED	1	Left	☑
Primary Volts	PRIMARY VOLTS	2	Left	☑
Secondary Volts	SECONDARY VOLTS	2	Left	☑
Number of Lugs	NUMBER OF LUGS	1	Left	☑
Note 1	NOTE 1	3	Left	☑
Note 2	NOTE 2	3	Left	☑
Location	LOCATION	1	Left	☑
AIC	SECONDARY BREAKER AIC RATING	1.5	Left	☑
Upstream OCP	UPSTREAM OCP	1.5	Left	☑
Feeder Size	FEEDER SIZE	4	Left	☑
Upstream Device	UPSTREAM DEVICE	1.5	Left	☑
Upstream Circuit	UPSTREAM CIRCUIT	1.5	Left	☑
Upstream Breaker and Poles	UPSTREAM BREAKER AND POLES	1.5	Left	☑
List Note 1	LIST NOTE 1	3	Left	☐
List Note 2	LIST NOTE 2	3	Left	☐
List Note 3	LIST NOTE 3	3	Left	☐
Impedance	IMPEDANCE	1.5	Left	☑
Primary Volts Only	PRIMARY VOLTS	1	Left	☐
Primary Short Voltage	PRIMARY VOLTS	1	Left	☐
Secondary Volts Only	SECONDARY VOLTS	1	Left	☐
Secondary Short Voltage	SECONDARY VOLTS	1	Left	☐
Fault	FAULT	1	Left	☐
Utility Fault	UTILITY FAULT	1	Left	☐
Manufacturer	MANUFACTURER	3	Left	☐
Model	MODEL	3	Left	☐
NEMA	NEMA	3	Left	☐

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the transformer definition.

Callout: The name of the transformer.

Fault: The total fault in amps at the transformer, including the fault from the utility and the fault from any motors in the system.

Feeder Size: This value is taken from the transformer definition.

Impedance: The impedance of the transformer.

KRated: This value is taken from the transformer definition.

List Note 1: This value is taken from the transformer definition.

List Note 2: This value is taken from the transformer definition.

List Note 3: This value is taken from the transformer definition.

Location: This value is taken from the transformer definition.

Manufacturer: This value is taken from the transformer definition.

Model: This value is taken from the transformer definition.

NEMA: This value is taken from the transformer definition.

Note 1: This value is taken from the transformer definition.

Note 2: This value is taken from the transformer definition.

Number of Lugs: This value is taken from the transformer definition.

Primary Short Voltage: The primary voltage value and poles of the transformer.

    Examples: 208Y/120V 1P, 208V 2P

Primary Volts: The full primary voltage value of the transformer. The volts, poles, and wires will be displayed.

    Examples: 208Y/120V 1P 2W, 208V 2P 2W

Primary Volts Only: The primary voltage value of the transformer. Only the volts will be displayed. The poles and wires will not be displayed.

    Examples: 208Y/120V, 208V

Secondary Short Voltage: The secondary voltage value and poles of the transformer.

    Examples: 208Y/120V 1P, 208V 2P

Secondary Volts: The full secondary voltage value of the transformer. The volts, poles, and wires will be displayed.

    Examples: 208Y/120V 1P 2W, 208V 2P 2W

Secondary Volts Only: The secondary voltage value of the transformer. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 208Y/120V, 208V

Total Calculated Amps: The total calculated load in amps of the transformer.

Total Calculated KVA: The total calculated load in kVA of the transformer.

Total Connected Amps: The total connected load in amps of the transformer.

Total Connected KVA: The total connected load in kVA of the transformer.

Upstream Breaker and Poles: The size of the breaker and number of poles for the transformer feeder.

Examples: 20/1, 30/3

Upstream Circuit: The circuit to which the transformer is connected on the upstream device.

Upstream Device: The upstream device to which the transformer is connected.

Upstream OCP: This value is taken from the transformer definition.

Utility Fault: The utility fault in amps at the transformer. Use this column instead of the *Fault* column if you do not want to include motor contributions.

XFMR KVA: This value is taken from the transformer definition.

Edit Wireway List

To edit wireway list project settings, go to

Ribbon: `DME: Customization->Project Customization->` WIRE LABEL `Wireway List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Wireway List`

To edit wireway list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL Edit Distribution Equipment Master Groups

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->`Edit Distribution Equipment Master Groups

Wireway List Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the wireway definition.

Callout: The name of the wireway.

Fault: The total fault in amps at the wireway, including the fault from the utility and the fault from any motors in the system.

Feeder Size: This value is taken from the wireway definition.

List Note 1: This value is taken from the wireway definition.

List Note 2: This value is taken from the wireway definition.

List Note 3: This value is taken from the wireway definition.

Location: This value is taken from the wireway definition.

Manufacturer: This value is taken from the wireway definition.

Model: This value is taken from the wireway definition.

Mounting: This value is taken from the wireway definition.

NEMA: This value is taken from the wireway definition.

Note 1: This value is taken from the wireway definition.

Note 2: This value is taken from the wireway definition.

Number of Taps: This value is taken from the wireway definition.

Short Voltage: The voltage value and poles of the wireway.

Examples: 208Y/120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the wireway.

Total Calculated KVA: The total calculated load in kVA of the wireway.

Total Connected Amps: The total connected load in amps of the wireway.

Total Connected KVA: The total connected load in kVA of the wireway.

Upstream Breaker and Poles: The size of the breaker and number of poles for the wireway feeder.

Examples: 20/1, 30/3

Upstream Circuit: The circuit to which the wireway is connected on the upstream device.

Upstream Device: The upstream device to which the wireway is connected.

Upstream OCP: This value is taken from the wireway definition.

Utility Fault: The utility fault in amps at the wireway. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the wireway. The volts, poles, and wires will be displayed.

Examples: 208Y/120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the wireway. Only the volts will be displayed. The poles and wires will not be

displayed.

Examples: 208Y/120V, 208V

Edit Bus Gutter / Duct List

To edit bus gutter and bus duct list project settings, go to

Ribbon: `DME: Customization->Project Customization-> ` **BUS LABEL** ` Bus Gutter / Bus Duct List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Bus Gutter / Duct List`

To edit bus gutter and bus duct list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization-> ` **PAN LABEL** ` Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->`Edit Distribution Equipment Master Groups

Bus Gutter List Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the bus gutter definition.

Bus Amps: This value is taken from the bus gutter definition.

Callout: The name of the bus gutter.

Fault: The total fault in amps at the bus gutter, including the fault from the utility and the fault from any motors in the system.

Feeder Size: This value is taken from the bus gutter definition.

List Note 1: This value is taken from the bus gutter definition.

List Note 2: This value is taken from the bus gutter definition.

List Note 3: This value is taken from the bus gutter definition.

Location: This value is taken from the bus gutter definition.

Manufacturer: This value is taken from the bus gutter definition.

Model: This value is taken from the bus gutter definition.

Mounting: This value is taken from the bus gutter definition.

NEMA: This value is taken from the bus gutter definition.

Note 1: This value is taken from the bus gutter definition.

Note 2: This value is taken from the bus gutter definition.

Number of Taps: This value is taken from the bus gutter definition.

Short Voltage: The voltage value and poles of the bus gutter.

   Examples: 208Y/120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the bus gutter.

Total Calculated KVA: The total calculated load in kVA of the bus gutter.

Total Connected Amps: The total connected load in amps of the bus gutter.

Total Connected KVA: The total connected load in kVA of the bus gutter.

Upstream Breaker and Poles: The size of the breaker and number of poles for the bus gutter feeder.

   Examples: 20/1, 30/3

Upstream Circuit: The circuit to which the bus gutter is connected on the upstream device.

Upstream Device: The upstream device to which the bus gutter is connected.

Upstream OCP: This value is taken from the bus gutter definition.

Utility Fault: The utility fault in amps at the bus gutter. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the bus gutter. The volts, poles, and wires will be displayed.

   Examples: 208Y/120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the bus gutter. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 208Y/120V, 208V

Edit Enclosed Breaker / Disconnect List

To edit enclosed breaker and disconnect list project settings, go to

Ribbon: `DME: Customization->Project Customization->` BKR LABEL `Enclosed Breaker / Disconnect List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Enclosed Breaker / Disconnect List`

To edit enclosed breaker and disconnect list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Distribution Equipment Master Groups`

Enclosed Breaker List Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the enclosed breaker definition.

Breaker Bus Amps: This value is taken from the enclosed breaker definition.

Callout: The name of the enclosed breaker.

Fault: The total fault in amps at the enclosed breaker, including the fault from the utility and the fault from any

motors in the system.

Feeder Size: This value is taken from the enclosed breaker definition.

Frame Size: This value is taken from the enclosed breaker definition.

List Note 1: This value is taken from the enclosed breaker definition.

List Note 2: This value is taken from the enclosed breaker definition.

List Note 3: This value is taken from the enclosed breaker definition.

Location: This value is taken from the enclosed breaker definition.

Lugs: This value is taken from the enclosed breaker definition.

Manufacturer: This value is taken from the enclosed breaker definition.

Model: This value is taken from the enclosed breaker definition.

Mounting: This value is taken from the enclosed breaker definition.

NEMA: This value is taken from the enclosed breaker definition.

Note 1: This value is taken from the enclosed breaker definition.

Note 2: This value is taken from the enclosed breaker definition.

Number of Taps: This value is taken from the enclosed breaker definition.

Short Voltage: The voltage value and poles of the enclosed breaker.

Examples: 208Y/120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the enclosed breaker.

Total Calculated KVA: The total calculated load in kVA of the enclosed breaker.

Total Connected Amps: The total connected load in amps of the enclosed breaker.

Total Connected KVA: The total connected load in kVA of the enclosed breaker.

Type: This value is taken from the enclosed breaker definition.

Upstream Breaker and Poles: The size of the breaker and number of poles for the enclosed breaker feeder.

Examples: 20/1, 30/3

Upstream Circuit: The circuit to which the enclosed breaker is connected on the upstream device.

Upstream Device: The upstream device to which the enclosed breaker is connected.

Upstream OCP: This value is taken from the enclosed breaker definition.

Utility Fault: The utility fault in amps at the enclosed breaker. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the enclosed breaker. The volts, poles, and wires will be displayed.

> Examples: 208Y/120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the enclosed breaker. Only the volts will be displayed. The poles and wires will not be displayed.

> Examples: 208Y/120V, 208V

Edit UPS / Phase Inverter List

To edit UPS and phase inverter list project settings, go to

Ribbon: `DME: Customization->Project Customization->` UPS `UPS / Phase Inverter List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit UPS / Phase Inverter List`

To edit UPS and phase inverter list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL Edit Distribution Equipment Master Groups

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->`Edit Distribution Equipment Master Groups

UPS List Label Display and Order Dialog Box

Key	Label	Width	Justification	Display
Schedule Title	UPS SCHEDULE			☑
Callout	CALLOUT	1	Left	☑
Total Connected KVA	TOTAL CONNECTED KVA	1.5	Left	☑
Total Calculated KVA	TOTAL CALCULATED KVA	1.5	Left	☑
Total Connected Amps	TOTAL CONNECTED AMPS	1.5	Left	☑
Total Calculated Amps	TOTAL CALCULATED AMPS	1.5	Left	☑
Output Volts	VOLTS	2	Left	☑
Mounting	MOUNTING	1.5	Left	☑
Taps	TAPS	1	Left	☑
IG Bus	IG BUS	1	Left	☑
Input KVA	INPUT KVA	1	Left	☑
Efficiency	EFFICIENCY	1.5	Left	☑
KVA	OUTPUT KVA	1	Left	☑
KW	OUTPUT KW	1	Left	☑
Power Factor	POWER FACTOR	1	Left	☑
Note 1	NOTE 1	3	Left	☑
Note 2	NOTE 2	3	Left	☑
Location	LOCATION	1	Left	☑
AIC	AIC RATING	1	Left	☑
Normal Upstream OCP	NORMAL UPSTREAM OCP	1	Left	☑
Bypass Upstream OCP	BYPASS UPSTREAM OCP	1	Left	☑
Normal Feeder Size	NORMAL FEEDER SIZE	4	Left	☑
Bypass Feeder Size	BYPASS FEEDER SIZE	4	Left	☑
Normal Upstream Device	NORMAL DEVICE	1.5	Left	☑
Normal Upstream Circuit	NORMAL CIRCUIT	1.5	Left	☑
Normal Upstream Breaker and Poles	NORMAL BREAKER AND POLES	1.5	Left	☑
Bypass Upstream Device	BYPASS DEVICE	1.5	Left	☑
Bypass Upstream Circuit	BYPASS CIRCUIT	1.5	Left	☑
Bypass Upstream Breaker and Poles	BYPASS BREAKER AND POLES	1.5	Left	☑
List Note 1	LIST NOTE 1	3	Left	☑
List Note 2	LIST NOTE 2	3	Left	☑
List Note 3	LIST NOTE 3	3	Left	☑
Output Volts Only	VOLTS	1	Left	☑
Output Short Voltage	VOLTS	1	Left	☑
Fault	FAULT	1	Left	☑
Utility Fault	UTILITY FAULT	1	Left	☑
Manufacturer	MANUFACTURER	3	Left	☑
Model	MODEL	3	Left	☑
NEMA	NEMA	3	Left	☑
Input Volts	INPUT VOLTS	2	Left	☑
Input Volts Only	INPUT VOLTS	1	Left	☑
Input Short Voltage	INPUT VOLTS	1	Left	☑

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the UPS definition.

Bypass Feeder Size: This value is taken from the UPS definition.

Bypass Upstream OCP: This value is taken from the UPS definition.

Bypass Upstream Breaker and Poles: The size of the breaker and number of poles for the UPS bypass feeder.

Examples: 20/1, 30/3

Bypass Upstream Circuit: The circuit to which the UPS is connected on the upstream device for bypass power.

Bypass Upstream Device: The upstream device to which the UPS is connected for bypass power.

Callout: The name of the UPS.

Efficiency: This value is taken from the UPS definition.

Fault: The total fault in amps at the UPS, including the fault from the utility and the fault from any motors in the system.

IG Bus: This value is taken from the UPS definition.

Input KVA: This value is taken from the UPS definition.

Input Short Voltage: The input voltage value and poles of the UPS.

Examples: 208Y/120V 1P, 208V 2P

Input Volts: The full input voltage value of the UPS. The volts, poles, and wires will be displayed.

Examples: 208Y/120V 1P 2W, 208V 2P 2W

Input Volts Only: The input voltage value of the UPS. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 208Y/120V, 208V

KVA: This value is taken from the UPS definition.

KW: This value is taken from the UPS definition.

List Note 1: This value is taken from the UPS definition.

List Note 2: This value is taken from the UPS definition.

List Note 3: This value is taken from the UPS definition.

Location: This value is taken from the UPS definition.

Manufacturer: This value is taken from the UPS definition.

Model: This value is taken from the UPS definition.

Mounting: This value is taken from the UPS definition.

NEMA: This value is taken from the UPS definition.

Normal Feeder Size: This value is taken from the UPS definition.

Normal Upstream Breaker and Poles: The size of the breaker and number of poles for the UPS normal feeder.

Examples: 20/1, 30/3

Normal Upstream Circuit: The circuit to which the UPS is connected on the upstream device for normal power.

Normal Upstream Device: The upstream device to which the UPS is connected normal power.

Normal Upstream OCP: This value is taken from the UPS definition.

Note 1: This value is taken from the UPS definition.

Note 2: This value is taken from the UPS definition.

Output Short Voltage: The output voltage value and poles of the UPS.

Examples: 208Y/120V 1P, 208V 2P

Output Volts: The full output voltage value of the UPS. The volts, poles, and wires will be displayed.

