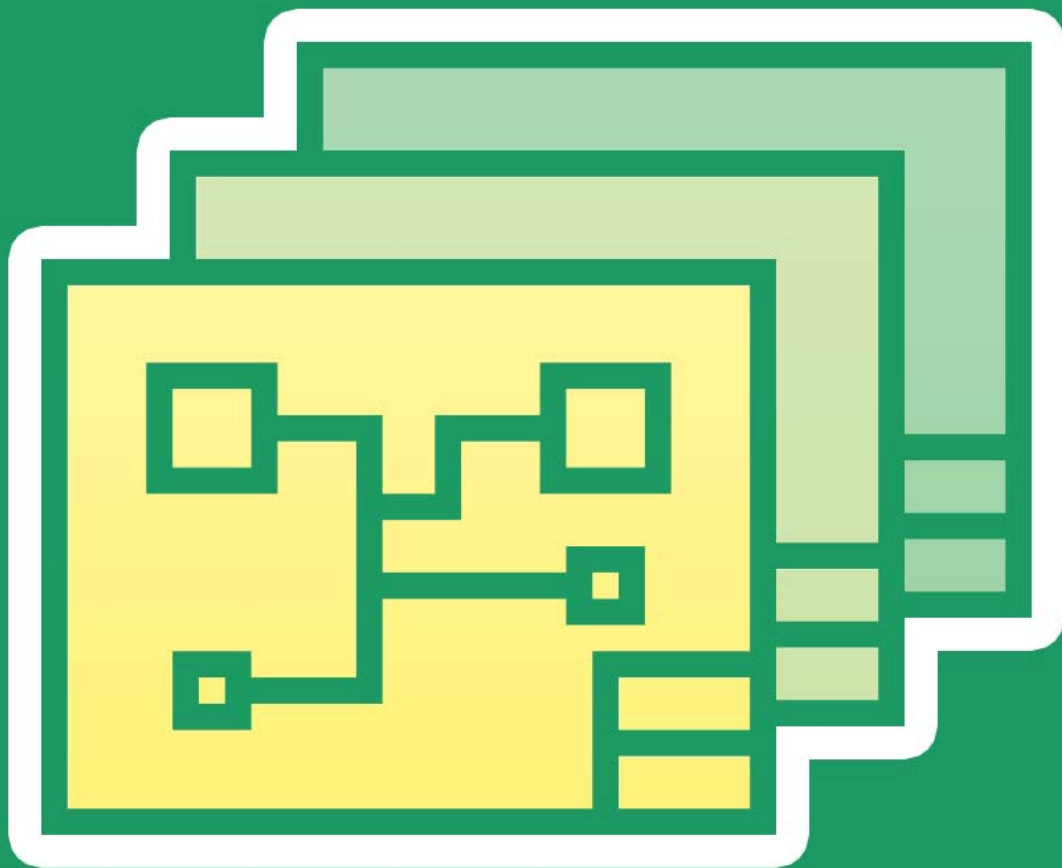# ALTIUM®

Quickly Create Project Libraries from the Schematic

Ivan Duprey

Applications Engineer

QUICKLY CREATE LIBRARIES FROM THE SCHEMATIC

When you are using an outside manufacturer, or sharing a design with another party that does not have access to your internal installed libraries, creating a project library makes the data easy to share. No matter how you created your library — from already installed libraries, Altium Vault® libraries, or even manually using a supplier search, you can localize the parts for easier creation. This allows for more portability of your design for external sharing.

There are methods to do this by editing each component properties or using the **Find Similar Objects** command and utilizing the Schematic Inspector, but they are not practical methods for large multi-sheet schematics. This paper shows how to use the Parameter Manager to quickly create schematic libraries, consolidating your components into a single, global project library.

