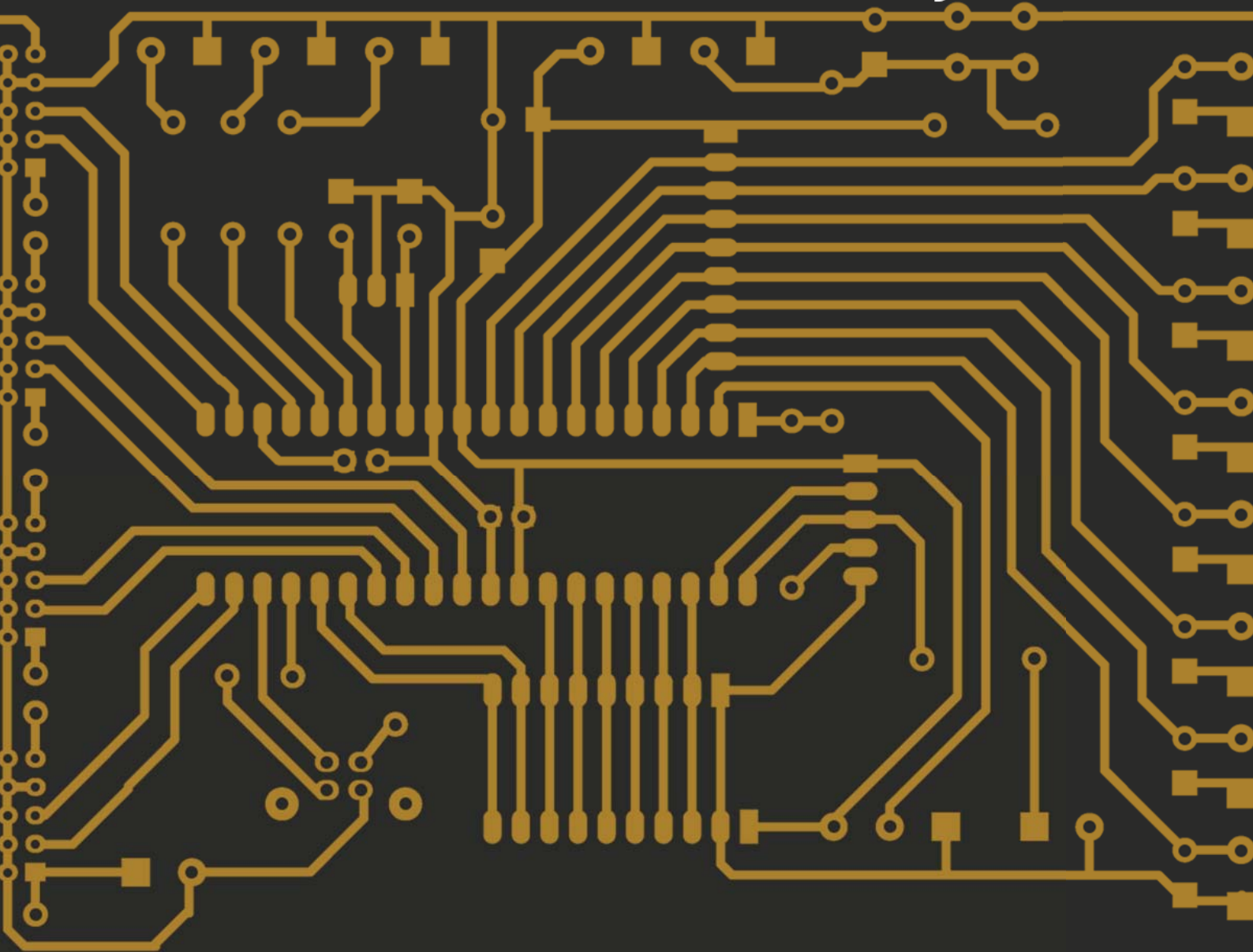




5 Ways to Customize Altium Designer for Better Efficiency



Christopher E. Carlson

Senior Field Applications Engineer

5 WAYS TO CUSTOMIZE ALTIUM DESIGNER FOR BETTER EFFICIENCY

Menu items, shortcut keys, and toolbar icons are the three ways of accessing features within the Altium Designer® environment. All of these are customizable and may enhance the user experience with Altium Designer. Any features that you feel are lacking in Altium Designer can be written with custom scripts and assigned to newly created menu items, shortcut keys, and toolbar icons. The entire Altium Designer API is accessible through scripting and new features can be added to the editors.

INTRODUCTION

Once you have been using Altium Designer for a while, you'll find the toolbar menu structure to be very intuitive, however, you may realize even more productivity gains with a bit of customization of the user interface. Customization is centric to each editor whether it's the PCB editor, schematic editor or any other editor within the Altium Designer environment. Whichever editor you are working in, the same approach may be taken in any of the other platform editors. It's a *LOT* easier that you might think.