Examples: 208Y/120V 1P 2W, 208V 2P 2W

Output Volts Only: The output voltage value of the UPS. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 208Y/120V, 208V

Power Factor: This value is taken from the UPS definition.

Taps: This value is taken from the UPS definition.

Total Calculated Amps: The total calculated load in amps of the UPS.

Total Calculated KVA: The total calculated load in kVA of the UPS.

Total Connected Amps: The total connected load in amps of the UPS.

Total Connected KVA: The total connected load in kVA of the UPS.

Utility Fault: The utility fault in amps at the UPS. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Edit Motor Control Center List

To edit the motor control center list project settings, go to

Ribbon: Ribbon: `DME: Customization->Project Customization->` MCC LABEL `Motor Control Center List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Motor Control Center List`

To edit motor control center/disconnect list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Distribution Equipment Master Groups`

Motor Control Center List Label Display and Order Dialog Box

Key	Label	Width	Justification	Display
Schedule Title	MOTOR CONTROL CENTER SCHEDULE			☑
Callout	CALLOUT	1	Left	☑
Total Connected KVA	TOTAL CONNECTED KVA	1.5	Left	☑
Total Calculated KVA	TOTAL CALCULATED KVA	1.5	Left	☑
Total Connected Amps	TOTAL CONNECTED AMPS	1.5	Left	☑
Total Calculated Amps	TOTAL CALCULATED AMPS	1.5	Left	☑
Volts	VOLTS	2	Left	☑
Horizontal Bus Amps	HORIZONTAL BUS AMPS	1.5	Left	☑
Vertical Bus Amps	VERTICAL BUS AMPS	1.5	Left	☑
Main Disconnect Type	MAIN DISCONNECT TYPE	1.5	Left	☑
Main Disconnect Amps	MAIN DISCONNECT AMPS	1.5	Left	☑
Mounting	MOUNTING	1.5	Left	☑
Neutral	NEUTRAL	1	Left	☑
Number of Taps	NUMBER OF TAPS	1	Left	☑
Number of Sections	SECTIONS	1	Left	☑
Section Height	SECTION HEIGHT	1	Left	☑
Note 1	NOTE 1	3	Left	☑
Note 2	NOTE 2	3	Left	☑
Location	LOCATION	1	Left	☑
AIC	AIC RATING	1	Left	☑
Upstream OCP	UPSTREAM OCP	1.5	Left	☑
Feeder Size	FEEDER SIZE	4	Left	☑
Upstream Device	UPSTREAM DEVICE	1.5	Left	☑
Upstream Circuit	UPSTREAM CIRCUIT	1.5	Left	☑
Upstream Breaker and Poles	UPSTREAM BREAKER AND POLES	1.5	Left	☑
List Note 1	LIST NOTE 1	3	Left	☐
List Note 2	LIST NOTE 2	3	Left	☐
List Note 3	LIST NOTE 3	3	Left	☐
Volts Only	VOLTS	1	Left	☐
Short Voltage	VOLTS	1	Left	☐
Fault	FAULT	1	Left	☐
Utility Fault	UTILITY FAULT	1	Left	☐
Manufacturer	MANUFACTURER	3	Left	☐
Model	MODEL	3	Left	☐
NEMA	NEMA	3	Left	☐

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the motor control center definition.

Callout: The name of the motor control center.

551

Fault: The total fault in amps at the motor control center, including the fault from the utility and the fault from any motors in the system.

Feeder Size: This value is taken from the motor control center definition.

Horizontal Bus Amps: This value is taken from the motor control center definition.

List Note 1: This value is taken from the motor control center definition.

List Note 2: This value is taken from the motor control center definition.

List Note 3: This value is taken from the motor control center definition.

Location: This value is taken from the motor control center definition.

Main Disconnect Amps: This value is taken from the motor control center definition.

Main Disconnect Type: This value is taken from the motor control center definition.

Manufacturer: This value is taken from the motor control center definition.

Model: This value is taken from the motor control center definition.

Mounting: This value is taken from the motor control center definition.

NEMA: This value is taken from the motor control center definition.

Neutral: This value is taken from the motor control center definition.

Note 1: This value is taken from the motor control center definition.

Note 2: This value is taken from the motor control center definition.

Number of Sections: This value is taken from the motor control center definition.

Number of Taps: This value is taken from the motor control center definition.

Section Height: This value is taken from the motor control center definition.

Short Voltage: The voltage value and poles of the motor control center.

Examples: 208Y/120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the motor control center.

Total Calculated KVA: The total calculated load in kVA of the motor control center.

Total Connected Amps: The total connected load in amps of the motor control center.

Total Connected KVA: The total connected load in kVA of the motor control center.

Upstream Motor Control Center and Poles: The size of the breaker and number of poles for the motor control center feeder.

Examples: 20/1, 30/3

Upstream Circuit: The circuit to which the motor control center is connected on the upstream device.

Upstream Device: The upstream device to which the motor control center is connected.

Upstream OCP: This value is taken from the motor control center definition.

Utility Fault: The utility fault in amps at the motor control center. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the motor control center. The volts, poles, and wires will be displayed.

Examples: 208Y/120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the motor control center. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 208Y/120V, 208V

Edit Meter Center List

To edit meter center list project settings, go to

Ribbon: `DME: Customization->Project Customization->` MC LABEL `Meter Center List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Meter Center List`

To edit meter center list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Distribution Equipment Master Groups`

Meter Center List Label Display and Order Dialog Box

Key	Label	Width	Justification	Display
Schedule Title	General METER CENTER SCHEDULE			☑
Callout	CALLOUT	1	Left	☑
Total Connected KVA	TOTAL CONNECTED KVA	1.5	Left	☑
Total Calculated KVA	TOTAL CALCULATED KVA	1.5	Left	☑
Total Connected Amps	TOTAL CONNECTED AMPS	1.5	Left	☑
Total Calculated Amps	TOTAL CALCULATED AMPS	1.5	Left	☑
Volts	VOLTS	2	Left	☑
Bus Amps	BUS AMPS	1	Left	☑
Main Disconnect Type	MAIN DISCONNECT TYPE	1.5	Left	☑
Main Disconnect Amps	MAIN DISCONNECT AMPS	1.5	Left	☑
Mounting	MOUNTING	1.5	Left	☑
Neutral	NEUTRAL	1	Left	☑
Number of Taps	NUMBER OF TAPS	1	Left	☑
Lugs	LUGS	1	Left	☑
IG Bus	IG BUS	1	Left	☑
Note 1	NOTE 1	3	Left	☑
Note 2	NOTE 2	3	Left	☑
Location	LOCATION	1	Left	☑
AIC	AIC RATING	1	Left	☑
Upstream Device	UPSTREAM DEVICE	1.5	Left	☑
Upstream Circuit	UPSTREAM CIRCUIT	1.5	Left	☑
Upstream OCP	UPSTREAM OCP	1.5	Left	☑
Upstream Breaker and Poles	UPSTREAM BREAKER AND POLES	1.5	Left	☑
Feeder Size	FEEDER SIZE	4	Left	☑
List Note 1	LIST NOTE 1	3	Left	☐
List Note 2	LIST NOTE 2	3	Left	☐
List Note 3	LIST NOTE 3	3	Left	☐
Volts Only	VOLTS	1	Left	☐
Short Voltage	VOLTS	1	Left	☐
Fault	FAULT	1	Left	☐
Utility Fault	UTILITY FAULT	1	Left	☐
Manufacturer	MANUFACTURER	3	Left	☐
Model	MODEL	3	Left	☐
NEMA	NEMA	3	Left	☐

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the meter center definition.

Bus Amps: This value is taken from the meter center definition.

Callout: The name of the meter center.

Fault: The total fault in amps at the meter center, including the fault from the utility and the fault from any motors in the system.

Feeder Size: This value is taken from the meter center definition.

IG Bus: This value is taken from the meter center definition.

List Note 1: This value is taken from the meter center definition.

List Note 2: This value is taken from the meter center definition.

List Note 3: This value is taken from the meter center definition.

Location: This value is taken from the meter center definition.

Lugs: This value is taken from the meter center definition.

Main Disconnect Amps: This value is taken from the meter center definition.

Main Disconnect Type: This value is taken from the meter center definition.

Manufacturer: This value is taken from the meter center definition.

Model: This value is taken from the meter center definition.

Mounting: This value is taken from the meter center definition.

NEMA: This value is taken from the meter center definition.

Neutral: This value is taken from the meter center definition.

Note 1: This value is taken from the meter center definition.

Note 2: This value is taken from the meter center definition.

Number of Taps: This value is taken from the meter center definition.

Short Voltage: The voltage value and poles of the meter center.

Examples: 208Y/120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the meter center.

Total Calculated KVA: The total calculated load in kVA of the meter center.

Total Connected Amps: The total connected load in amps of the meter center.

Total Connected KVA: The total connected load in kVA of the meter center.

Upstream Breaker and Poles: The size of the breaker and number of poles for the meter center feeder.

Examples: 20/1, 30/3

Upstream Circuit: The circuit to which the meter center is connected on the upstream device.

Upstream Device: The upstream device to which the meter center is connected.

Upstream OCP: This value is taken from the meter center definition.

Utility Fault: The utility fault in amps at the meter center. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the meter center. The volts, poles, and wires will be displayed.

Examples: 208Y/120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the meter center. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 208Y/120V, 208V

Edit Generator List

To edit generator list project settings, go to

Ribbon: `DME: Customization->Project Customization->` GEN LABEL `Generator List Label Group`
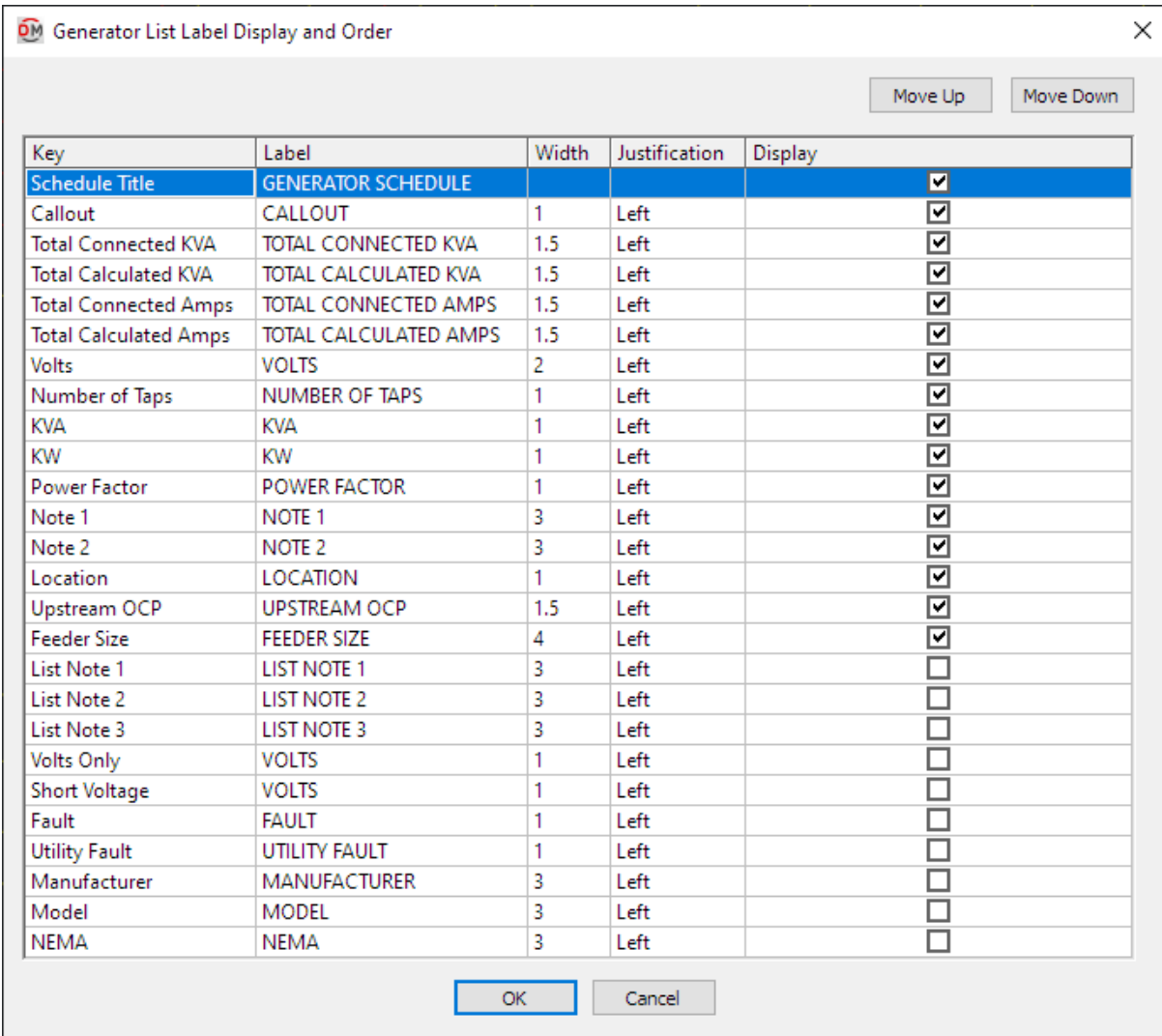
Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Generator List`

To edit generator list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Distribution Equipment Master Groups`

Generator List Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Callout: The name of the generator.

Fault: The total fault in amps at the generator, including the fault from the utility and the fault from any motors in the system.

Feeder Size: This value is taken from the generator definition.

KVA: This value is taken from the generator definition.

KW: This value is taken from the generator definition.

List Note 1: This value is taken from the generator definition.

List Note 2: This value is taken from the generator definition.

List Note 3: This value is taken from the generator definition.

Location: This value is taken from the generator definition.

Manufacturer: This value is taken from the generator definition.

Model: This value is taken from the generator definition.

NEMA: This value is taken from the generator definition.

Note 1: This value is taken from the generator definition.

Note 2: This value is taken from the generator definition.

Number of Taps: This value is taken from the generator definition.

Power Factor: This value is taken from the generator definition.

Short Voltage: The voltage value and poles of the generator.

Examples: 208Y/120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the generator.

Total Calculated KVA: The total calculated load in kVA of the generator.

Total Connected Amps: The total connected load in amps of the generator.

Total Connected KVA: The total connected load in kVA of the generator.

Upstream OCP: This value is taken from the generator definition.

Utility Fault: The utility fault in amps at the generator. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the generator. The volts, poles, and wires will be displayed.

Examples: 208Y/120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the generator. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 208Y/120V, 208V

Edit Transfer Switch List

To edit the transfer switch list project settings, go to

Ribbon: `DME: Customization->Project Customization->` TS LABEL `Transfer Switch List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Transfer Switch List`

To edit transfer switch list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->`Edit Distribution Equipment Master Groups

Transfer Switch List Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the , *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Amps: This value is taken from the transfer switch definition.

Callout: The name of the transfer switch.

Emergency Feeder Size: This value is taken from the transfer switch definition.

Emergency Upstream Breaker and Poles: The size of the breaker and number of poles for the transfer switch emergency feeder.

Examples: 20/1, 30/3

Emergency Upstream Circuit: The circuit to which the transfer switch is connected on the upstream device for emergency power.

Emergency Upstream Device: The upstream device to which the transfer switch is connected for emergency power.

Emergency Upstream OCP: This value is taken from the transfer switch definition.

Fault: The total fault in amps at the transfer switch, including the fault from the utility and the fault from any motors in the system.

IG Bus: This value is taken from the transfer switch definition.

