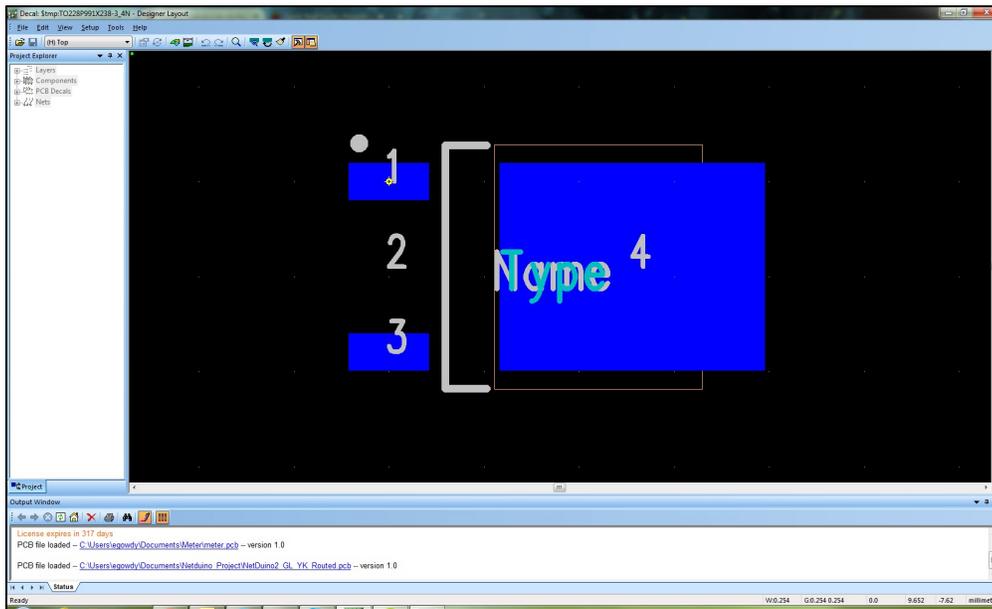


# Creating and Editing Decals with the Decal Editor

A PCB decal is a representation of the outline, terminals, and attributes of an electrical component and is used in placement and routing of the component during the design layout process.

You can use the PCB Decal Editor to create new decals or to modify existing ones. To open the Decal Editor, go to **Tools > PCB Decal Editor**.



## Contents

- 1 Decal Creation and Editing Basics
  - 1.1 Set the Origin
  - 1.2 Add and Place Terminals
    - 1.2.1 Using Step and Repeat to Add and Array of Terminals
  - 1.3 Create Copper Shapes
    - 1.3.1 Associating Copper with Terminals
  - 1.4 Define Terminal Pad Stacks
  - 1.5 Create Part Outlines
    - 1.5.1 Creating Assembly Drawing Decal Objects
    - 1.5.2 Creating Silkscreen Decal Objects
    - 1.5.3 Creating a Placement (Nudge) Decal Outline

## Decal Creation and Editing Basics

The operations listed below will help you to create a basic decal or edit a preexisting one.

### Set the Origin

The location of a decal's origin affects how decals attach to the cursor when you move a component in the Move by Origin mode. It also affects how pick and place machines populate the board with components. Use one of the procedures below to set the origin of the decal.

1. Go to **Setup > Set Origin**
2. Click the location in the decal window where you would like the origin to be placed.

or

1. Type the modeless command `SO <x> <y>` to set the origin according to relative coordinates, or `SOA <x> <y>` for absolute coordinates.

## Add and Place Terminals

Terminals are the pads or pins associated with a decal. The **Add Terminals** dialog box will allow you to easily create terminals for your decal:

1. Click the **Drafting Toolbar** icon to open the Drafting Toolbar.
2. Select the **Terminal** icon.
3. In the **Add Terminals** dialog box, in the **Start pin number** area, type values in the **Prefix** and/or **Suffix** boxes for the pin numbering. A preview of pin numbers based on your input is displayed below the boxes.
  - Alphabetic and numeric values can be used in either box. For example, A1 or 1A.
  - For a single numeric, use either Prefix or Suffix box, and void the other box.
4. In the **Increment options** area, choose what to increment by clicking either the **Increment prefix** or **Increment suffix** option.
5. In the **Step value** box, type a positive or negative number by which to increase or decrease the pin numbers with consecutive or stepped values.
6. If using alphanumerics, you can select the **Use JEDEC pin numbering** check box to ensure that legal alphanumeric values are used. This option only ensures that valid alpha and numeric combinations are used. To arrange rows and columns according to JEDEC, use the **Assign JEDEC Pinning** option on the **Tools** menu.
7. Click **OK**.
8. Click in the decal window to indicate a position for the new terminal. Repeat as necessary, then press **Esc** to exit place terminal mode.

You can also add terminals by replicating an existing terminal using **Step and Repeat**.

## Using Step and Repeat to Add and Array of Terminals

You can easily add an array of terminals using the **Step and Repeat** function.

1. Select an existing terminal, right-click, and select **Step and Repeat**.
2. In the Step and Repeat dialog box, click the tab of the type of array to create: **Linear**, **Polar**, or **Radial**, and set the options for the array.
3. Click **OK** to create the array.

Terminals are replicated with associated coppers.