INTRODUCTION

For this paper, we begin with a project that does not have a project library and all of the components have originated from installed libraries. See Figure 1.
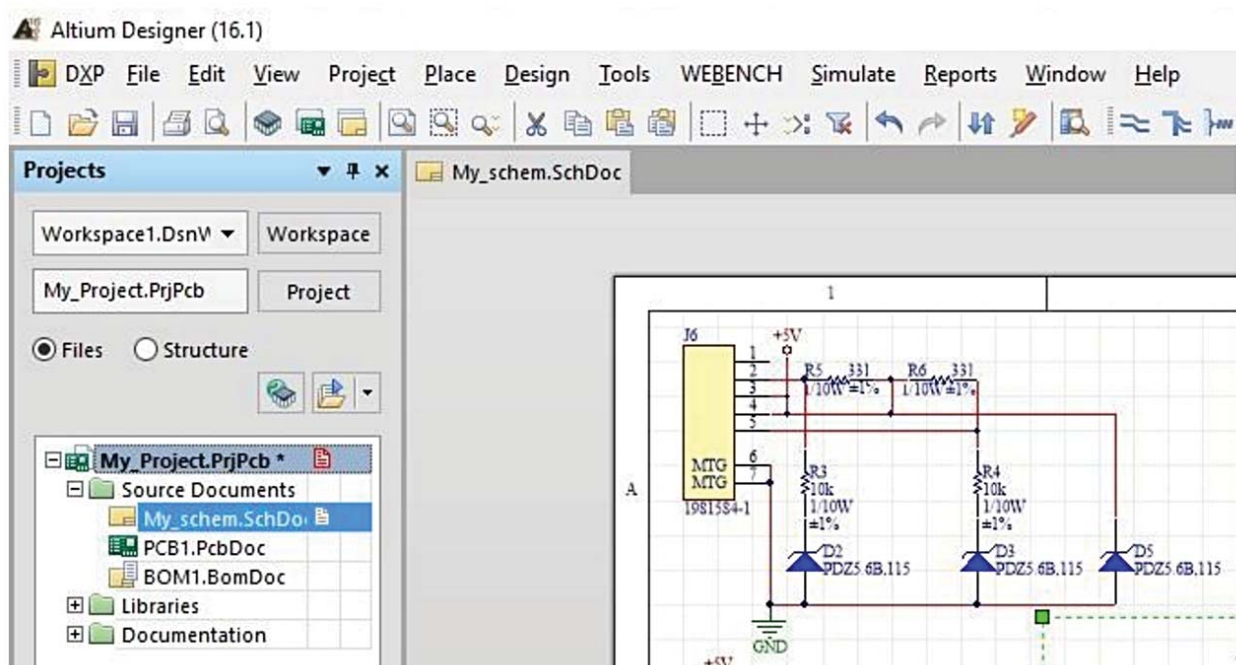

Figure 1: A project without a schematic library.

There is only one sheet in the example design. For a larger scale schematic, be aware that all sheets in the design must be open to use this method.

CREATING THE LIBRARY

We begin by creating a project library using all of the components in the design. You can create a schematic library out of the components that have been already placed on schematic documents in a project using the **Design » Create Schematic Library** command, which is available in the Schematic Editor. See Figure 2.

Altium.