List Note 1: This value is taken from the transfer switch definition.

List Note 2: This value is taken from the transfer switch definition.

List Note 3: This value is taken from the transfer switch definition.

Location: This value is taken from the transfer switch definition.

Lugs: This value is taken from the transfer switch definition.

Mounting: This value is taken from the transfer switch definition.

NEMA: This value is taken from the transfer switch definition.

Neutral: This value is taken from the transfer switch definition.

Normal Feeder Size: This value is taken from the transfer switch definition.

Normal Upstream Breaker and Poles: The size of the breaker and number of poles for the transfer switch normal feeder.

Examples: 20/1, 30/3

Normal Upstream Circuit: The circuit to which the transfer switch is connected on the upstream device for normal power.

Normal Upstream Device: The upstream device to which the transfer switch is connected for normal power.

Normal Upstream OCP: This value is taken from the transfer switch definition.

Note 1: This value is taken from the transfer switch definition.

Note 2: This value is taken from the transfer switch definition.

Number of Taps: This value is taken from the transfer switch definition.

Short Voltage: The voltage value and poles of the transfer switch.

Examples: 208Y/120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the transfer switch.

Total Calculated KVA: The total calculated load in kVA of the transfer switch.

Total Connected Amps: The total connected load in amps of the transfer switch.

Total Connected KVA: The total connected load in kVA of the transfer switch.

Type: This value is taken from the transfer switch definition.

Utility Fault: The utility fault in amps at the transfer switch. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the transfer switch. The volts, poles, and wires will be displayed.

Examples: 208Y/120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the transfer switch. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 208Y/120V, 208V

Edit Low Voltage Panel List

To edit the low voltage panel list project settings, go to

Ribbon: `DME: Customization->Project Customization->` LVD LABEL `Low Voltage Panel List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Low Voltage Panel List`

To edit low voltage panel list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Distribution Equipment Master Groups`

Low Voltage Panel List Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

Callout: The name of the low voltage panel.

Group: This value is taken from the low voltage panel definition.

List Note 1: This value is taken from the low voltage panel definition.

List Note 2: This value is taken from the low voltage panel definition.

List Note 3: This value is taken from the low voltage panel definition.

Location: This value is taken from the low voltage panel definition.

Manufacturer: This value is taken from the low voltage panel definition.

Model: This value is taken from the low voltage panel definition.

Mounting: This value is taken from the low voltage panel definition.

NEMA: This value is taken from the low voltage panel definition.

Note 1: This value is taken from the low voltage panel definition.

Note 2: This value is taken from the low voltage panel definition.

Number of Taps: This value is taken from the low voltage panel definition.

Total Calculated Amps: The total calculated load in amps of the low voltage panel.

Total Calculated KVA: The total calculated load in kVA of the low voltage panel.

Total Connected Amps: The total connected load in amps of the low voltage panel.

Total Connected KVA: The total connected load in kVA of the low voltage panel.

Edit Recurring Panel Template List

To edit recurring panel template list project settings, go to

Ribbon: `DME: Customization->Project Customization->` PAN LABEL `Recurring Panel Template List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Recurring Panel Template List`

To edit recurring panel template list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Distribution Equipment Master Groups`

Recurring Panel Template List Label Display and Order Dialog Box

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the recurring panel template definition.

Bus Amps: This value is taken from the recurring panel template definition.

Callout: The name of the recurring panel template.

Fault: The total fault in amps at the recurring panel template, including the fault from the utility and the fault from any motors in the system.

Feeder Size: This value is taken from the recurring panel template definition.

IG Bus: This value is taken from the recurring panel template definition.

Instance, Line: Lists the recurring panel instances based upon the template. Instances will be listed on one line, separated by commas.

Instance, List: Lists the recurring panel instances based upon the template. Each instance will be listed on a separate line.

List Note 1: This value is taken from the recurring panel template definition.

List Note 2: This value is taken from the recurring panel template definition.

List Note 3: This value is taken from the recurring panel template definition.

Location: This value is taken from the recurring panel template definition.

Lugs: This value is taken from the recurring panel template definition.

Main Disconnect Amps: This value is taken from the recurring panel template definition.

Main Disconnect Type: This value is taken from the recurring panel template definition.

Manufacturer: This value is taken from the recurring panel template definition.

Model: This value is taken from the recurring panel template definition.

Mounting: This value is taken from the recurring panel template definition.

NEMA: This value is taken from the recurring panel template definition.

Note 1: This value is taken from the recurring panel template definition.

Note 2: This value is taken from the recurring panel template definition.

Panel Neutral: This value is taken from the recurring panel template definition.

Poles: This value is taken from the recurring panel template definition.

Short Voltage: The voltage value and poles of the recurring panel template.

Examples: 120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the recurring panel template.

Total Calculated KVA: The total calculated load in kVA of the recurring panel template.

Total Connected Amps: The total connected load in amps of the recurring panel template.

Total Connected KVA: The total connected load in kVA of the recurring panel template.

Upstream Breaker and Poles: The size of the breaker and number of poles for the recurring panel template feeder.

Examples: 20/1, 30/3

Upstream Circuit: The circuit to which the recurring panel template is connected on the upstream device.

Upstream Device: The upstream device to which the recurring panel template is connected.

Upstream OCP: This value is taken from the recurring panel template definition.

Utility Fault: The utility fault in amps at the recurring panel template. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the recurring panel template. The volts, poles, and wires will be displayed.

Examples: 120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the recurring panel template. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 120V, 208V

Edit Recurring Panel Instance List

To edit recurring panel instance list project settings, go to

Ribbon: `DME: Customization->Project Customization->` PAN LABEL `Recurring Panel Instance List Label Group`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Project List->Edit Recurring Panel Instance List`

To edit recurring panel instance list master settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Distribution Equipment Master Groups`

Recurring Panel Instance List Label Display and Order Dialog Box

Recurring Panel Instance List Label Display and Order

Key	Label	Width	Justification	Display
Schedule Title	GENERAL RECURRING PANEL INSTANCE SCHEDULE			☑
Callout	CALLOUT	1	Left	☑
Recurring Panel Template	RECURRING PANEL TEMPLATE	4	Left	☑
Total Connected KVA	TOTAL CONNECTED KVA	1.5	Left	☑
Total Calculated KVA	TOTAL CALCULATED KVA	1.5	Left	☑
Total Connected Amps	TOTAL CONNECTED AMPS	1.5	Left	☑
Total Calculated Amps	TOTAL CALCULATED AMPS	1.5	Left	☑
Volts	VOLTS	2	Left	☑
Volts Only	VOLTS	1	Left	☐
Short Voltage	VOLTS	1	Left	☐
Bus Amps	BUS AMPS	1	Left	☑
Main Disconnect Type	MAIN DISCONNECT TYPE	1.5	Left	☑
Main Disconnect Amps	MAIN DISCONNECT AMPS	1.5	Left	☑
Mounting	MOUNTING	1.5	Left	☑
Panel Neutral	PANEL NEUTRAL	1	Left	☑
Poles	POLES	1	Left	☑
Lugs	LUGS	1	Left	☑
IG Bus	IG BUS	1	Left	☑
Manufacturer	MANUFACTURER	3	Left	☐
Model	MODEL	3	Left	☐
NEMA	NEMA	3	Left	☐
Note 1	NOTE 1	3	Left	☑
Note 2	NOTE 2	3	Left	☑
List Note 1	LIST NOTE 1	3	Left	☑
List Note 2	LIST NOTE 2	3	Left	☑
List Note 3	LIST NOTE 3	3	Left	☑
Location	LOCATION	1	Left	☑
AIC	AIC RATING	1	Left	☑
Upstream Device	UPSTREAM DEVICE	1.5	Left	☑
Upstream Circuit	UPSTREAM CIRCUIT	1.5	Left	☑
Upstream OCP	UPSTREAM OCP	1.5	Left	☑
Upstream Breaker and Poles	UPSTREAM BREAKER AND POLES	1.5	Left	☑
Feeder Size	FEEDER SIZE	4	Left	☑
Fault	FAULT	1	Left	☐
Utility Fault	UTILITY FAULT	1	Left	☐

See the **Common Label and Schedule Information** section for more information about using the **Move Up** and **Move Down** buttons and the *Key*, *Label*, *Width*, *Display,* and *Schedule Title* fields.

Values in the *Key* column are listed alphabetically below.

AIC: This value is taken from the recurring panel instance definition.

Bus Amps: This value is taken from the recurring panel instance definition.

Callout: The name of the recurring panel instance.

Fault: The total fault in amps at the recurring panel instance, including the fault from the utility and the fault from any motors in the system.

Feeder Size: This value is taken from the recurring panel instance definition.

IG Bus: This value is taken from the recurring panel instance definition.

List Note 1: This value is taken from the recurring panel instance definition.

List Note 2: This value is taken from the recurring panel instance definition.

List Note 3: This value is taken from the recurring panel instance definition.

Location: This value is taken from the recurring panel instance definition.

Lugs: This value is taken from the recurring panel instance definition.

Main Disconnect Amps: This value is taken from the recurring panel instance definition.

Main Disconnect Type: This value is taken from the recurring panel instance definition.

Manufacturer: This value is taken from the recurring panel instance definition.

Model: This value is taken from the recurring panel instance definition.

Mounting: This value is taken from the recurring panel instance definition.

NEMA: This value is taken from the recurring panel instance definition.

Note 1: This value is taken from the recurring panel instance definition.

Note 2: This value is taken from the recurring panel instance definition.

Panel Neutral: This value is taken from the recurring panel instance definition.

Poles: This value is taken from the recurring panel instance definition.

Recurring Panel Instance: The template the recurring panel instance is based upon.

Short Voltage: The voltage value and poles of the recurring panel instance.

Examples: 120V 1P, 208V 2P

Total Calculated Amps: The total calculated load in amps of the recurring panel instance.

Total Calculated KVA: The total calculated load in kVA of the recurring panel instance.

Total Connected Amps: The total connected load in amps of the recurring panel instance.

Total Connected KVA: The total connected load in kVA of the recurring panel instance.

Upstream Breaker and Poles: The size of the breaker and number of poles for the recurring panel instance feeder.

Examples: 20/1, 30/3

Upstream Circuit: The circuit to which the recurring panel instance is connected on the upstream device.

Upstream Device: The upstream device to which the recurring panel instance is connected.

Upstream OCP: This value is taken from the recurring panel instance definition.

Utility Fault: The utility fault in amps at the recurring panel instance. Use this column instead of the *Fault* column if you do not want to include motor contributions.

Volts: The full voltage value of the recurring panel instance. The volts, poles, and wires will be displayed.

Examples: 120V 1P 2W, 208V 2P 2W

Volts Only: The voltage value of the recurring panel instance. Only the volts will be displayed. The poles and wires will not be displayed.

Examples: 120V, 208V

Edit Distribution Equipment Master Groups

The `Edit Distribution Equipment Master Groups` command is used to make changes to the master schedule groups and list settings for distribution equipment.

To edit distribution equipment master groups settings, go to

Ribbon: `DME: Customization->Master & Standards Customization->` PAN LABEL `Edit Distribution Equipment Master Groups`

Pulldown Menu: `DM Electrical->Customization->Labels and Schedules->Standards and Master Lists->Edit Distribution Equipment Master Groups`

Select Distribution Equipment Dialog Box
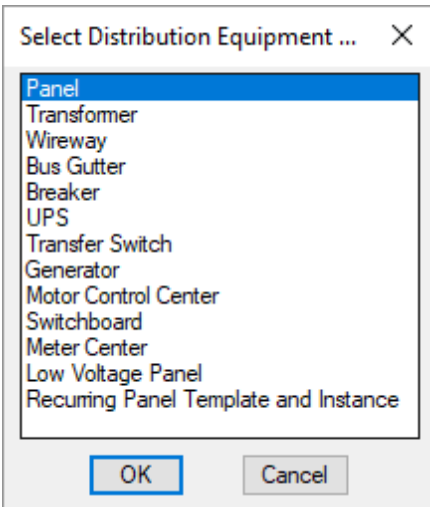
Select the distribution equipment type to which you want to make changes and press the **OK** button. The **<Distribution Equipment> Master Schedule Groups** dialog box will appear.

<Distribution Equipment> Master Schedule Groups Dialog Box
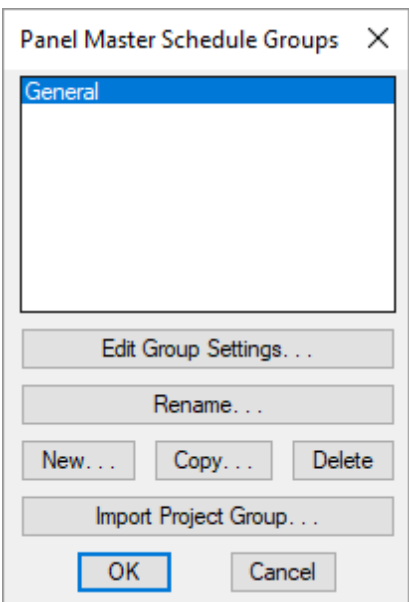
See the **Schedule Groups** section for more information about about how this dialog box works.

Edit Group/Template/Instance Settings: Press this button to open the **Insert <Distribution Equipment> List** dialog box.

Insert <Distribution Equipment> List Dialog Box

Insert Panel List

Schedule Notes:	[ ] Edit...
☐ Maximum Schedule Height:	[ ]

Edit Title, Column Labels, and Column Order...

[OK]   [Cancel]
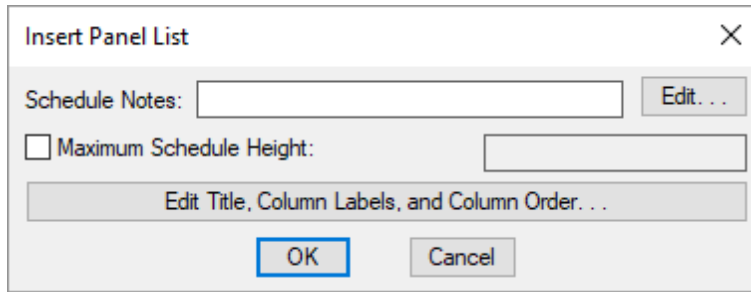
The **Insert <Distribution Equipment> List** dialog box is used to change the master database settings for how the selected distribution equipment list is displayed when it is inserted on the drawing.

Schedule Notes: A note that will appear below the schedule when inserted on the drawing.

Maximum Schedule Height: Whether the schedule has a maximum height. This height corresponds to inches on the printed page.

If this is not checked, the schedule will be in a single schedule.

If this is checked, you can specify the maximum height of the schedule. When the schedule exceeds this height, it will be continued in a second schedule next to the first. The label of the second schedule will have the *Schedule Title Continued Label* option added to it to indicate that it is a continuation of the first schedule. See the **Schedules** options section for more information.

Edit Title, Column Labels, and Column Order: Press this button to modify the layout of the schedule on the drawing. The **Label Display and Order** dialog box will appear for the selected distribution equipment list.

Text Styles

The `Text Styles` commands allow you to customize the text heights and fonts used for labels and schedules.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Standards List`, `Copy Project List to Standards`, and `Copy Standards List to Project` commands work.

To customize text styles in the current project, go to

Ribbon: `DME: Customization->Project Customization->` TEXT P `Text Styles`

Pulldown Menu: `DM Electrical->Customization->Text Styles->Edit Project List`

To customize standard text styles, go to

Ribbon: `DME: Customization->Master & Standards Customization->` TEXT S `Text Styles`

Pulldown Menu: `DM Electrical->Customization->Text Styles->Edit Standards List`

Text Styles Dialog Box

Text Style Key: This list indicates where the text style is used. These values cannot be edited.