## Create Copper Shapes

You can create copper in the decal and associate it with a pin to create a custom pad shape or heat sink. You can also create unassociated copper heat sink or shielding shapes that move with the decal in the design. To create a copper shape, follow this procedure:

1. Click the **Drafting Toolbar** icon.
2. Select the **Copper** icon.
3. Right-click to change values of the drafting object as needed.
4. Draw the desired shape. Once the shape is complete, it becomes a filled shape.

## Associating Copper with Terminals

You can associate a drawn copper shape with a terminal/pad. This assigns the copper shape the same net connection as the terminal/pad. You can also use associated copper to create custom pad shapes.

1. Draw the copper shape.
2. Select a terminal, right-click, then select **Associate**.
3. Select the copper item to associate with the terminal. This electrically connects the copper item with the terminal. You may want to physically connect the terminal and copper item by overlaying part or all of the terminal.
4. If you are using the associated copper to create an odd shaped pad, maximize the pad shape inside the overlapping associated-copper shape. The underlying pad is still the routing target and also the connection point for terminals if this pad will ever be connected to a copper pour or plane area. If associated copper surrounds the terminal pin by a large amount it could prevent thermal spokes from being generated.

### Note

Routing is only completed to the center of the pad. Any thermal connections are made to the pad only, not to the associated copper shape.

## Define Terminal Pad Stacks

You can customize the pad stacks of the pins in your decal. You can choose to customize one pin, or multiple pins simultaneously.

1. Go to **Setup > Pad Stacks**
2. In the **Pad Stack Properties** dialog box, ensure that your decal is selected in the **Decal** name list.
3. Under the **Pin: Plated:** list, select the pin to customize. If you do not want to customize all the pins simultaneously, you can add one or

more specific pin numbers to the list using the **Add Pin** dialog box.

4. Under the **Sh.:** **Sz.:** **Layer:** list, select the layer of the pad you want to customize.
5. In the **Parameters** area, specify the settings for all three **Pad**, **Terminal**, and **Antipad** pad styles if needed. The style of pad used is selected automatically in the design depending on the situation. The thermal and antipad styles are used when the pin is located within a plane.
6. Below the Parameters area, set the options that apply to the pads on all layers for the selected pin(s).
7. If you need to customize other pins in the decal, return to step 3.
8. Click **OK** to save all pad stack customizations.

## Create Part Outlines

If any of these outlines do not appear after you create them, the color set for the item is probably the same as the background color. Set a different color in the **Display Colors Setup** dialog box (**Setup > Display Colors**).

## Creating Assembly Drawing Decal Objects

The assembly outline and assembly refdes are used on the assembly drawing to define the exact outline and identify the body of the part. It shows where the part is to be placed during the board assembly process.

1. On the Assembly Drawing Top layer, draw the outline by selecting the **Drafting Toolbar** icon, clicking the **2D Line** button, and drawing the desired shape.
2. Add a second reference designator label on the Assembly Outline Top layer. The reference designator label should be large and located inside the outline.
  - a. Click the **Add New Label** button.
  - b. In the **Add New Label** dialog box, in the **Attribute** list, select the Ref.Des. attribute.
  - c. In the **Layer** list, select the Assembly Drawing Top layer.
  - d. Click **OK**. The label appears with the value "Name."
  - e. Right-click and select **Move**.
  - f. Position the new label.

## Creating Silkscreen Decal Objects

The silkscreen outline is printed on the board, and remains visible on the board after the part has been placed. This outline, along with its reference designator, pin number and polarity labels, identifies the part and its pins to persons assembling, testing, or servicing the board. The silkscreen also serves as a nudge outline if no actual nudge outline exists on Layer 20.

1. On the Silkscreen Top layer, draw the outline by selecting the **Drafting Toolbar** icon, clicking the **2D Line** icon, and drawing the desired shape.
2. Move the silkscreen refdes to the Silkscreen Top layer. The silkscreen refdes label is automatically added at the creation of the new decal. But by default it is added to the Top (Mounted) layer.
  - a. Double-click the "Name" label in the decal.
  - b. In the Decal Label Properties dialog box, in the Layer list, select the Silkscreen Top layer.
  - c. Click **OK**.
3. Add pin number and polarity labels if needed. The labels should be located *outside* the outline, where they can be seen after the part is placed.
4. Silkscreens for large components with many pins often have miniature dots or pin numbers visible next to selected pins to make it easier locate a pin on the board during testing or troubleshooting sessions. If you use text in the decal for these, they will not be movable when the component is placed in the design. Use labels instead; labels can be moved or deleted in the design if necessary.
  - a. Click the **Add New Label** button.
  - b. In the **Add New Decal Label** dialog box, in the **Attribute** box, type a name for the new attribute. The value can only be assigned once the decal is used in the design.
  - c. In the **Layer** list, select the Silkscreen Top layer.
  - d. Click **OK**. The label appears with the attribute name you typed.
  - e. Right-click and select **Move**.
  - f. Position the new placeholder.

## Creating a Placement (Nudge) Decal Outline

Some parts need additional spacing compared to others to allow for machine-placement on the physical board. The placement (or nudge) outline is utilized to define these spacings.

You can use a larger outline on layer 20 to ensure that your design placement is correct even though you apply the same body-to-body clearance for all components. On layer 20, draw the 2D line item(s) defining the nudge outline by selecting the **Drafting Toolbar** icon to display the Drafting Toolbar, then click the **2D Line** icon.