In this paper, we will focus on customization of the PCB editor, however, the customization features discussed can be extended to all other editors within the Altium Designer environment.

FIVE WAYS TO CUSTOMIZE THE EDITORS

Customization of the Altium Designer editors is accomplished through the Customizing Editor which is specific to each of the editor domains. The specific editor to be customized must be the active open editor. The Customizing Editor is accessed from the top toolbar **DXP >> Customize**. Alternatively, you can double-click in the open area of the toolbars. See Figure 1.

The Customizing Editor dialog for each editor is specific to that editor and the title bar reflects which editor is being customized, in this case in Figure 2, the PCB editor. A new blank PCB document was first opened in order to bring focus to the PCB editor and then the Customizing Editor opened.

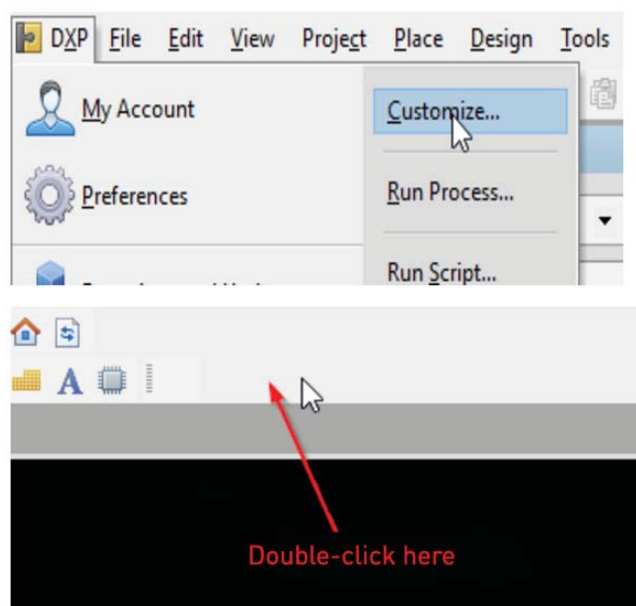


Figure 1: Access the Customizing Editor via the dropdown or double-clicking in the open area of the toolbars.

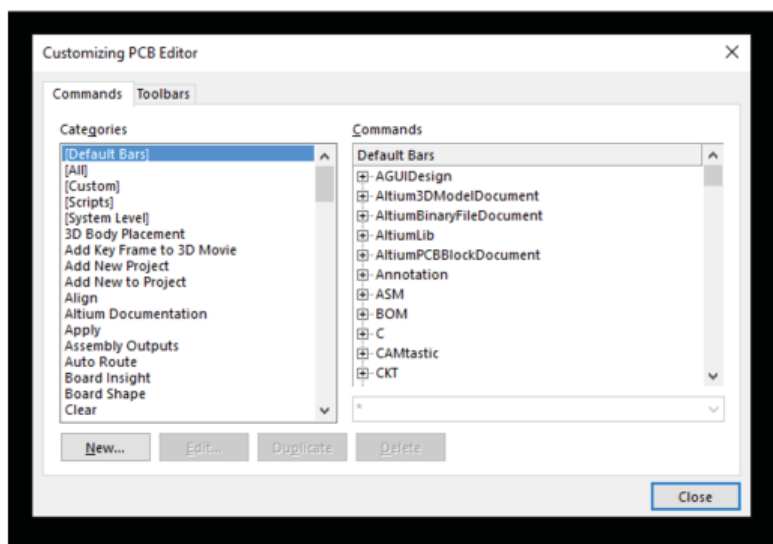


Figure 2: The Customizing Editor has choices specific to the editor being customized.

5 WAYS TO CUSTOMIZE ALTIUM DESIGNER FOR BETTER EFFICIENCY

1 — SHORTCUT KEYS

It is often useful to map the shortcut keys of the editor to features which may be more intuitive to the individual. This can also be useful when transitioning from another EDA tool to Altium Designer. The shortcut keys may be mapped to coincide with the familiar features of a former tool. Once the Customizing PCB Editor dialog is open a new toolbar icon appears titled **PCB Shortcuts** with a dropdown menu structure, as seen in Figure 3.

Each feature of the PCB editor which can be assigned to a shortcut key is available on this drop-down menu. To remap a specific shortcut key (or a combination of keys) to that feature, simply choose the feature of interest from the drop-down list by double clicking on the future name from the list. The **Edit Command** dialog will appear presenting all of the details of that editor command. The lower portion is where the shortcut key or keys are assigned, as shown in Figures 4 and 5.

If the chosen shortcut key is currently assigned to another command, that command will appear in the **Currently in use by** portion of the dialog and another shortcut key must be chosen or the previously mapped command must be remapped. Closing the Customizing PCB Editor dialog saves these changes.