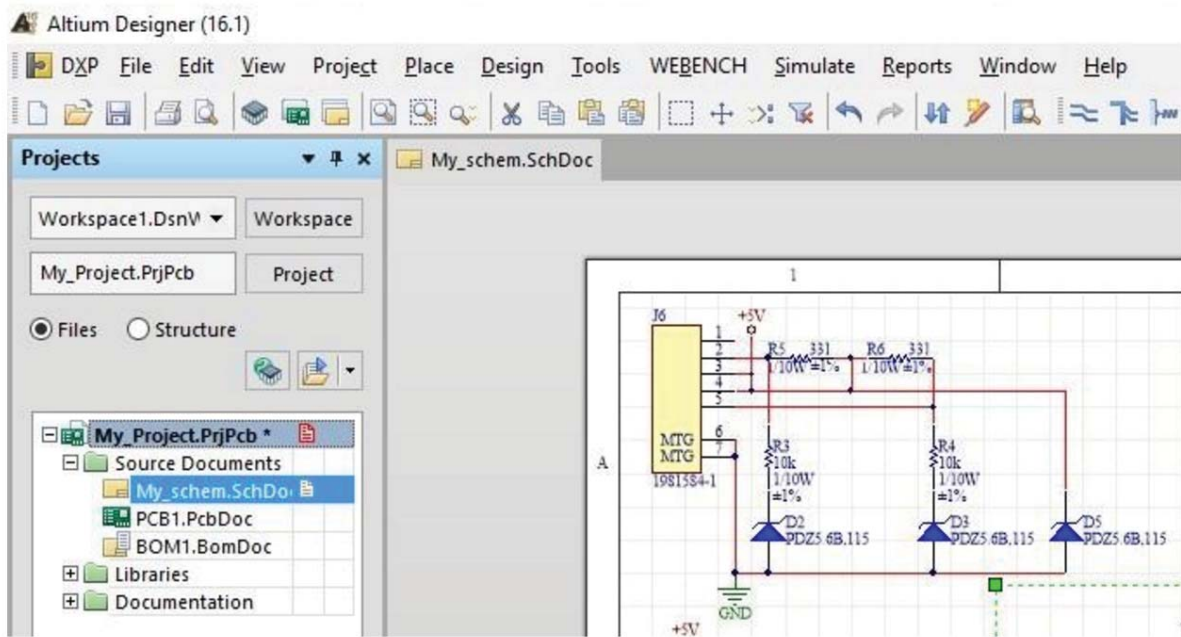# QUICKLY CREATE LIBRARIES FROM THE SCHEMATIC

Figure 2: ***Make a Schematic Library*** *menu command.*

A new schematic library will open in the Schematic Library Editor when it is created. All the components in the open schematic files are copied to the new schematic library, named **Project_name.SCHLIB**, stored in the same folder as the project file (**Project_name.PRJPCB**). The filename will appear in the **Projects** panel in the *Libraries\Schematic Library Documents* folder. An information dialog box will open. Click **OK** to confirm.

Save or rename the new schematic library using **File » Save As** and close it. The components on the schematic retain their library links associated with the libraries from which they originated. You can verify this by double clicking on any component on the schematic to open the **Properties for Schematic Component in Sheet** dialog and checking the **Link to Library Component** as shown in Figure 3.
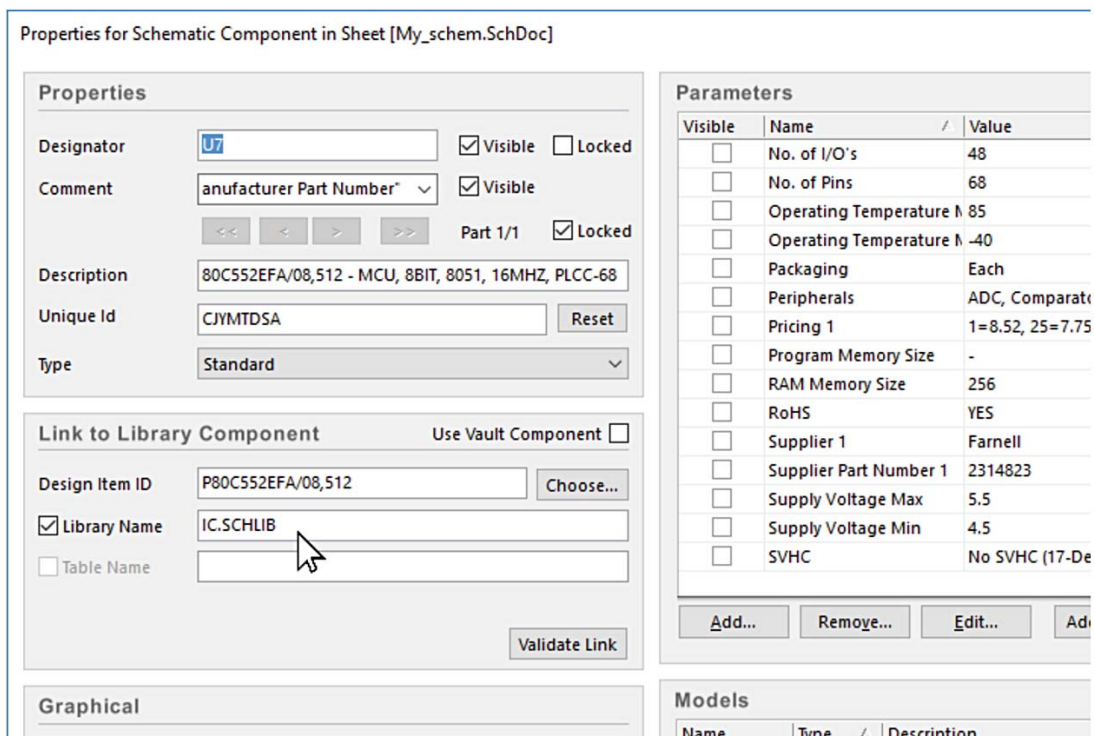