Text Style: Sets the text style in your CAD program to be associated with the corresponding *Text Style Key*. The **Default** key must have a text style defined. The text style in other keys may be left blank. Blank keys will use the text style defined for the **Default** key.

If the text style currently exists on the drawing, it will be used as defined.

If the text style does not exist, it will be created using an SHX file of the same name. If an SHX file cannot be found, the **Standard** text style will be used.

To use a TrueType font, create the text style and associate it with the desired font.

Height: Sets the text height for the key. The **Default** key must have a height defined. The height in other keys may be left blank. Blank keys will use the height defined for the **Default** key.

The units for the height are inches on the printed page. To set 1/10" text for the drawing, use the value **0.1**. To set 1/8" text, use **0.125**. When text is inserted, this value will be multiplied by `DIMSCALE` to determine the height of the text on the drawing.

Scale Factor: Sets the character spacing for the text. This value is not currently used.

Wire Sizing

The `Wire Sizing` command is used to customize the wires used in your project.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To customize the wire sizing tables in the current project, go to

Ribbon: `DME: Customization->Project Customization->` WIRE `P` `Wire Sizing`

Pulldown Menu: `DM Electrical->Customization->Wire Sizing->Edit Project List`

To customize the master wire sizing tables, go to

Ribbon: `DME: Customization->Master & Standards Customization->` WIRE `M` `Wire Sizing`

Pulldown Menu: `DM Electrical->Customization->Wire Sizing->Edit Master List`

Training Videos

- **Wire Sizing Customization**

Wire Sizing Dialog Box

The **Wire Sizing** dialog box consists of four sections.

The top-left section is used to specify the sizing options. Each sizing option controls which conduit sizes and breaker sizes are used by default when sizing feeders and branch circuits in your project.

The top-middle section is used to specify the wire sizes available.

The top-right section is used to specify the conduit sizes available. The sizing option controls whether each conduit size is used when sizing feeders and branch circuits automatically.

The bottom section is used to specify the breaker and feeder sizes available. The sizing option controls whether each breaker size is used when sizing feeders and branch circuits automatically.
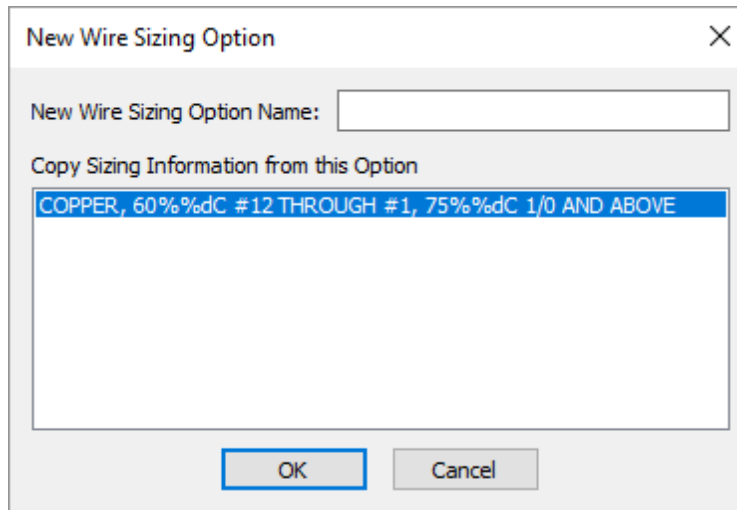
Sizing Option

This section of the dialog box provides a list of all of the sizing options. Each sizing option controls what is used when sizing wires automatically. In the *Conduit Size* list, the *Standard Branch* and *Standard Feeder* columns are based upon the sizing option. In the *Breakers and Conductors* list, the *Standard Branch Circuit*, *Equipment Branch Circuit*, and *Standard Feeder* columns are based upon the sizing option. Selecting a different sizing option will change the settings of all of those columns.

Move Up: Press this button to move the selected row up in the *Sizing Option* list.

Move Down: Press this button to move the selected row down in the *Sizing Option* list.

New Option: Press this button to create a new sizing option. The **New Wire Sizing Option** dialog box will appear.

> **New Wire Sizing Option Name:** The name of the new sizing option.
>
> **Copy Sizing Information from this Option:** The sizing option from which to copy settings. This sets the starting values for the columns controlled by the sizing option.

Delete Option: Press this button is delete the selected sizing option.

Sizing Option: The name of the sizing option.

This value can be displayed at the bottom of the feeder schedule inserted on the drawing. See the **Insert Feeder Schedule** section for more information.

The default wire sizing table uses copper conductors, rigid steel conduit, and references *NEC Table 310.16:* 60°C #12 through #1, 75°C 1/0 and above.

Current: The sizing option that is active in the project. Branch circuit and feeder sizes are chosen based upon the settings of the selected sizing option. Only one sizing option can be active in a project at a time.

Wire Size

This section of the dialog box provides a list of all of the wire sizes.

New Wire Size: Press this button to create a new wire size. The new row will be added to the bottom of the list when it is created. When you close and reopen the dialog box, the size will be sorted to the correct location in the list based upon the area.

Delete Wire Size: Press this button to delete the selected wire size.

Undo Wire Changes: Press this button to revert the wire size settings to their values when the dialog box was opened.

Wire Size: The name of the wire size. This label is shown in the wire callout.

Schedule Label: The label for the wire size when it is shown in a schedule. The wire size can be shown in the panel schedule, fault schedule, and voltage drop schedule.

Wire + Insulation Area: The area of the wire with insulation, in square inches. See *NEC Table 5* and *NEC Table 5A*. Used to calculate the conduit fill.

Wire Only Area: The area of the wire itself, in circular mils.

Group: The type of material for the wire size. If the ground wire is upsized, it will be upsized to the next wire size in the same *Group*.

Conduit Size

This section of the dialog box provides a list of all of the conduit sizes.

New Conduit Size: Press this button to create a new conduit size. The new row will be added to the bottom of the list when it is created. When you close and reopen the dialog box, the size will be sorted to the correct location in the list based upon the area.

Delete Conduit Size: Press this button to delete the selected conduit.

Undo Conduit Changes: Press this button to revert the conduit settings to their values when the dialog box was opened.

Conduit Size: The name of the conduit size. The conduit size is included at the front of wire callouts that include conduit.

Conduit Area: The area of the conduit, in square inches. See *NEC Table 4*. Used to calculate the conduit fill. Conduits are sized automatically so that the maximum fill is 40%, as specified in *NEC Table 1*.

Standard Branch: Whether the conduit size is used when automatically sizing branch circuits.

Standard Feeder: Whether the conduit size is used when automatically sizing feeders.

Breakers and Conductors

This section of the dialog box provides a list of the breaker sizes and corresponding conductors.

Multiple breakers of a single size may be included in the list. One breaker can be the default used when sizing branch circuits and feeders automatically. The other breakers can be chosen when sizing them manually.

Move Up: Press this button to move the selected row up in the list. Breaker sizes can be moved within the breaker of the same size. You cannot move breakers so that the sizes are no longer in numerical order.

Move Down: Press this button to move the selected row down in the list. Breaker sizes can be moved within the breaker of the same size. You cannot move breakers so that the sizes are no longer in numerical order.

New Breaker: Press this button to add a new breaker.

Delete Breaker: Press this button to delete the selected breaker.

Undo Breaker Changes: Press this button to revert the breaker settings to their values when the dialog box was opened.

Amperage: The breaker size.

Wire Size: The size of the phase wires for the breaker. The list of wires is set in the *Wire Size* section of the dialog box.

Service Ground: The size of the service ground for the breaker. This sizing should be selected based upon *NEC Table 250.102(C)(1)*. This wire size is used for the ground when the distribution equipment is connected to the utility or to a transformer. The list of wires is set in the *Wire Size* section of the dialog box.

Equipment Ground: The size of the equipment ground for the breaker. This sizing should be selected based upon *NEC Table 250.122*. This wire size is used for the ground when the distribution equipment is connected to anything except the utility or a transformer. The list of wires is set in the *Wire Size* section of the dialog box.

Parallel Runs: The number of parallel runs of conduit for the breaker. Each run contains the same size wires.

X: The reactance of the breaker. See *NEC Table 9* for values. The reactance must include an adjustment for the number of parallel runs.

R: The resistance of the breaker. See *NEC Table 9* for values. The reactance must include an adjustment for the number of parallel runs.

Standard Branch Circuit: Whether the breaker size is used when automatically sizing standard branch circuits. Only one breaker of each size can have this box checked. You do not need to have it checked for each size breaker. Breaker sizes that are not checked can still be selected manually.

Equipment Branch Circuit: Whether the breaker size is used when automatically sizing branch circuits based upon the MCA and MOCP of equipment connected to them. Only one breaker of each size can have this box checked. You do not need to have it checked for each size breaker. Breaker sizes that are not checked can still be selected manually.

Standard Feeder: Whether the breaker size is used when automatically sizing feeders to distribution equipment. Only one breaker of each size can have this box checked. You do not need to have it checked for each size breaker. Breaker sizes that are not checked can still be selected manually.

Include Conduit: Whether the wire size label for this breaker includes the conduit size.

Description: A label that is added after the *Amperage* when breakers are listed in other dialog boxes. When manually selecting a breaker, the list will include all of the breaker sizes. If there are two definitions of the same size breaker, there is no way to tell them apart. This label is added to help you choose the correct breaker. This label is not displayed on the drawing.

Conduit Groups

The `Conduit Groups` command is used to create conduit groups. Conduit groups are used to insert conduits on separate layers.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands

work.

To create or delete a conduit group in the current project, go to

Ribbon: `DME: Customization->Project Customization->` ^{COND}`P` `Conduit Groups`

Pulldown Menu: `DM Electrical->Customization->Conduit Groups->Edit Project List`

To create or delete a standard conduit group, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ^{COND}`S` `Conduit Groups`

Pulldown Menu: `DM Electrical->Customization->Conduit Groups->Edit Standards List`

Conduit 3D Group Dialog Box
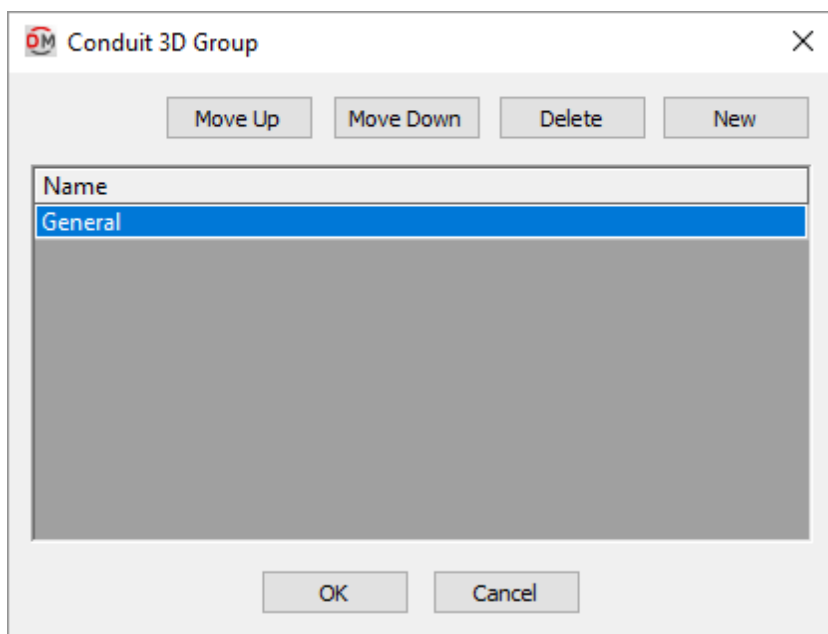
Name: The name of the conduit group.

Move Up: Press this button to move the selected group up in the list of groups.

Move Down: Press this button to move the selected group down in the list of groups.

Delete: Press this button to delete the selected group. If the group is currently used by a record in the schedule, you will be asked if you want to replace the group in the schedule with another group.

New: Press this button to create a new group. The **Select Conduit Group** dialog box will appear.
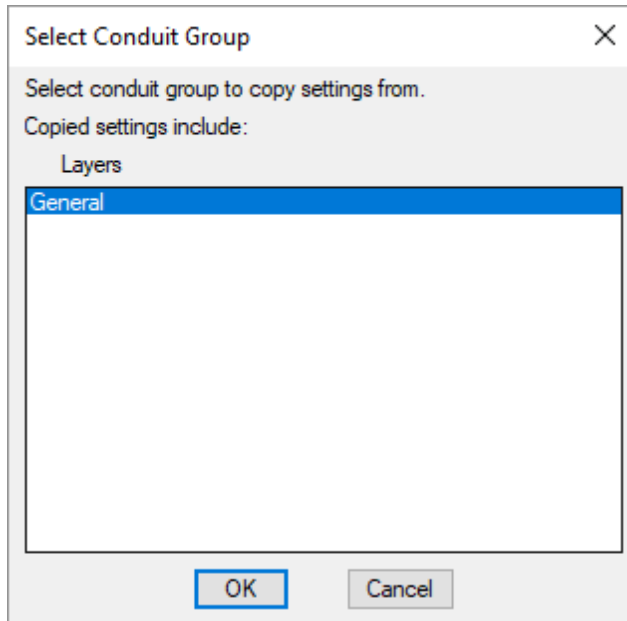
Select the conduit group to copy settings from and press the **OK** button. A new group will be created. The layers in the new group will match the layers in the copied group.

Cable Tray Groups

The `Cable Tray Groups` command is used to create cable tray groups. Cable tray groups are used to insert cable trays on separate layers.

See the **General Customization Commands** section for more information about how the `Edit Project List`, `Edit Master List`, `Copy Project List to Master`, and `Copy Master List to Project` commands work.

To create or delete a cable tray group in the current project, go to

Ribbon: `DME: Customization->Project Customization->` ^{TRAY} `P` `Cable Tray Groups`

Pulldown Menu: `DM Electrical->Customization->Cable Tray Groups->Edit Project List`

To create or delete a standard cable tray group, go to

Ribbon: `DME: Customization->Master & Standards Customization->` ^{TRAY} `S` `Cable Tray Groups`

Pulldown Menu: `DM Electrical->Customization->Cable Tray Groups->Edit Standards List`

Cable Tray Group Dialog Box
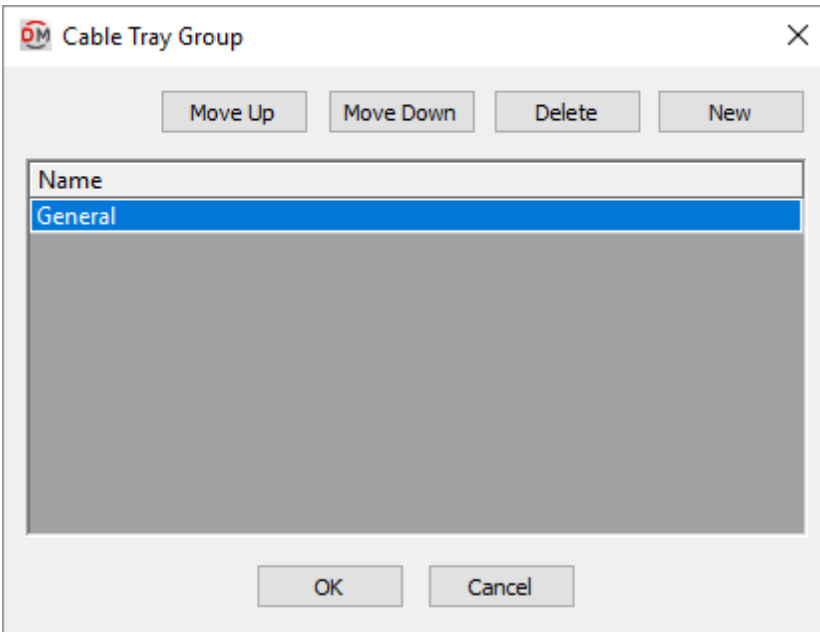
Name: The name of the cable tray group.