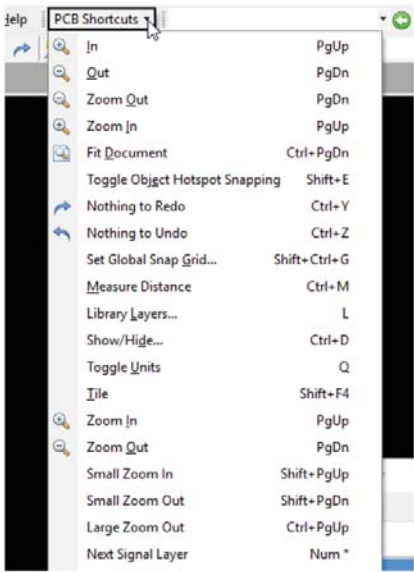


Figure 3: From the Customizing PCB Editor, click **PCB Shortcuts**.

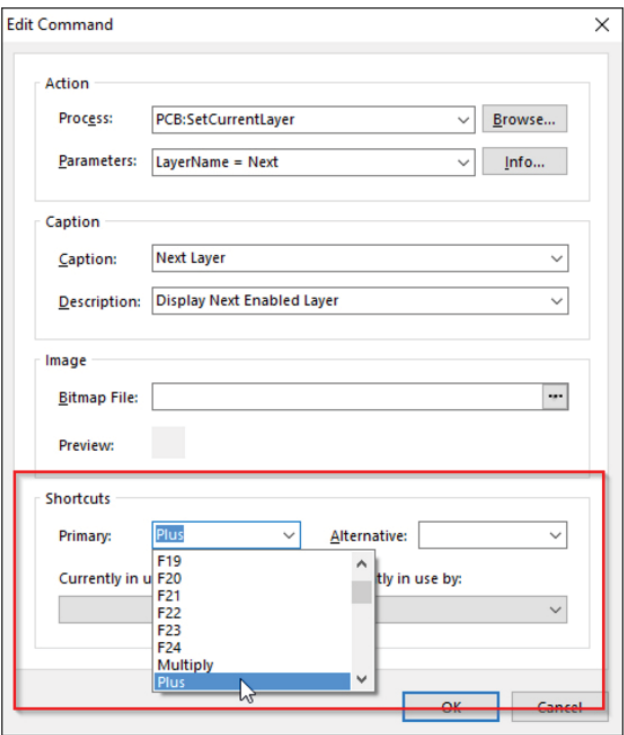


Figure 4: Assign shortcut keys using the dropdowns at the bottom of the dialog.

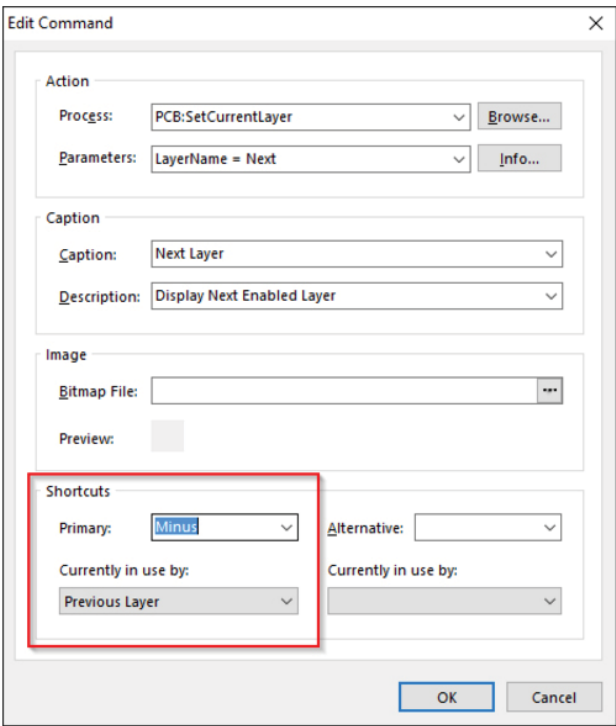


Figure 5: Once defined, click **OK**.

5 WAYS TO CUSTOMIZE ALTIUM DESIGNER FOR BETTER EFFICIENCY

2 — MENU BAR ITEMS

The menu bar items may also be customized in the Altium Designer environment as well. Opening the Customizing PCB Editor allows dragging and dropping menu bar items from one menu to another or removing them altogether.

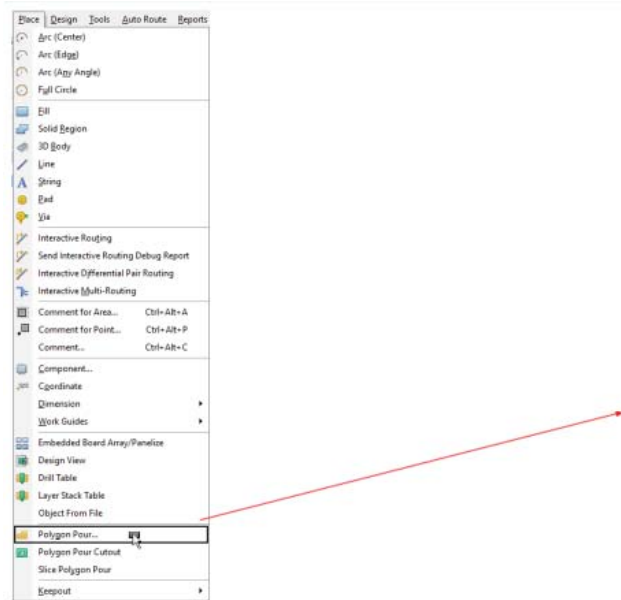


Figure 6: Items can be deleted by dragging off into space.