Figure 3: ***Properties for Schematic Component in Sheet*** *dialog. You can also see this in the Parameter Manager.*

QUICKLY CREATE LIBRARIES FROM THE SCHEMATIC

Parameters can be added and edited individually or using the Parameter Table Editor to add and edit them across the entire design, or across a whole library. When you open the dialog, it gathers all parameter data for the entire design and presents it in a table-like grid. The Parameter Table Editor is launched by selecting **Tools » Parameter Manager**. See Figure 4.
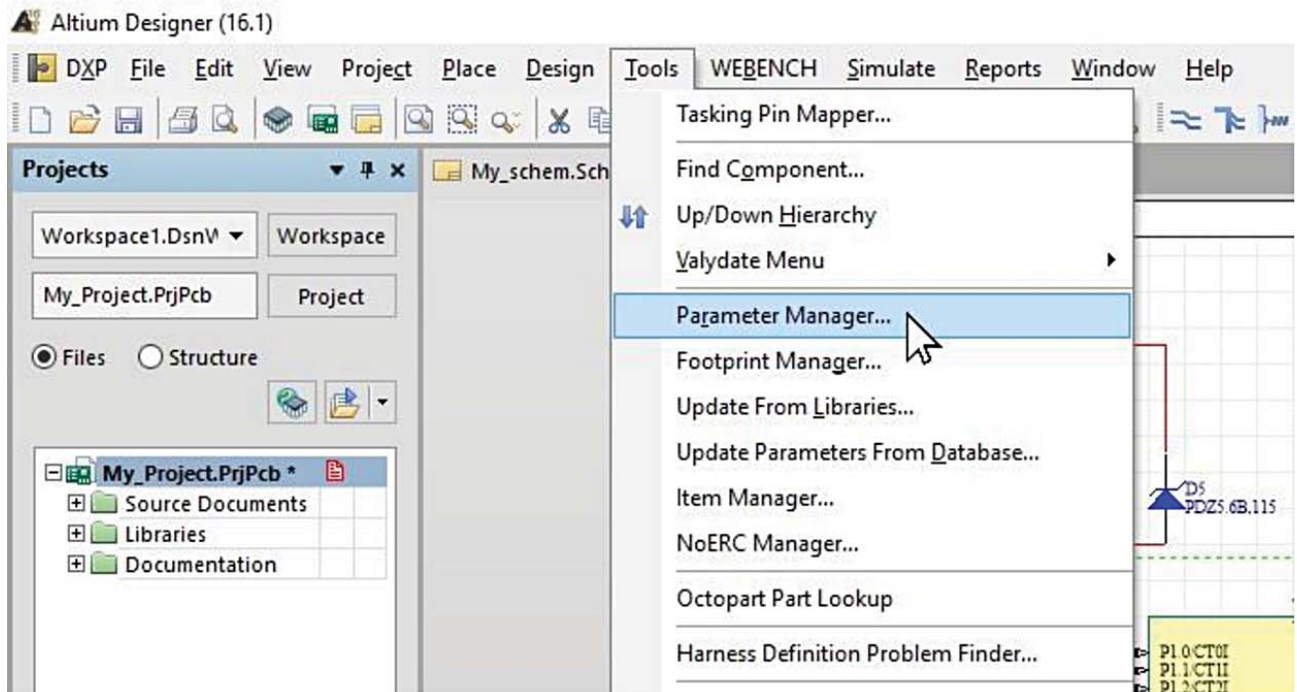