Move Up: Press this button to move the selected group up in the list of groups.

Move Down: Press this button to move the selected group down in the list of groups.

Delete: Press this button to delete the selected group. If the group is currently used by a record in the schedule, you will be asked if you want to replace the group in the schedule with another group.

New: Press this button to create a new group. The **Select Cable Tray Group** dialog box will appear.
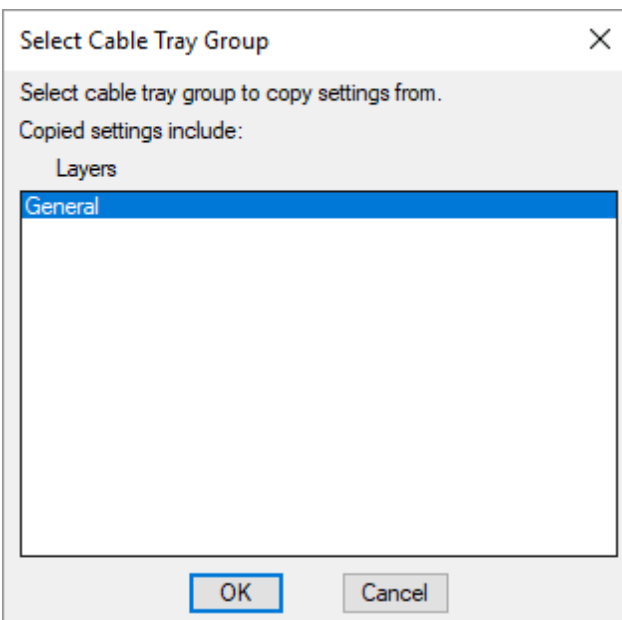
Select the cable tray group to copy settings from and press the **OK** button. A new group will be created. The layers in the new group will match the layers in the copied group.

Recreate Missing Customization

To recreate customization that is missing in the project or standards database, go to

Ribbon: `DME: Customization->Customization Utilities->Recreate Missing Customization`

Pulldown Menu: `DM Electrical->Customization->Recreate Missing Customization`
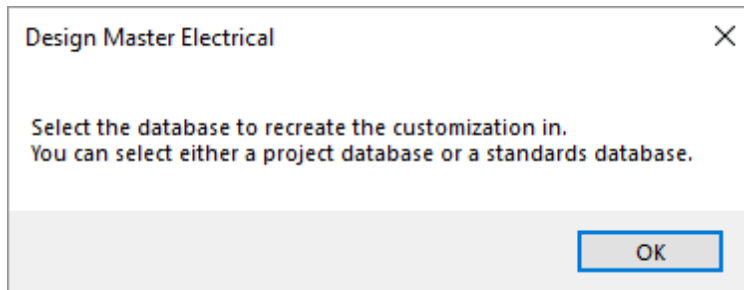
Press the **OK** button. The **Select Standards or Project Database** dialog box will appear.
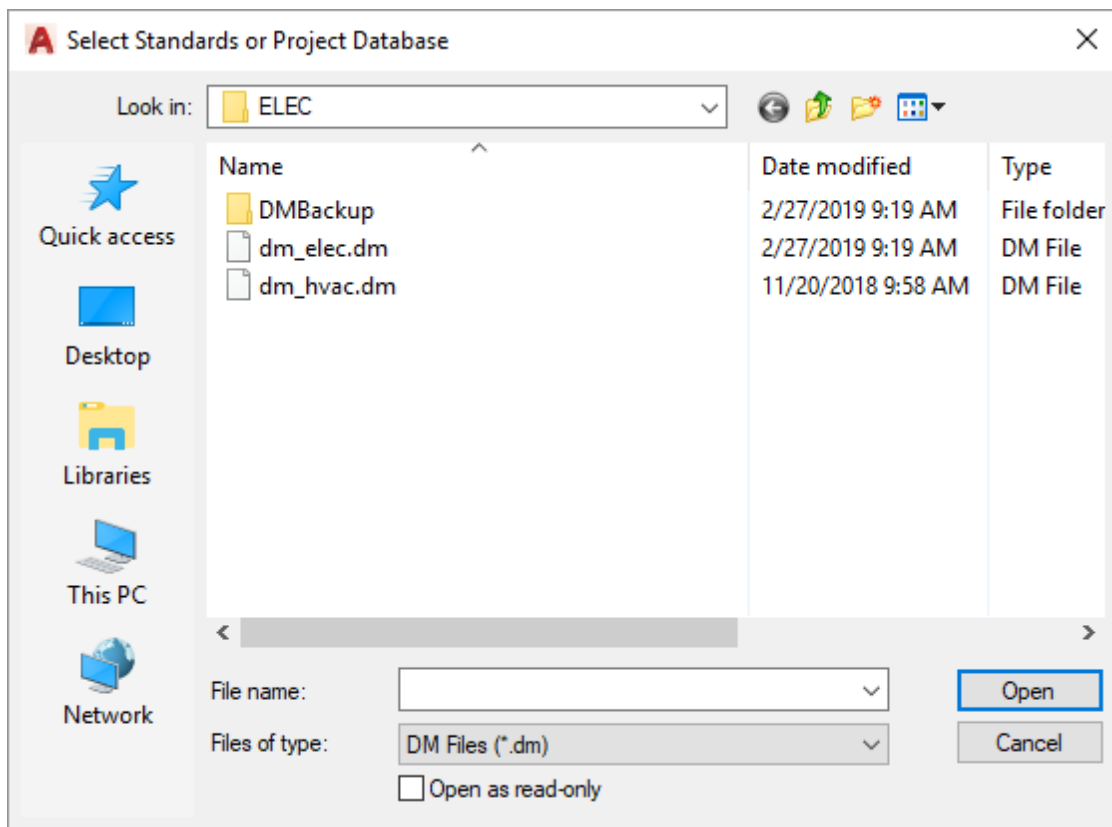
Select the standards or project database and press the **Open** button. Any missing customization records will be added to the database. It will not change any existing customization values.

The **Select Standards or Project Database** dialog box will open the current project folder by default. To update your standards database, you must browse to your customization folder.

The default location for the customization folder is `C:\Users\<User Name>\AppData\Roaming\Design Master Software\Electrical Customization` where <User Name> is the name you use to log in to your computer.

Update Laptop Customization

The `Update Laptop Customization` command is used to update the local customization on a laptop to match the customization stored on the network. The local customization is automatically updated once a day when you open your CAD program. This command is used to manually update the local customization. It is only necessary to use this command if the customization on your network has changed during the day.

To update the local customization on a laptop, go to

Ribbon: `DME: Customization->Customization Utilities->Update Laptop Customization`

Pulldown Menu: `DM Electrical->Customization->Update Laptop Customization`

The customization will be copied from the network to the local drive.

See the **Installation Settings** section for more information about configuring your computer as a laptop.

Manage Standards Databases

The `Manage Standards Databases` command is used to create, modify, and delete standards databases. Each standards database has its own set of customization settings. Multiple standards databases are used when more than one set of standards is used in a company.

To manage your standards databases, go to

Ribbon: `DME: Customization->Customization Utilities->Manage Standards Databases`

Pulldown Menu: `DM Electrical->Customization->Manage Standards Databases`

If only one standards database exists, you will be prompted to create a new one. If you do not create another database, the command will end. If you do create another database, the **Select Standards Database** dialog box will then appear.

If more than one standards database exists, the **Select Standards Database** dialog box will appear.

Select Standards Database Dialog Box
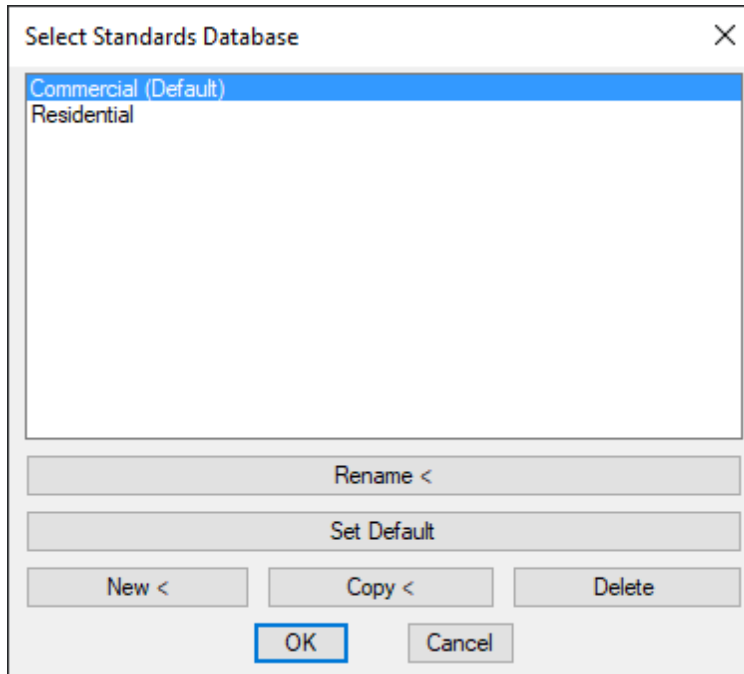
Each standards database is a separate file stored in your customization folder. The file name will be "dm_elec-standards<Name>.dm", where <Name> is the name of the standards database as displayed in the dialog box. If a standards database is named "dm_elec-standards.dm", it will display as "Default".

The default location for the customization folder is `C:\Users\<User Name>\AppData\Roaming\Design Master Software\Electrical Customization` where <User Name> is the name you use to log in to your computer.

Rename: Press this button to rename the selected standards database. The name that is displayed and the name of the file will be changed.

Rename: Press this button to set the selected standards database as the default when starting a new project.

New: Press this button to create a new standards database. A new standards database file will be created in the customization folder.

Copy: Press this button to copy the selected standards database. A new standards database file with the same settings will be created in the customization folder.

Delete: Press this button to delete the selected standards database.

Alignment Points

This section describes how to use alignment points and alignment point areas.

The alignment point is the origin of the coordinate system used for the devices inserted on the drawings. The location relative to the alignment point is used when exporting 3D-BIM elements or calculating distances between devices.

Make sure you set a location for the alignment point that will be easy to locate on all of the floors of the building

and that will not move during the course of the project. A corner of the building, a column, or the intersection of two architectural grids lines are all examples of good alignment point locations.

Multiple Alignment Points

A drawing can have more than one alignment point. The first alignment point inserted is used for all devices on the drawing. Additional alignment points are inserted with boxes, called alignment point areas, around them. All devices in the alignment point area are associated with the corresponding alignment point. Any devices not inside an alignment point area are associated with the first alignment point.

The following example shows a drawing file with multiple alignment point areas and floor plans. Notice how an alignment point area is drawn around the second alignment point.
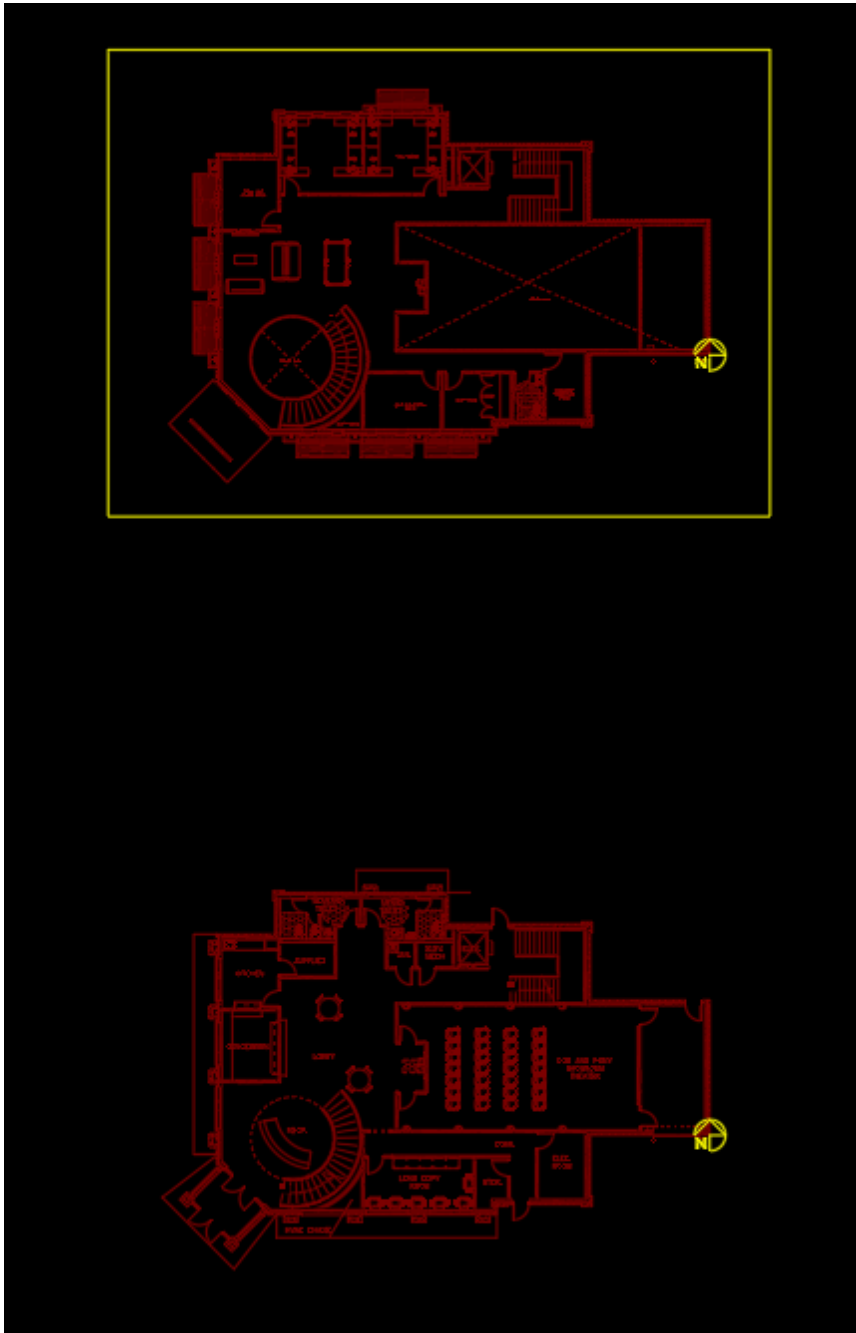
Common Alignment Point Information

The values that can be specified for alignment points are listed below.

Elevation: The elevation of the alignment point. Typically, this is the height of the floor. Device elevations are typically relative to this elevation. See the **Elevation** section for more information.

For example, consider a second floor area with light fixtures. The elevation of the alignment point is set to the height of the floor, 14'. The elevation of each light fixture is set to the elevation relative to the floor they are on, 8'.

The height of the light fixtures when exported or used for distance calculations would be the sum of the two heights: 14' + 8' = 22'.

Insert Alignment Point

The `Insert Alignment Point` command is used to insert alignment points and alignment point areas on the drawing. Multiple alignment points can be inserted on a drawing.

To insert an alignment point on the current drawing, go to

Ribbon: `DM Electrical->Alignment Points->` `Insert Alignment Point`

Pulldown Menu: `DM Electrical->Alignment Points->Insert Alignment Point`

Alignment Point Elevation Dialog Box
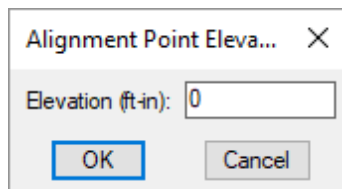
See the **Common Alignment Point Information** section for more information about the *Elevation* field.