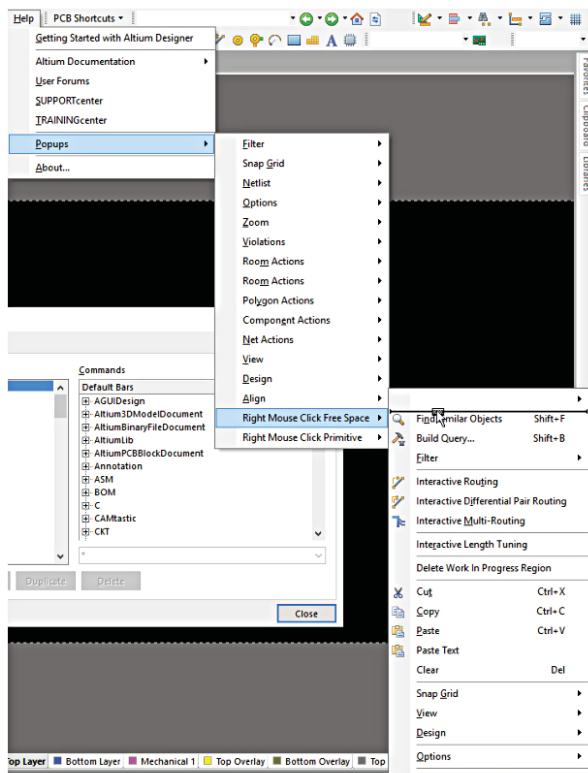


Figure 7: Any dropdown item can be clicked and dragged to a different dropdown menu.

To move a toolbar menu item from one location to another simply browse to that menu item to be moved, left click on the toolbar item and drag it to another toolbar.

To move a copy of the toolbar menu item, hold the **Ctrl** key while dragging the source menu item and to delete a menu item simply drag it off the menu into free space.

NOTE: If menu items are lost and recovery is needed to select the **Toolbars** tab in the Customizing PCB Editor, select the toolbar to be restored and click the **Restore** button, shown in Figure 8, when it becomes active. This is the process to restore any of the toolbars or shortcut keys to factory defaults.

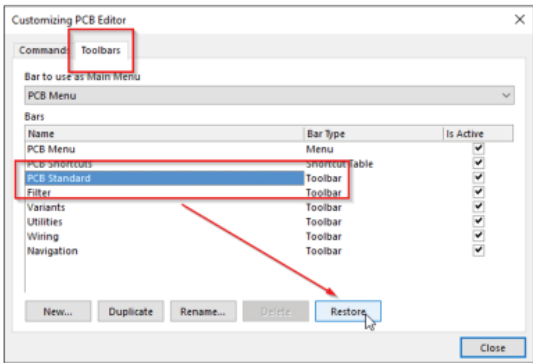


Figure 8: The **Restore** button reinstates factory settings.

5 WAYS TO CUSTOMIZE ALTIUM DESIGNER FOR BETTER EFFICIENCY

3 — TOOLBAR ICONS

The toolbar icons in the Altium Designer environment are a quick way of accessing commonly used features of the tool and may also be customized. The toolbar graphic, as well as the function evoked through the icon, is available for editing. In this section, we will create a new toolbar icon and use it to run a script to add previously unavailable functionality to the PCB editor.



Figure 9: Toolbar icons and graphics can be customized.

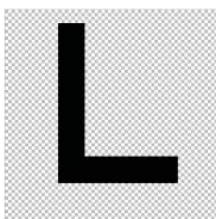
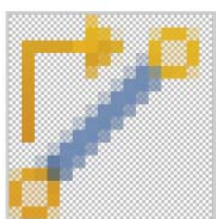


Figure 10: Examples of logos created for customizing the toolbar.

The graphics for the toolbar icons are located in the Altium Designer system directory under the subfolder **Buttons**, for example, `C:\Program Files (x86)\Altium\AD16\System\Buttons`. Each toolbar icon is represented by a 16 x 16 pixel *.png graphics file. The easiest way to create a new icon graphic is to pick one of the small icons from the **Buttons** directory, edit it in a graphics editor and save it with a new and meaningful name.

NOTE: If the graphic is larger than 16 x 16 pixels, the icon will not show up on the toolbar.

We will use a script used to create a PCB logo from a monochrome bitmap to demonstrate this capability. First, we must create a new icon from one of the existing available icons. In this case, the **Track.png** file was opened in a graphics editor and the graphics replaced with a captor letter **L** for **Logo Creator** and saved in the **Buttons** directory with the new name **Logo.png**.