Figure 4: Opening the Parameter Manager

After selecting Parameter Manager from the menu, the Parameter Editor Options dialog appears first. In this dialog (Figure 5), you determine which type of parameters you want to be loaded into the Parameter Table Editor dialog.

For this paper, you will be working on component parameters so you would disable all options in the **Include Parameters Owned By** section, except for the **Parts** option. In the **Exclude System Parameters** option uncheck both items. Once the Parameter Table Editor dialog is open, you can clearly see the component library links to different libraries. See Figure 6 on the next page.
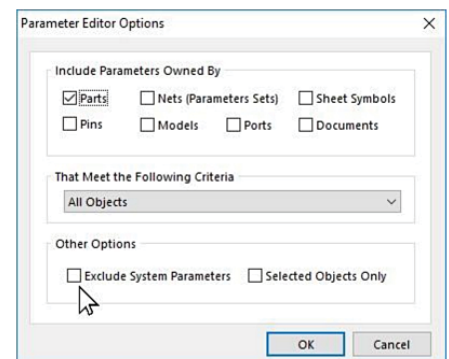
.

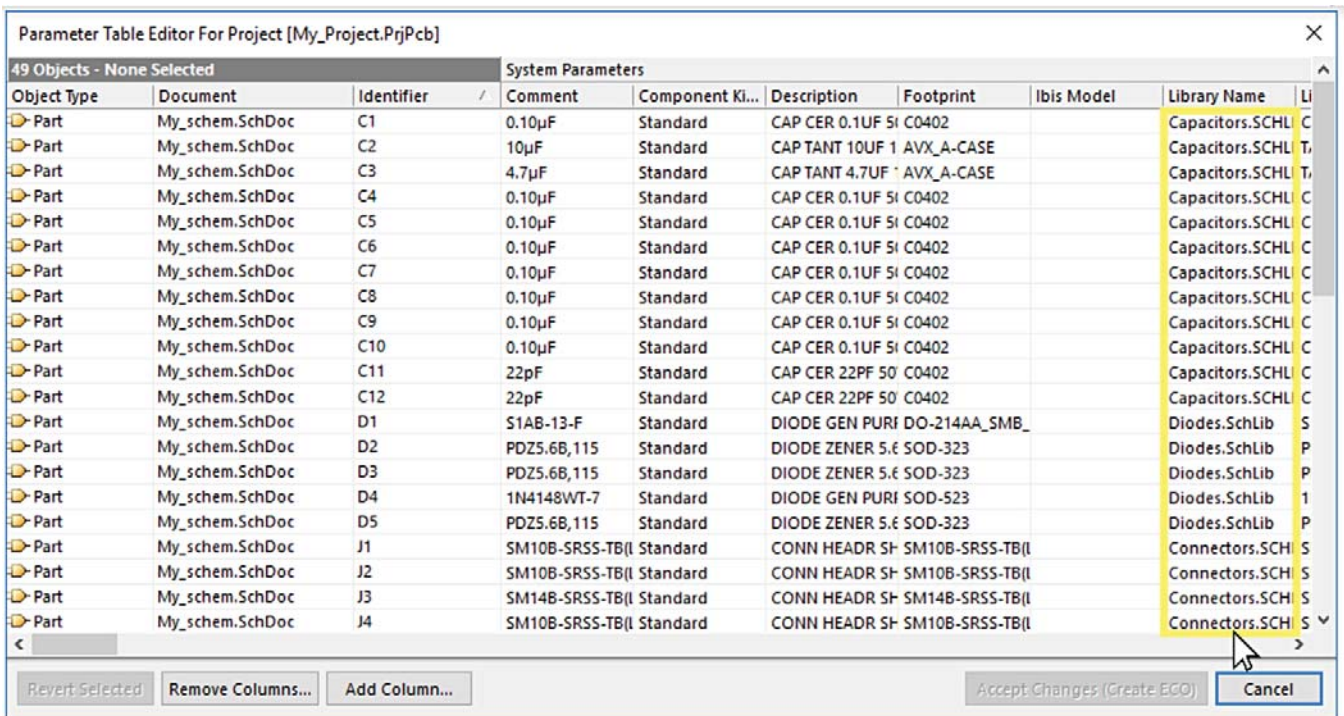*Figure 5: Parameter Editor Options.*

Altium.

*Figure 6: Parameter Table Editor (**Library Name** field highlighted).*

Now, link all of the components to the newly created **My_Project.SCHLIB**. First, right click any library name in the column **Library Name** and click on **Select Column** from the menu. See Figure 7
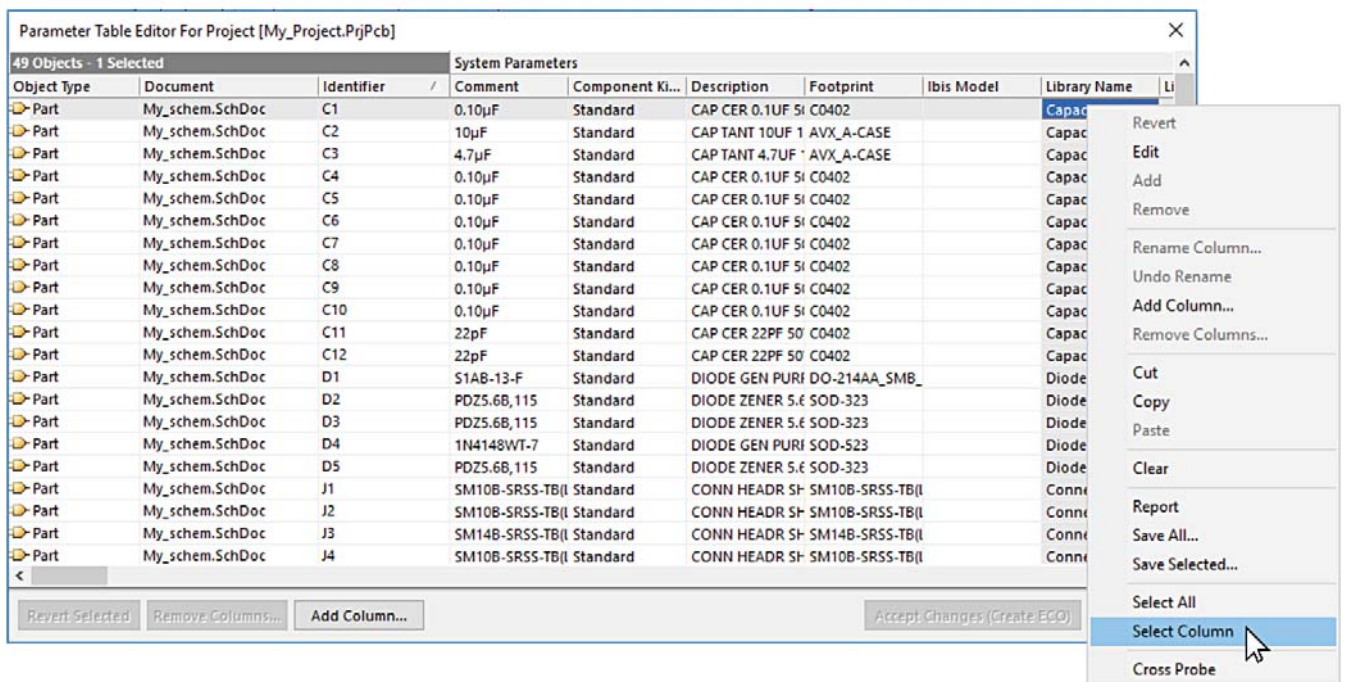

Figure 7: Selecting an entire column in the Parameter Table Editor.