Enter the *Elevation* and press the **OK** button. You will be prompted to identify the location of the alignment point on the drawing.

Specify insertion point:

The alignment point will be used to coordinate the current drawing with other drawings. The alignment point should be set in the same place on every drawing in the project folder. You will then be prompted to identify the direction of north on the drawing.

Specify rotation angle:

If this is the first alignment point inserted on the drawing, the command will end.

If there are other alignment points on the drawing, you will be prompted to specify the boundaries of the alignment point area.

First corner of alignment point area:

Second corner of alignment point area:

The alignment point area must surround its respective alignment point.

Query Alignment Point

To query and edit an existing alignment point, go to

Ribbon: `DM Electrical->Alignment Points->` `Query Alignment Point Area`

Pulldown Menu: `DM Electrical->Alignment Points->Query Alignment Point`

If there is only one alignment point on the drawing, the **Alignment Point Elevation** dialog box will appear.

If there is more than one alignment point on the drawing, you will be prompted to indicate which alignment point you want to query.

Specify point in alignment point area to use:

Select a point inside the alignment point area that corresponds to the alignment point you want to query.

The **Alignment Point Elevation** dialog box will appear.

Alignment Point Elevation Dialog Box
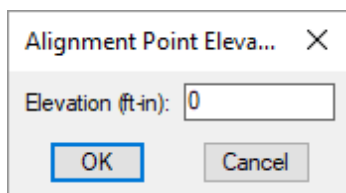
Changes can be made to the alignment point in the same way as when it was inserted. See the **Common Alignment Point Information** section for more information.

Move Alignment Point

The `Move Alignment Point` command can be used to move an alignment point on the drawing. You can also use standard CAD commands to move the alignment point.

To move an alignment point, go to

Ribbon: `DM Electrical->Alignment Points->` `Move Alignment Point`

Pulldown Menu: `DM Electrical->Alignment Points->Move Alignment Point`

If there is only one alignment point on the drawing, you will be prompted for the new location.

If there is more than one alignment point on the drawing, you will be prompted to indicate which alignment point you want to move.

Specify point in area to move alignment point in:

Select a point inside the alignment point area that corresponds to the alignment point you want to move.

You will then be prompted to indicate the new location for the alignment point.

Specify second point or <use first point as displacement>:

After the alignment point is moved, the location of all of the devices in the database will be updated to reflect the new location.

You cannot move an alignment point outside of its alignment point area.

Rotate Alignment Point

The `Rotate Alignment Point` command can be used to rotate an alignment point on the drawing. You can also use standard CAD commands to rotate the alignment point.

To rotate an alignment point, go to

Ribbon: `DM Electrical->Alignment Points->` `Rotate Alignment Point`

Pulldown Menu: `DM Electrical->Alignment Points->Rotate Alignment Point`

If there is only one alignment point on the drawing, you will be prompted for the new rotation angle.

If there is more than one alignment point on the drawing, you will be prompted to indicate which alignment point you want to rotate.

Specify point in area to rotate alignment point in:

Select a point inside the alignment point area that corresponds to the alignment point you want to rotate.

You will then be prompted to indicate the new rotation for the alignment point.

Specify rotation angle or [Copy/Reference] <0>:

After the alignment point is rotated, the location of all of the devices in the database will be updated to reflect the new rotation angle.

Offset Alignment Point for 3D Export

The `Offset Alignment Point for 3D Export` command can be used to specify the origin when exporting a drawing to 3D for collision detection. By default, the alignment point acts as the origin. Use this command if having the alignment point as the origin causes problems in the 3D export.

To offset the origin from an alignment point, go to

Ribbon: `DM Electrical->Alignment Points->` `Offset Alignment Point for 3D Export`

Pulldown Menu: `DM Electrical->Alignment Points->Offset Alignment Point for 3D Export`

You will be prompted to specify the offset relative to the first alignment point inserted on the drawing.

Specify origin for 3D export / <Reset origin to alignment point location>:

Specify a point on the drawing to act as the origin when the drawing is exported. Press **ENTER** to reset the origin to the location of the alignment point.

Remove Alignment Points from non-DM Drawing

The `Remove Alignment Points from non-DM Drawing` command is used to remove all alignment points from a drawing that does not use Design Master features. This command is the only way to fully remove alignment points from a drawing.

To remove all alignment points from a drawing, go to

Ribbon: `DM Electrical->Alignment Points->` `Remove Alignment Points from non-DM Drawing`

Pulldown Menu: `DM Electrical->Alignment Points->Remove Alignment Points from non-DM Drawing`

Remove Alignment Points from non-DM Drawing Dialog Box
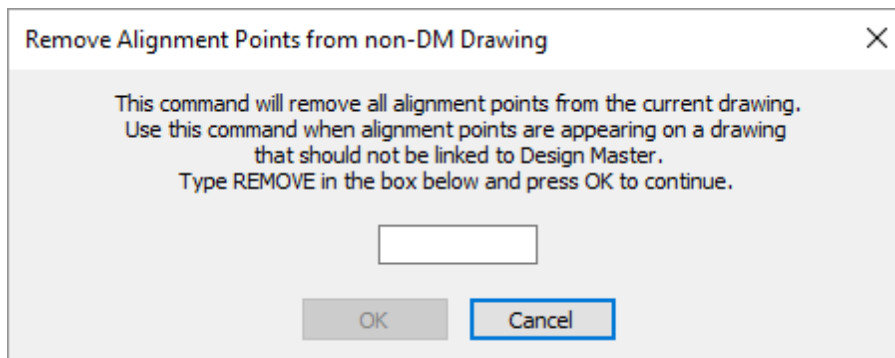
To remove the alignment points, type **REMOVE** and press the **OK** button. This step helps prevent you from removing the alignment points accidentally.

Start New Electrical Project

This section describes how to start a new project.

When you start a new project, a database file will be created in the same folder as the current drawing. The file name will be "dm_elec<Name>.dm", where <Name> is the name of the project. This is the database file where all of the information about your project is stored.

To create a new database file, go to

Ribbon: `DM Electrical->Utilities->` NEW PROJ `Start New Electrical Project`

Pulldown Menu: `DM Electrical->Start New Electrical Project`

Start New Electrical Project Dialog Box

Project Name (Optional): The project name that is added to the end of the file name. This name is only used on the file name. It is not used anywhere else in the project. If you leave this field blank, the project database name will be "dm_elec.dm".

Project Directory: The folder in which the database will be created. The folder displayed and used is the folder that contains the current drawing.

Standards Database: The standards database from which to copy project options and settings. See the **Master and Standards Databases** section for more information about the standards database and which settings are stored in it.

Groups to Import

This section is used to select groups of devices to automatically import from their respective master databases to the new project. See the **Master Database Groups** section for more information.

Multiple groups can be selected by holding down the CTRL key while making the selections.

Select Distribution Equipment Groups to Import: Press this button to open the **Select Distribution Equipment Groups to Import** dialog box.

Select the distribution equipment groups to be imported and press the **OK** button.

Room Types: Check this box to import a room group. See the **Room Master Schedule** section for more information.

Concepts and Procedures

This section describes concepts and procedures you will use with Design Master Electrical. It describes how to apply the commands in the **Command Reference** section to create your electrical project.

Definitions: Distribution Equipment and Branch Circuit Device

We have two definitions that are unique to our software: distribution equipment and branch circuit device. The electrical industry as a whole does not have words that match these terms. We use these terms in our software because they help organize how we create the model of your electrical system.

Distribution Equipment

Distribution equipment refers to the equipment in the project that provides power to other equipment or devices. It includes panels, transformers, switchboards, bus ducts, and other similar equipment.

The connections to distribution equipment are always referred to as feeders. The feeder sizes are set in the definition of the distribution equipment.

Fault calculations, voltage drop calculations, and the one-line diagram typically include distribution equipment and not branch circuit devices.

Branch Circuit Device

Branch circuit device refers to electrical devices that are connected to distribution equipment and do not have anything else connected to them. It includes receptacles, light fixtures, switches, mechanical equipment, kitchen equipment, and other similar devices.

The connections to branch circuit devices are referred to as branch circuits. The branch circuit sizes are set using the **Circuiting** command.

Equipment connections are an exception to branch circuit devices. They can be included in the fault calculation, voltage drop calculation, and on the one-line diagram.

Project Drawing and Database Management

When you create a new project, a database file will be created in the same folder as the current drawing. The file name will be "dm_elec<Name>.dm", where <Name> is the name of the project. This is the database file where all of the information about your project is stored. Your project drawings are now connected to this database file, and the information contained in the drawings and the database must match.

You must consider how changes to your drawings will affect the database. Recommendations for common scenarios when working on a Design Master Electrical project are listed below. You can also visit the **Project and Database Management Webinar** in the knowledge base.

Project Folders

Each project needs to be in a separate folder that contains one dm_elec.dm database file. All of the drawing files for the project need to be in the same folder as the database. You should not copy drawings from one project to another.

Saving Your Changes

Always save your changes to a drawing with Design Master Electrical devices in it. Never close a drawing without saving your changes.

Any changes you make to the drawing are saved to the database immediately. If you close the drawing without saving, the database and drawing will no longer match. This can result in missing or extra devices, incorrect calculations, and other problems.

If you close a drawing without saving, or your CAD program crashes, use the **Coordinate Drawings and Database** command to make the drawing and database match again.

Creating a Backup

To back up your project, you must copy both the drawings and the database file. Copying just the drawings is not enough.

The two simplest methods are copying the entire folder to a new location, or using the **Copy or Backup Project** command.

Alternative Designs

To try an alternative design in your project, do not copy drawings inside the same project folder. This will result in duplicate devices and inaccurate calculations. Instead, make a copy of the project in its own folder. Treat the alternative design as a separate project.

Copy the entire folder to a new location, or use the **Copy or Backup Project** command.

Copying a Drawing

To copy a drawing that contains Design Master devices, use the **Copy Drawing** command. This will create an exact copy of the drawing in the project folder. Copies of all of the devices on the drawing will be created in the database.

(You can use Windows Explorer or the standard CAD `SAVEAS` command to copy a drawing, but these methods add some complications so we do not recommend them.)

Renaming a Drawing

To rename a drawing that contains Design Master devices, use the **Rename Drawing** command. This will rename the drawing. The original drawing will no longer exist.

(You can use Windows Explorer or the standard CAD `SAVEAS` command to rename a drawing, but these methods add some complications so we do not recommend them.)

CAD Program Crashes

CAD program crashes can leave your drawing and your database in an inconsistent state. This is a special situation that needs to be handled carefully to prevent your database from being corrupted.

Your CAD Program Created a Recover Drawing

If your CAD program created a recovery drawing, try to open that file. If it opens, compare it with the original drawing file. Decide which you want to use. If you want to use the recovery drawing, rename the original drawing something different using Windows Explorer, then rename the recovery drawing to the original drawing name using Windows Explorer. Run the **Coordinate Drawings and Database** command on the drawing to make the drawing and database match, then continue working.

If you choose to work in the original drawing, delete any recovery drawings that were created. Run the **Coordinate Drawings and Database** command on the original drawing to make the drawing and database match, then continue working.

Your CAD Program Did Not Create a Recover Drawing

If your CAD program did not create a recover drawing, then you must open your original drawing. Run the **Coordinate Drawings and Database** command on the drawing to make the drawing and database match, then continue working.

Master and Standards Databases

The master and standards databases are used to store default settings for new projects. You can make changes to the customization settings, store them in these files, and reuse them.

These databases are stored in your electrical customization folder. For companies with more than one user, we recommend that the electrical customization folder be on your network. This allows you to share the customization between all of your users. Otherwise, synchronizing the customization on all of the different computers is complicated.

When you create a project, the settings from the master and standards databases are copied to its project database. Once it is created, the project database is no longer associated with the master or standards database. Changes to the project database will not affect the master or standards database, and changes to the master and standards databases will not affect the project database.

Master Database

There is only one master database. All of the blocks are stored in the master, as well as the various schedules.

When a project is created, all of the block records are copied from the master to the project database. You can choose to copy specific groups from each of the master schedules to the project database.

The master database stores the following information:
- All of the block lists
- Junction Box Master Schedule
- Light Fixture Master Schedule
- Low Voltage Device Master Schedule
- Receptacle Master Schedule

- Room Master Schedule
- Wire Sizing Master List
- Feeder IDs

Master Database Groups

Groups allow you to organize the records in your master schedule.

When you create a project, all of the records from a group can be copied to the project. Copying a group is helpful when you have a common type of project with similar schedules. You can set up the schedule once in the master as a group, then reuse it in new projects.

While working on a project, you can import records from a group into the project. You can create multiple groups for different types of schedule entries or different manufacturers. The organization of the groups makes it easier to find the record you want to import.

Standards Databases

Multiple standards databases can be created. Each standards database has its own set of customization settings.

When a project is created, the customization settings are copied from a selected standards database. Everything from the standards database is copied to the project and used as the default settings.

The standards database stores the following information:
- Cable tray groups
- Conduit groups
- Label group settings
- Layer systems
- Layer definitions
- Options settings
- Text styles

Common Schedule Dialog Box Features

All of the schedule dialog boxes work in a similar way. The buttons available that you use to create and modify records in the schedule are all the same.

New: Press this button to create a new record in the schedule. The record will be created with default information.

Copy: Press this button to copy the displayed record in the schedule. The new record will be created with the same information as the displayed record.

Save: Press this button to save any changes made to the displayed record in the schedule. If you make changes and do not press this button, you will be prompted to save your changes before selecting a new record or closing the schedule dialog box.

Delete: Press this button to delete the displayed record from the schedule. If there are any devices inserted on a drawing in the project that use the record, you will not be able to delete it. You must first delete all of the devices from the drawings before you can delete the record.

Rename: Press this button to rename the displayed record in the schedule. Any labels on the drawing will be updated.

Find: Press this button to select a schedule record from a list of all of the items in the schedule. The record you select will be displayed in the dialog box, allowing you to view and modify it.

For most schedules, a list of entries will be displayed.

For distribution equipment, the **Find Distribution Equipment** dialog box will be used. This dialog box works the same way as the **Edit Distribution Equipment** command.

Next: Press this button to display the next record in the schedule. The next record is determined by sorting the entries in the schedule alphabetically. If you press this button when viewing the last record, it will wrap back to the beginning and display the first record.

Previous: Press this button to display the previous record in the schedule. The previous record is determined by sorting the entries in the schedule alphabetically. If you press this button when viewing the first record, it will wrap to the end and display the last record.

Current Group: The active group. All of the items that are created and modified will be associated with this group.

Select Group: Press this button to open the **Groups** dialog box. In addition to allowing you to select a new group, the **Groups** dialog box also allows you to create and modify groups. See the **Schedule Groups** section for more information.

Move to Another Group: Press this button to move the displayed record to another group. A dialog box will prompt you to select the other group to which the item will be moved. The *Current Group* will be changed to the selected group. This button is disabled if only one group exists.

Copy to Another Group: Press this button to copy the displayed record to another group. A dialog box will prompt you to select the other group to which the item will be copied. A copy of the displayed record will be created in the other group. This button is disabled if only one group exists.

Import from Another Database

The **Import from Another Database** button is used to import records from a schedule in another project to the current project. This button can be used to copy an entire schedule or a single record. This button allows you to reuse schedules in new projects.