There are a number of useful scripts available from the TechDocs page on the Altium website located here, click any of the download links to download the entire collection: <http://techdocs.altium.com/display/SCRT/Script+Examples+Reference#PCBScripts>

The target script is loaded into the Altium Designer preferences under **DXP >> Preferences >> Scripting System >> Global Projects**. In this case, the referenced project script is the **PCB Logo Creator**.

Click the **Install** button (Figure 11) and browse to the script of choice. Once the icon has been created and the script loaded into the Altium Designer platform we can now create a new toolbar command by opening the Customizing Editor from **DXP >> Customize....**

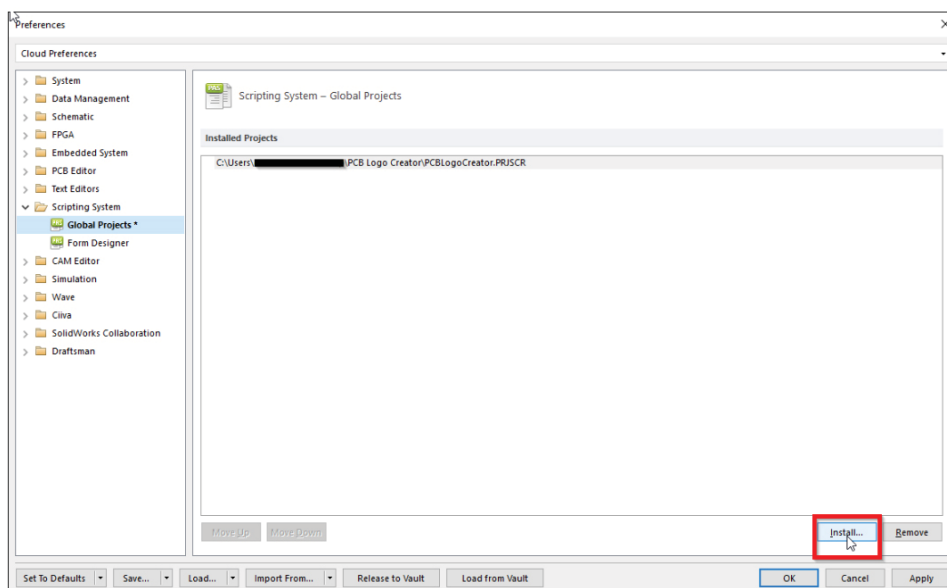


Figure 11: The Install button lets you browse for the script you wish.

5 WAYS TO CUSTOMIZE ALTIUM DESIGNER FOR BETTER EFFICIENCY

Locate **[Scripts]** under categories and drag the **RunConverterScript** option up to the top toolbar (Figure 12). This is just one of the script processes contained in the project we loaded, so these may be different in your loaded script project.

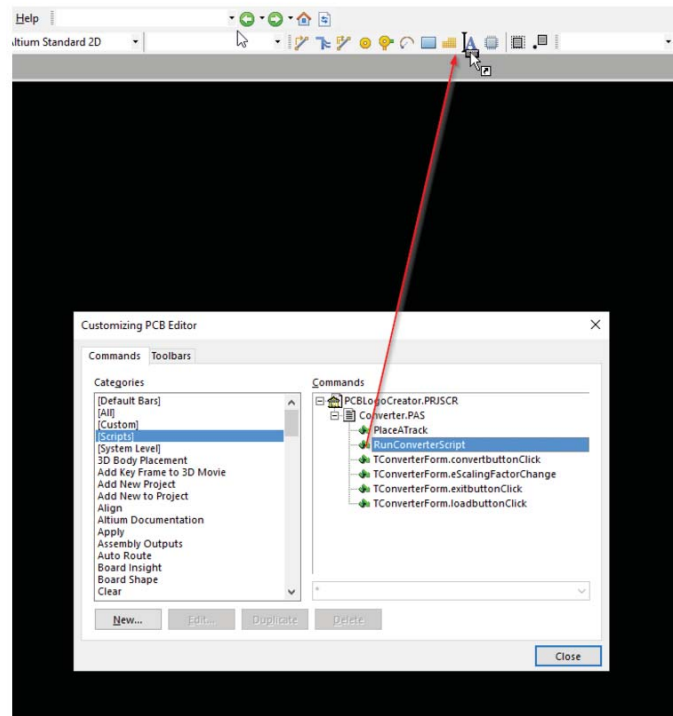


Figure 12: Grab the **RunConverterScript**.

As soon as you drop the command on the toolbar the **Edit Command** dialog will pop-up. The **Action** area of the dialog will automatically populate with the proper command process and parameters. Click the browse button next to the **Image >> Bitmap File** portion of the dialog and browse to the proper *.png file, in this case, **Logo.png**. At this point, a shortcut key may also be assigned to this command if desired.

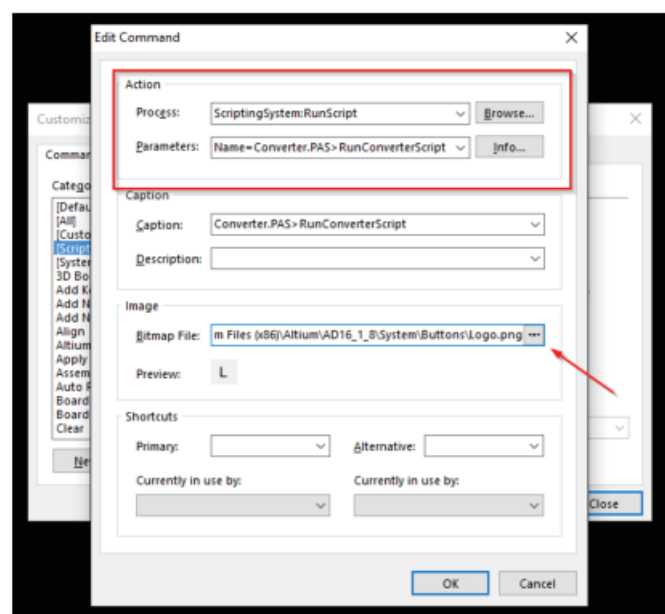


Figure 13: Find the image file you want and click **OK**.

5 WAYS TO CUSTOMIZE ALTIUM DESIGNER FOR BETTER EFFICIENCY

Now when you click the new icon the **PCB Logo Creator** script will run.

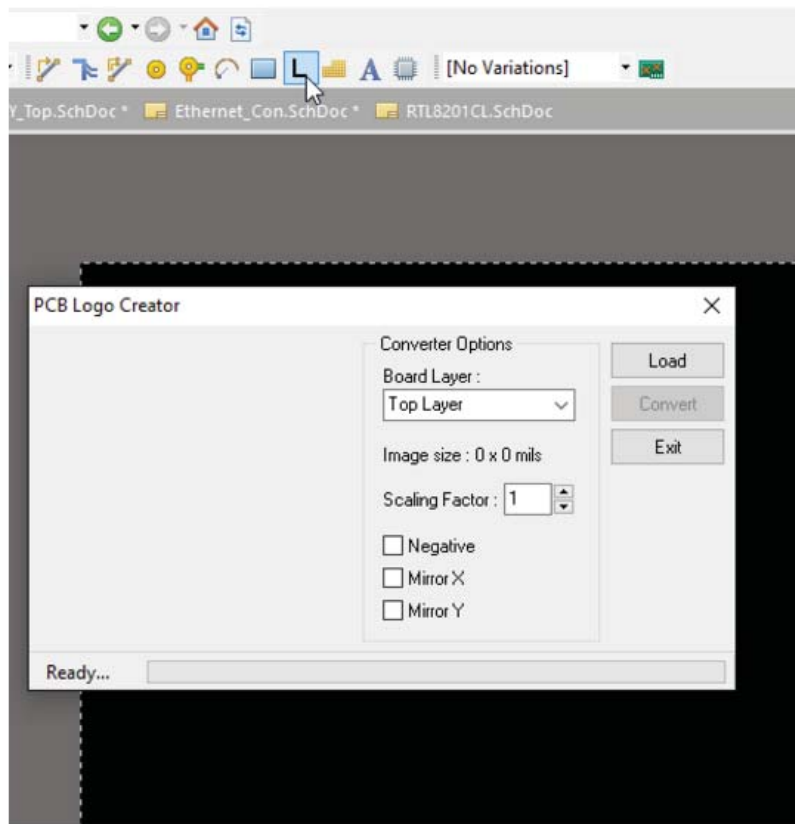


Figure 14: Run the script.

4 — DROPDOWN MENUS

Custom drop-down menus can also be added to the Altium Designer Environment. This is done by opening the Customizing Editor and right-mouse clicking on an existing toolbar and selecting **Insert Dropdown**.