When you press this button, the **Select Database to Import From** dialog box will appear and prompt you to select the project from which the schedule records will be imported.

Select the database from which to import. You must select a Design Master Electrical database. The database must be updated to the most current version. If it is not, you will be prompted to open a drawing in the project to convert the database. After it is updated, you will then be able to import from it.

After you select the database, the **Import Record from Another Project** dialog box will appear and prompt you to select the records to import.
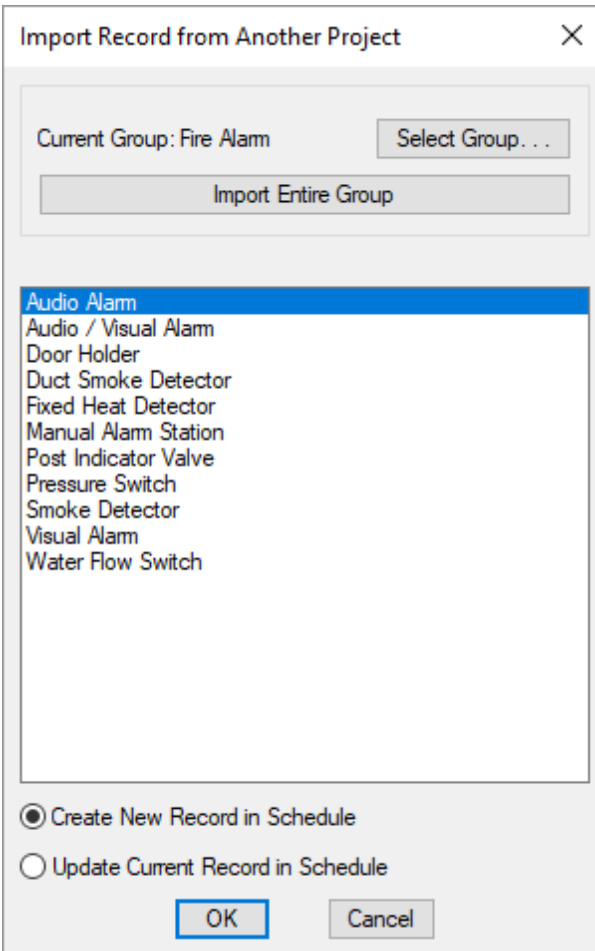
Current Group: The group in the other project schedule from which the records will be imported.

Select Group: Press this button to select a new group from which to import records.

Import Entire Group: Press this button to import all of the records from the *Current Group* to the project.

New records will be created in the project for each record being imported. If a record with the same name already exists in the current schedule, a new record will be created with **(Copy)** appended to the record name. The existing record will not be modified.

You will be prompted if you want to delete existing records in the project schedule. If you delete the existing records, any records that have devices associated with them will not be deleted.

Record List: The list of records in the *Current Group*. Select the records to import from this list. You can select multiple records using the SHIFT and CTRL keys.

Create New Record in Schedule: Select this option to create a new record in the project schedule for each record being imported. The record will be created with the same name as the record in the other project. If a record with the same name already exists in the current schedule, a new record will be created with **(Copy)** appended to the record name. The existing record will not be modified.

Update Current Record in Schedule: Select this option to update the current record in the project schedule with

599

the information from the imported record. All of the information in the project schedule will be overwritten.

Import from Master Schedule

The **Import from Master Schedule** button is used to import records from the master schedule to the project schedule after the project has been created. This button allows you to import an entire group or individual records.

When you press this button, the **Import Master Record** dialog box will appear.
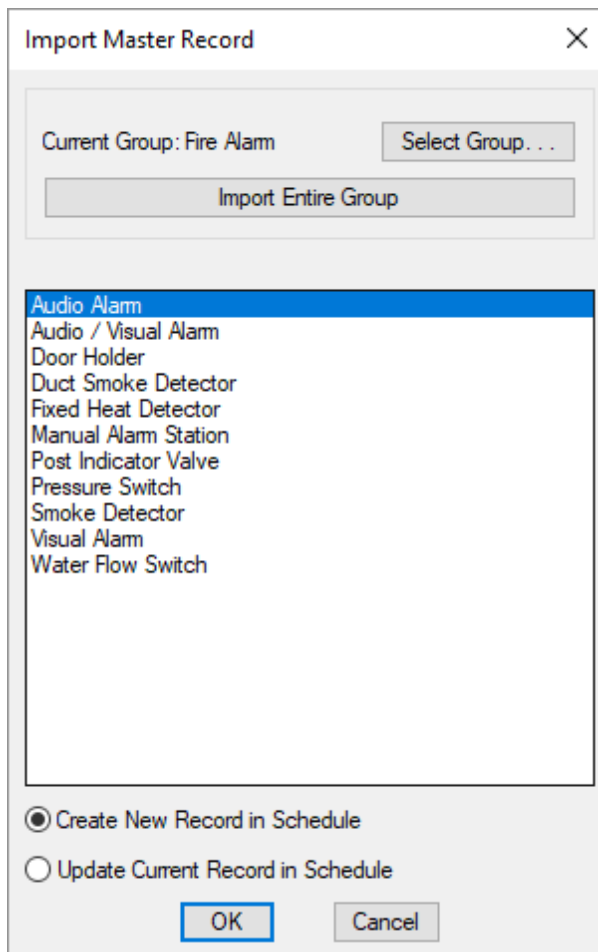
Current Group: The group in the master schedule from which the records will be imported.

Select Group: Press this button to select a new group from which to import records.

Import Entire Group: Press this button to import all of the records from the *Current Group* to the project.

New records will be created in the project for each record being imported. If a record with the same name already exists in the current database, a new record will be created with **(Copy)** appended to the record name. The existing record will not be modified.

You will be prompted if you want to delete existing records in the project schedule. If you delete the existing records, any records that have devices associated with them will not be deleted.

Record List: The list of records in the *Current Group*. Select the record to import from this list. You can select multiple records using the **SHIFT** and **CTRL** keys.

Create New Record in Schedule: Select this option to create a new record in the project schedule for each record being imported. The record will be created with the same name as the record in the master schedule. If a record with the same name already exists in the current database, a new record will be created with **(Copy)** appended to the record name. The existing record will not be modified.

Update Current Record in Schedule: Select this option to update the current record in the project schedule with the information from the imported record. All of the information in the project schedule will be overwritten.

Import from Project Schedule

The **Import from Project Schedule** button is used to import records from a project schedule to the master schedule. This button can be used to copy an entire project schedule or a single record. This button allows you to create your master schedule from a project schedule.

When you press this button, the **Select Database to Import From** dialog box will appear and prompt you to select the project from which the schedule records will be imported.

Select the database from which to import. You must select a Design Master Electrical database. The database must be updated to the most current version. If it is not, you will be prompted to open a drawing in the project to convert the database. After it is updated, you will then be able to import from it.

After you select the database, the **Import Record from Another Project** dialog box will appear and prompt you to

select the records to import.
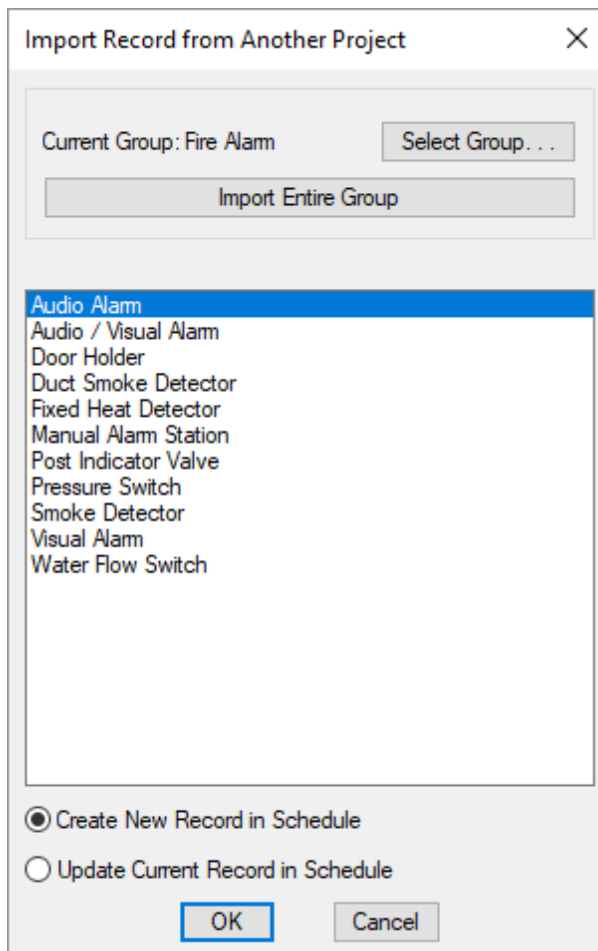
Current Group: The group in the project schedule from which the records will be imported.

Select Group: Press this button to select a new group from which to import records.

Import Entire Group: Press this button to import all of the records from the *Current Group* to the master schedule.

New records will be created in the master schedule for each record being imported. If a record with the same name already exists in the master schedule, a new record will be created with **(Copy)** appended to the record name. The existing record will not be modified.

You will be prompted if you want to delete existing records in the master schedule.

Record List: The list of records in the *Current Group*. Select the records to import from this list. You can select multiple records using the `SHIFT` and `CTRL` keys.

Create New Record in Schedule: Select this option to create a new record in the master schedule for each record being imported. The record will be created with the same name as the record in the project schedule. If a record with the same name already exists in the master schedule, a new record will be created with **(Copy)** appended to the record name. The existing record will not be modified.
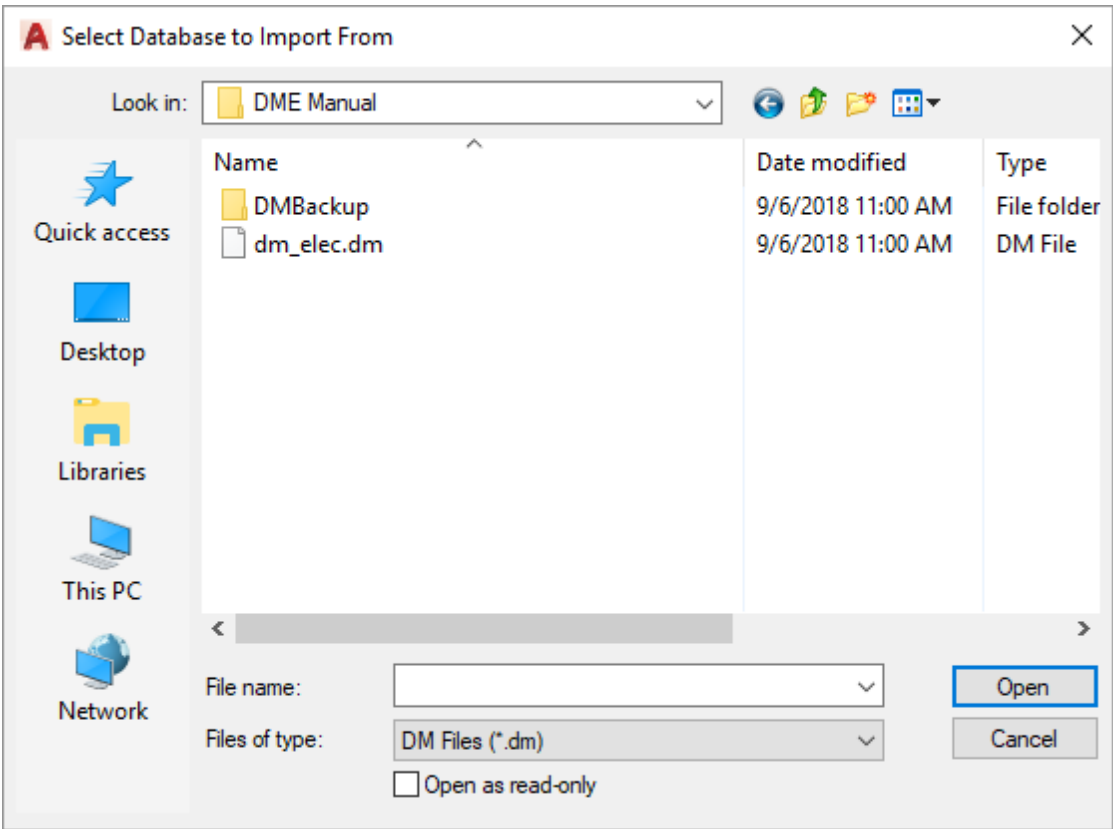
Update Current Record in Schedule: Select this option to update the current record in the master schedule with the information from the imported record. All of the information in the master schedule will be overwritten.

Common Device Settings

This section describes settings that are common to multiple types of devices.

Elevation

All devices can be assigned an elevation. The 2D and 3D blocks for the device will be inserted at the specified elevation. For 3D blocks, whether this elevation is the top, center, or bottom elevation is based upon the block.

If the elevation is not set, a default value of **0** will be used. This will not create any errors, but some features will not be useful if you do not provide elevations. The 3D model will not be correct, and distances between devices will not account for differences in elevation.

Relative to alignment point area: The elevation is relative to the alignment point area for the device. The elevation for the device is added to the elevation of the alignment point to give the true elevation of the device.

Relative to ground: This option applies only to panels. The elevation is relative to the ground. Typically, this option is used when the **Generate Riser** command will be performed before distribution equipment is inserted on a drawing.

Elevations are specified as **feet-inches**. For example, an elevation of 5 feet 6 inches should be entered as "5-6". A single number without a dash is interpreted as an elevation in feet.

Layer System

All devices that can be inserted on a drawing are assigned a layer system. The layer system controls the layers that are used when the device is inserted.

Layer systems can be created using the **Layers** command. See the **Layers** section for more information.

To change the layer of a device, change the layer system. Do not change the layer directly in your CAD program; if you do, it will likely be changed back at some point in the future.

Schedule Notes

Schedule records have three *Schedule Note* fields.

These notes can be used as a generic note field that is displayed in the schedule on the drawing.

They can also be used to include information for which the software does not include a specific location. To use the notes this way, designating one of the note fields for this purpose is recommended. In the column labels, you can change the label on the column for that note. When the schedule is printed, you will have a new field that includes the custom information.

Load Information

Loads on devices are defined by the following fields.

Panel Description: The description shown on the panel schedule for the load. Each receptacle and low voltage

device load can have a different description.

This field is not used for equipment loads. All of the equipment loads use the single *Panel Description* field for the equipment.