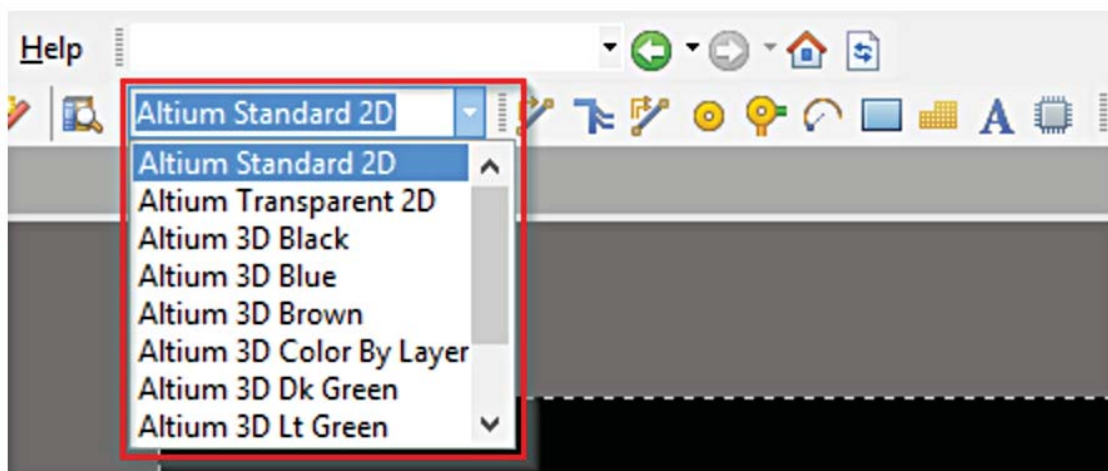


Figure 15: Dropdown menus can be customized as well.

5 WAYS TO CUSTOMIZE ALTIUM DESIGNER FOR BETTER EFFICIENCY

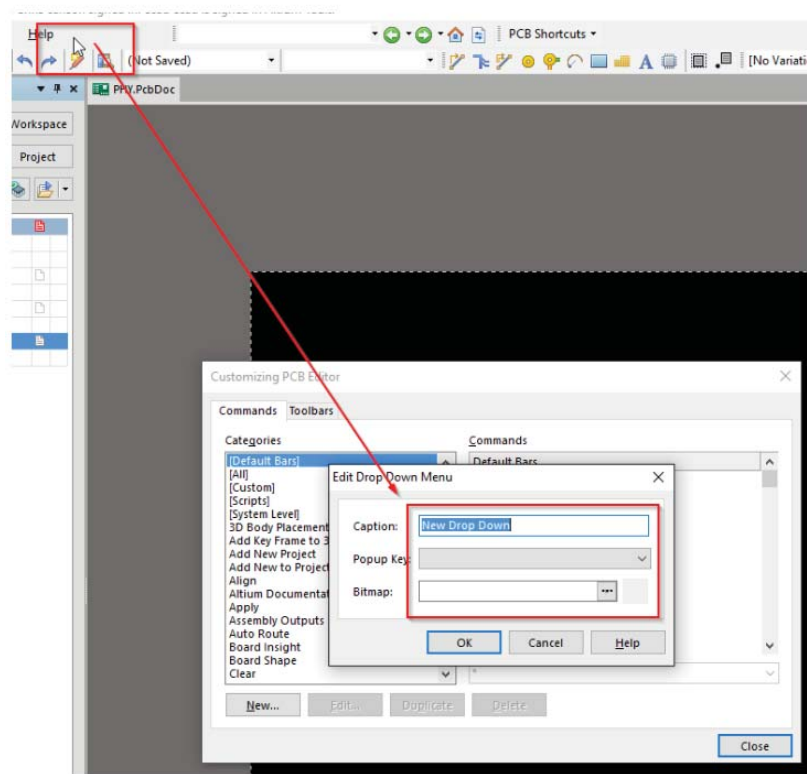


Figure 16: Right click on an open area of the menu bar to open the Customizing PCB Editor.

The **Edit Dropdown Menu** will appear with **Caption**, **Popup Key**, and **Bitmap** fields. Menu bar items can be dragged and dropped onto this new drop down menu, remember to hold the **<Ctrl>** key if dragging from existing menus or toolbars to make a copy rather than moving them. Commands may be given a **Caption** with the **&** entered into the string ahead of a specific character which will add an underline (indicating the accelerator key) and a shortcut key. This allows all three standard means of running a command: drop down menu item, toolbar icon, and a shortcut key.

Other items may be dragged onto this new drop down menu such as components from the Libraries Panel and other toolbar menu items.

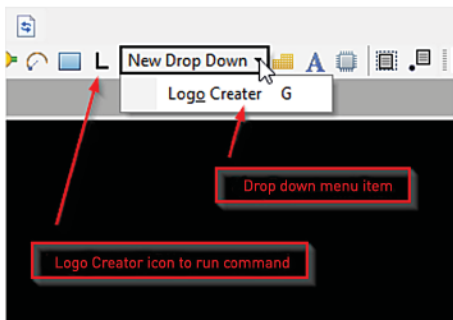


Figure 18: New command now on the menu.

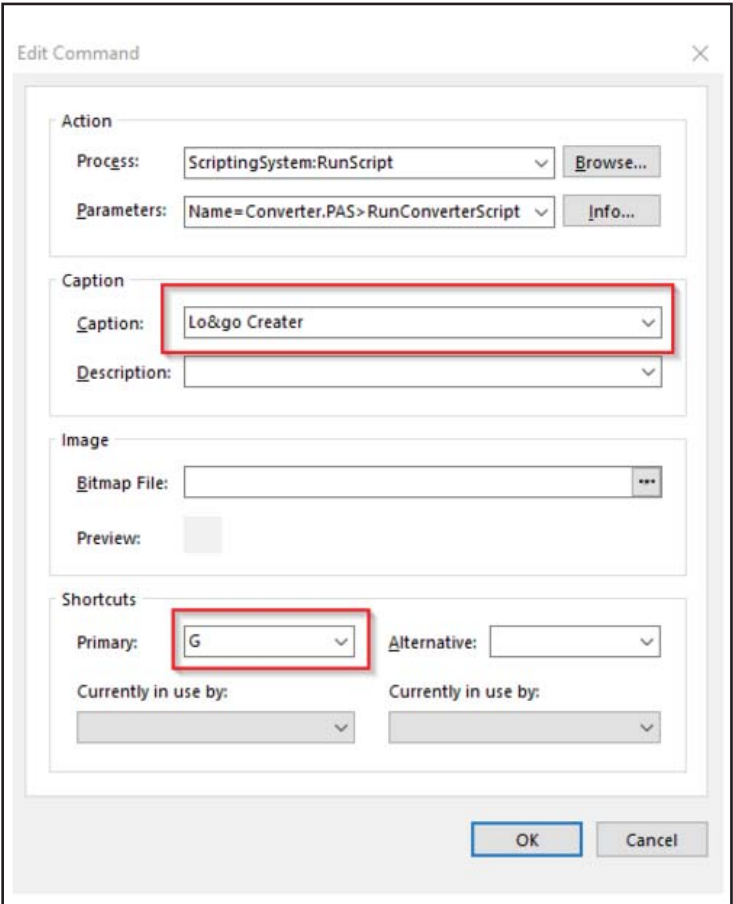


Figure 17: Assign a shortcut.

5 WAYS TO CUSTOMIZE ALTIUM DESIGNER FOR BETTER EFFICIENCY

5 — NEW TOOLBARS

By clicking on the **Toolbars** tab of the Customizing Editor, access may be gained to the various toolbars associated with the specific editor. As earlier mentioned this is where the **Restore** button may be used to set any of the standard toolbars back to their default settings.

Click the **New** button to create a new toolbar, given a default name, “*New ToolBar*”. This name can be changed by double clicking on the name “*New ToolBar*” and editing it to give it a meaningful name. The toolbars in Altium Designer are groupings of iconic command buttons and drop down menus. These may be rearranged by dragging them around the workspace, docking them on any margin, or in free space which does not require the Customizing Editor to be open.

Once a new blank toolbar has been created, drop-down menus, icon menu commands, and other objects may be added to this custom toolbar simply by dragging items to the blank toolbar.

CONCLUSION

Customization of the Altium Designer environment is very simple and adds valuable optimization to the user experience. If ever the toolbars become disorganized or menu items are lost the **Restore** button is always available to reset the Altium Designer toolbars back to their factory default.

To see some of these features in action there is a short video available on the Altium website located at: <https://altiumvideos.live.altium.com/#Detail/1213>

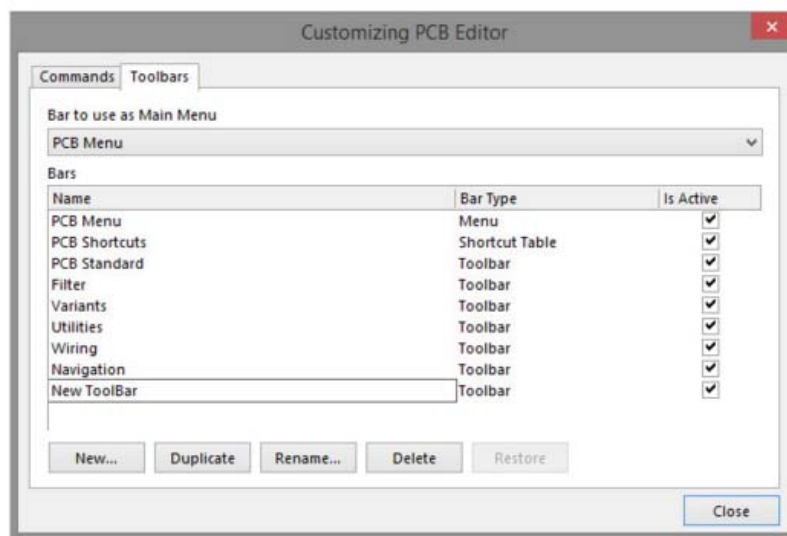


Figure 19: Click the **Toolbars** tab to edit or create toolbars.

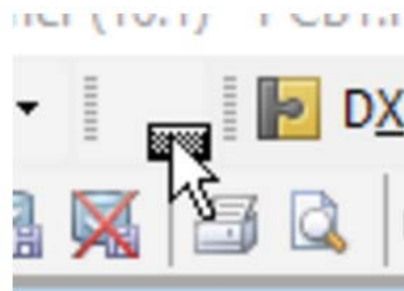


Figure 20: Populate your new toolbar by dragging items and dropping.