Load Type: The type of load. The calculated load for each load type is handled differently.
- **Lighting:** The calculated load is fixed at 125% of the connected load.
- **Single Motor:** The calculated load is fixed at 125% for the largest single motor. The calculated load of all of the other single motor loads is fixed at 100%.
- **Multiple Motors:** The calculated load is fixed at 100% for multiple motors. If there are no single motor loads, the largest multiple motors load is treated as the largest single motor load and the calculated load for that motor is fixed at 125%.
- **Receptacle:** The calculated load is fixed at 100% of the first 10 kVA and 50% of the load after that.
- **Continuous:** The calculated load is fixed at 125% of the connected load.
- **Heating:** The calculated load is fixed at 125% of the connected load.
- **Cooling, Single Motor:** The calculated load is fixed at 125% for the largest single motor. The calculated load of all of the other single motor loads is fixed at 100%.
- **Cooling, Multiple Motors:** The calculated load is fixed at 100% for multiple motors. If there are no single motor loads, the largest multiple motors load is treated as the largest single motor load and the calculated load for that motor is fixed at 125%.
- **Heating and Cooling Motor:** The calculated load is fixed at 125% of the connected load.
- **Noncontinuous:** The calculated load is fixed at 100% of the connected load.
- **Kitchen:** The calculated load is based upon the number of pieces of kitchen equipment, using *NEC Table 220.56*. The number of devices represented by the load is entered in the *Demand Factor* field.
- **Diverse:** The calculated load is based upon the specified demand factor. The *Demand Factor* value is a percentage applied to the connected load (e.g., enter "0.8" for 80% of the connected load).
- **Custom 1:** The calculated load is based upon the *Custom Load 1: Calculation Method* option. See the **Circuits, Load Types, and Wire Sizes** options section for more information.
- **Custom 2:** The calculated load is based upon the *Custom Load 2: Calculation Method* option. See the **Circuits, Load Types, and Wire Sizes** options section for more information.
- **Small Appliance:** The calculated load is fixed at 125% of the connected load.
- **Laundry:** The calculated load is fixed at 125% of the connected load.
- **Appliance, Continuous:** The calculated load is based upon the number of appliances, using *NEC Table 220.53*. The number of devices represented by the load is entered in the *Demand Factor* field.
- **Appliance, Noncontinuous:** The calculated load is based upon the number of appliances, using *NEC Table 220.53*. The number of devices represented by the load is entered in the *Demand Factor* field.
- **Appliance Motor:** The calculated load is fixed at 125% of the connected load.
- **Electric Dryer:** The calculated load is based upon the number of appliances, using *NEC Table 220.54*. The number of devices represented by the load is entered in the *Demand Factor* field.
- **Electric Cooking:** The calculated load is based upon the number of appliances, using *NEC Table 220.55*. The number of devices represented by the load is entered in the *Demand Factor* field.
- **Dwelling Unit Heating:** The calculated load is based upon the number of heating units, using *NEC 220.82 (C)*. The number of devices represented by the load is entered in the *Demand Factor* field.
- **Dwelling Unit Cooling, Single Motor:** The calculated load is fixed at 125% for the largest single motor. The calculated load of all of the other single motor loads is fixed at 100%.
- **Dwelling Unit Cooling, Multiple Motors:** The calculated load is fixed at 100% for multiple motors. If there are no single motor loads, the largest multiple motors load is treated as the largest single motor load and the calculated load for that motor is fixed at 125%.
- **Dwelling Unit Heating and Cooling Motor:** The calculated load is fixed at 125% of the connected load.
- **Marina / Mobile Home / RV:** The calculated load is based upon the number of marina receptacles, mobile home lots, or RV sites. The number of units represented by the load is entered in the *Demand Factor* field.

Horsepower: The horsepower of the motor load. This field only exists for equipment. The *Connected* field will be set based upon the horsepower using *NEC Table 430.248* for single-phase motors and *NEC Table 430.250* for

three-phase motors.

Connected (kVA): The connected load in kVA.

Demand (kVA): The connected load after the demand factor has been calculated. For most load types, this field will be disabled and filled in automatically based upon the connected load. It will be enabled for **Custom 1** and **Custom 2** if the *Calculation Method* is set to **Variable Calculated Load**.

Demand Factor: The multiplier used to calculate the demand load from the connected load. For most load types, this field will be disabled and automatically filled in.

It will be enabled for **Kitchen**, **Diverse**, **Appliance, Continuous**, **Appliance, Noncontinuous**, **Electric Dryer**, **Electric Cooking**, **Dwelling Unit Heating**, and **Marina / Mobile Home / RV**. The value is the number of units of that load type. It will also be enabled for **Custom 1** and **Custom 2** if the *Calculation Method* is set to **Variable Demand Factor**.

Distance Input Format

Distances are specified as **feet-inches**. For example, a distance of 5 feet 6 inches should be entered as "5-6". A single number without a dash is interpreted as a distance in feet or inches as denoted by the item label.

Schedule Groups

All schedules in the project and master database can be organized into groups. Groups allow you to organize the records in your schedule.

Each group is a separate set of types within each device type. Each group has its own schedule. When you insert a device, you first select the group, then select the type of device.

The **Select Group** button on project and master schedule dialog boxes opens the **Schedule Groups** dialog box. This dialog box is used to create and modify schedule groups. The specific labels will vary based upon the type of schedule you are modifying, but the functionality is the same for all types.

When you create a project, all of the records from a group can be copied to the project. Copying a group is helpful when you have a common type of project with similar schedules. You can set up the schedule once in the master as a group, then reuse it in new projects.

While working on a project, you can import records from a group into the project. You can create multiple groups for different types of schedule entries or different manufacturers. The organization of the groups makes it easier to find the record you want to import.

In a project, groups are used to organize the devices. Each group has a separate schedule that can be inserted on the drawing. The layout of each schedule can be different for each group.

Training Videos

- **Master Schedule Groups and Labels**

<Schedule Type> <Project / Master> Schedule Groups Dialog Box
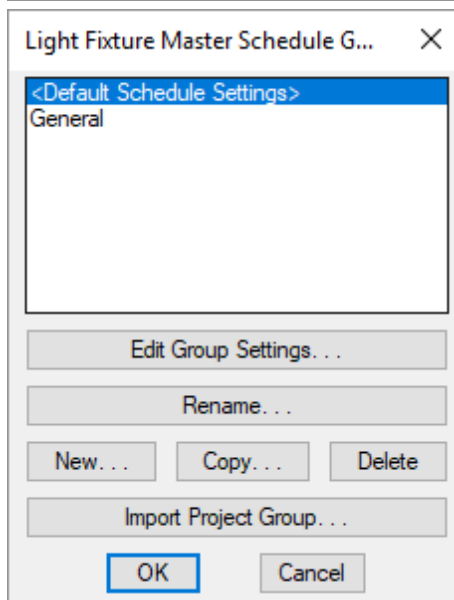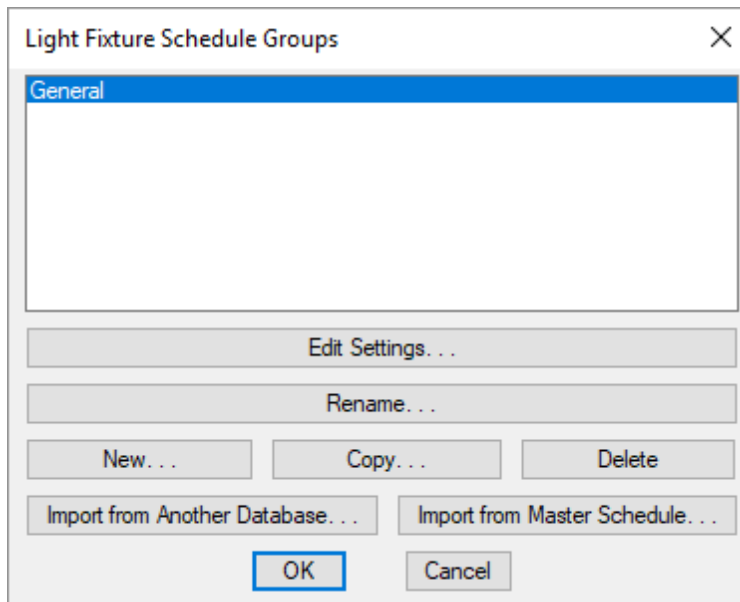
Delete: Press this button to delete the selected group. You will be asked to confirm whether you want to delete the group. If the group has schedule entries inserted on the drawing, those entries will not be deleted.

Import from Another Database: See the **Import from Another Database** section for more information about this button.

Import from Master Schedule: See the **Import from Master Schedule** section for more information about this button.

Import Project Group: See the **Import from Project Schedule** section for more information about this button.

Press the **OK** button to return to the previous dialog box with the selected group active.

Inserting Notes and Leaders on the Drawing

All notes are inserted using a common user interface. You will be prompted to specify the location of the note on the drawing by dragging the text to the proper location.

When locating the note, a leader will automatically be inserted if the note is located far enough away from the item. When the note is too close the the item, the leader line will be removed.

You can toggle the leader on and off by right-clicking or pressing **SPACE**.

Related Options

You can customize the block used for the leader arrow and its length. See the **Leaders Options** section for more information.

Calculation Methods and Formulas

This section describes some of the calculation methods used by Design Master Electrical.

Fault Calculation Method

The fault calculations are based upon *IEEE Std 242-2001: IEEE Recommended Practice for Protection and Coordination of Industrial and Commercial Power Systems*.

The fault that is calculated is the three-phase line-to-line fault.

Fault = Voltage / $\sqrt{(Resistance^2 + Reactance^2)}$

The fault calculation includes motor contributions for all devices with a load type of **Single Motor** or **Multiple Motors**. The motor current is equal to 4 times the rated current of the motor. The X/R ratio of motors is 4. The impedance of the branch circuit wires from the motor to the panel is not included. These values cannot be modified by the user.

The feeder impedance values are based upon *NEC Table 9*. These values can be modified by the user.

The transformer impedance values are for dry-type indoor transformers. These values can be modified by the user.

Voltage Drop Calculation Method

*Three-Phase % Voltage Drop = Load in Amps * Effective Z * (Wire Length / 1,000) * 100 / Line-to-Neutral Voltage*

*Single-Phase % Voltage Drop = Load in Amps * Effective Z * (Wire Length / 1,000) * 100 * 2 / Voltage*

Load in Amps is different for feeders and branch circuits. For feeders, it is the calculated load on the panel assuming a balanced load. For branch circuits, it is based upon the connected load.

*Effective Z = R * Power Factor + X* sin(arccos(*Power Factor*)) (See *NEC Table 9 Note 2*)

R = *Resistance* set in the **Wire Sizing** command.

X = *Reactance* set in the **Wire Sizing** command.

Power Factor = 0.85

Line-to-Neutral Voltage is used for three-phase calculations based upon *NEC Table 9 Note 2*: "Multiplying current by effective impedance gives a good approximation for line-to-neutral voltage drop."

Voltage is the line-to-neutral voltage for single-pole circuits and line-to-line voltage for two-pole circuits.

The feeder impedance values are based upon *NEC Table 9*. These values can be modified by the user.

The transformer impedance values are for dry-type indoor transformers. These values can be modified by the user.

Feeder Length Calculation

The feeder length between distribution equipment can either be input by the user or calculated based upon the location of the distribution equipment on the drawing.

Feeder length can be calculated using two methods:
- **Right Angles:** Length is based upon the distance using right angles between the two devices. The angle can be offset by the user with the *Building Angle* field.
- **Straight Line:** Length is based upon the distance between the two devices along a straight line.

When the length is calculated, the difference in elevation is included based upon the alignment point area elevation or floor elevation. An additional length is added to account for the wire needed to make the connection to the distribution equipment. The user can specify this value using the *Feeder make-up* option. See the **Distribution Equipment** options section for more information about how to set this value.

Wire Sizing Calculation Method

The wire size for a given ampacity is set in the **Wire Sizing** dialog box. See the **Wire Sizing** section for more information about setting these values.

The default wire sizes are based upon *NEC Table 310.16*. The *60°C Copper* column is used for wires up to #1. The *75°C Copper* column is used for wires 1/0 and larger. These wire sizes are used when the wire size is chosen automatically.

The *Aluminum or Copper-Clad Aluminum* column sizes are included and can be chosen manually by the user. They are not used for sizing the wires automatically.

Arc-Flash Overview

This section provides an overview of how arc-flash calculations work.

The goal of arc-flash calculations is to calculate the **incident energy** at a specific distance from the equipment during an arc-flash event. The calculated incident energy is used to determine how to keep workers safe while working on the equipment live.

The arc-flash protection boundary is the distance unprotected workers must keep between themselves and the

equipment. This distance will limit the injury for workers to second-degree burns.

There are three features you will use when performing arc-flash calculations:

The **Arc-Flash Settings** dialog box is used to specify additional information about panels required to perform arc-flash calculations. Most of the work related to arc-flash is done using this command.

The **Select Curve** dialog box is used to specify the time-current curve for the device's main disconnect and/or upstream overcurrent protection. The curve affects the *Arcing Time* section of the **Arc-Flash Settings** dialog box.

The **Insert Arc-Flash Schedule** command will insert a schedule on the drawing documenting the arc-flash calculations.

The **Print Arc-Flash Stickers** command will create stickers for each of your panels and display them in your web browser. You can then print them using a color printer and sticker paper and attach them to your panels in the field.

Exempt Light Fixtures

Light fixtures can be set as exempt in the light fixture schedule and on individual fixtures.

Exempt light fixtures will have the *Exempt light fixture block* inserted next to them on the drawing to identify them as exempt. The exempt block is inserted on the **LIGHT FIXTURES-GROUP CALLOUT** layer key. See the **Light Fixtures** options section for more information about setting the block.

This setting is only used when calculating light fixture quantities using the Excel Add-in. It is not used in any other calculations.

Addressing Distribution Equipment Calculation Errors

If there are errors in the distribution equipment definitions, an alert dialog box will be displayed alerting you to the errors. The specific errors and their solutions will be listed on the command line. Press **F2** to expand the command line to see all of the errors.

An "X" will be drawn over the schedule if any errors occur. Once the errors are fixed and the schedule is updated, the "X" will disappear.

Overloaded Distribution Equipment

When running the **Update Distribution Equipment Schedule** or **Update All Schedules and One-Line Diagram** command, if any of the affected schedules show an overloaded distribution equipment, the **Overloaded Distribution Equipment** dialog box will appear.

Overloaded Distribution Equipment Dialog Box

All of the overloaded distribution equipment will be displayed in the list, along with the equipment size and the load on the equipment.

Edit: Press this button to open the **Distribution Equipment Dialog Box** for the selected distribution equipment.

Overloaded Threshold: The percentage of the load capacity at which the distribution equipment will be considered overloaded. Press the **Set Threshold** button to modify the threshold value.

Ignore Overload on Selected Distribution Equipment: Press this button to remove the selected distribution equipment from the list and stop receiving warnings when it is overloaded.

View Ignored Distribution Equipment: Press this button to view distribution equipment for which overload warnings are being ignored. The **Ignored Overloaded Distribution Equipment** dialog box will appear.

```
Ignored Overloaded Distribution Equipment                           ×

The following distribution equipment is overloaded but ignored.

EQUIPMENT NAME      SIZE      LOAD
L2                  100 A     97.15 A




















                            Edit. . .

        Warn on Overload of Selected Distribution Equipment

                             OK
```

Edit: Press this button to open the **Distribution Equipment Dialog Box** for the selected distribution equipment.

Warn on Overload of Selected Distribution Equipment: Press this button to return the selected distribution equipment to the list of overloaded distribution equipment and receive warnings when it is overloaded.

Ignore Overload on This Project: Press this button to change the *Warn if distribution equipment is overloaded* setting in the **Options** command.

Ignore Overload on All Projects: Press this button to change the *Warn if distribution equipment is overloaded* setting in the **User Options** command.

Finish Command: Press this button to close the dialog box and continue running the command.

Related Options

Warn if distribution equipment is overloaded (User Options): Sets whether this dialog box appears in projects for the current user.

Warn if distribution equipment is overloaded (Options): Sets whether this dialog box appears in the current project for all users.

Distribution equipment overloaded threshold (% of panel capacity): Sets the *Overload Threshold*.

Technical Support

Technical support is available by phone, email, or live chat on our website. We are available Monday through Friday, 9am to 5pm Eastern time.

Limited support by email is available evenings, weekends, and holidays.

Email: support@designmaster.biz

Phone: 1-866-516-9497 x2

Live Chat: http://www.designmaster.biz/chat/

System Requirements

To run Design Master Electrical, you need AutoCAD 2007 or later or BricsCAD within the last 3 years, and a computer capable of running your specific version of AutoCAD or BricsCAD.

Complete system requirements are available on our website.

Purchasing

[Pricing information and purchase links are available on our website.](#)