QUICKLY CREATE LIBRARIES FROM THE SCHEMATIC

Once the entire **Library Name** column is highlighted, right click any field within the column and click on **Edit** from the menu. See Figure 8.
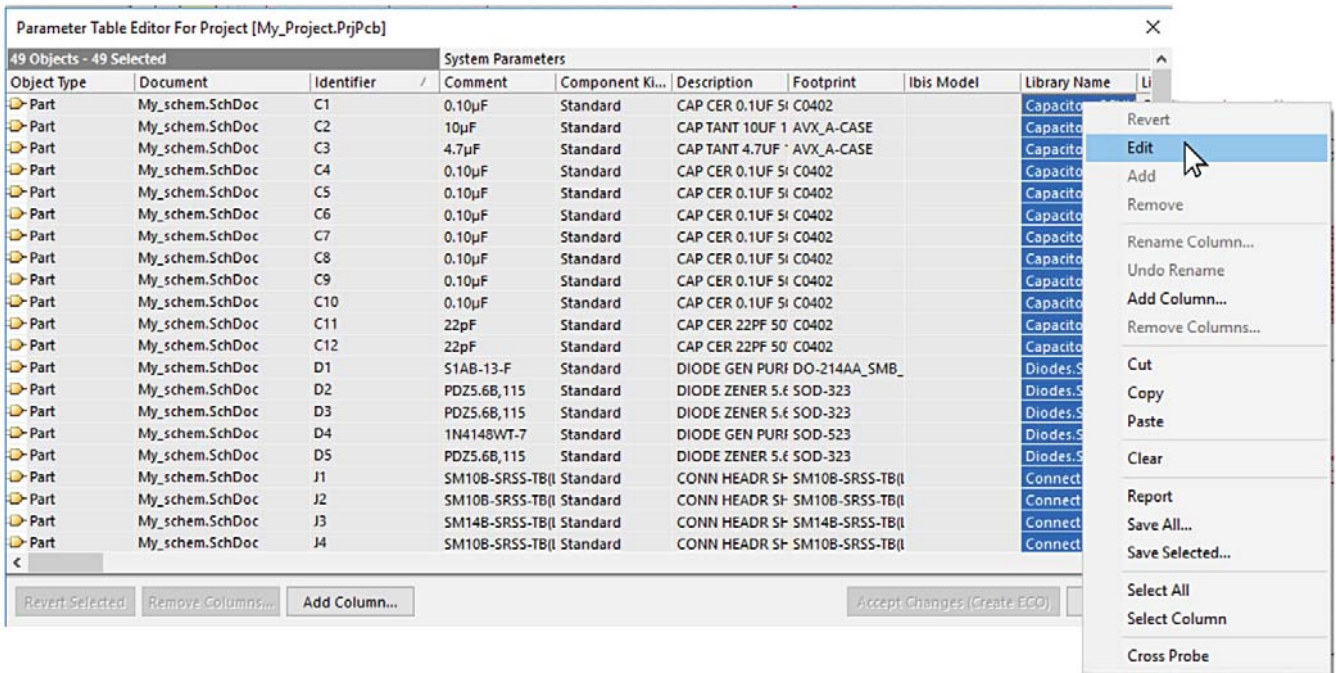

*Figure 8: Right click and select **Edit**.*

Within the selected cell, enter the name of the library you created earlier in this exercise **My_Project.SCHLIB** and then hit **Enter**. The entire field in the column will update with the library name you have entered.
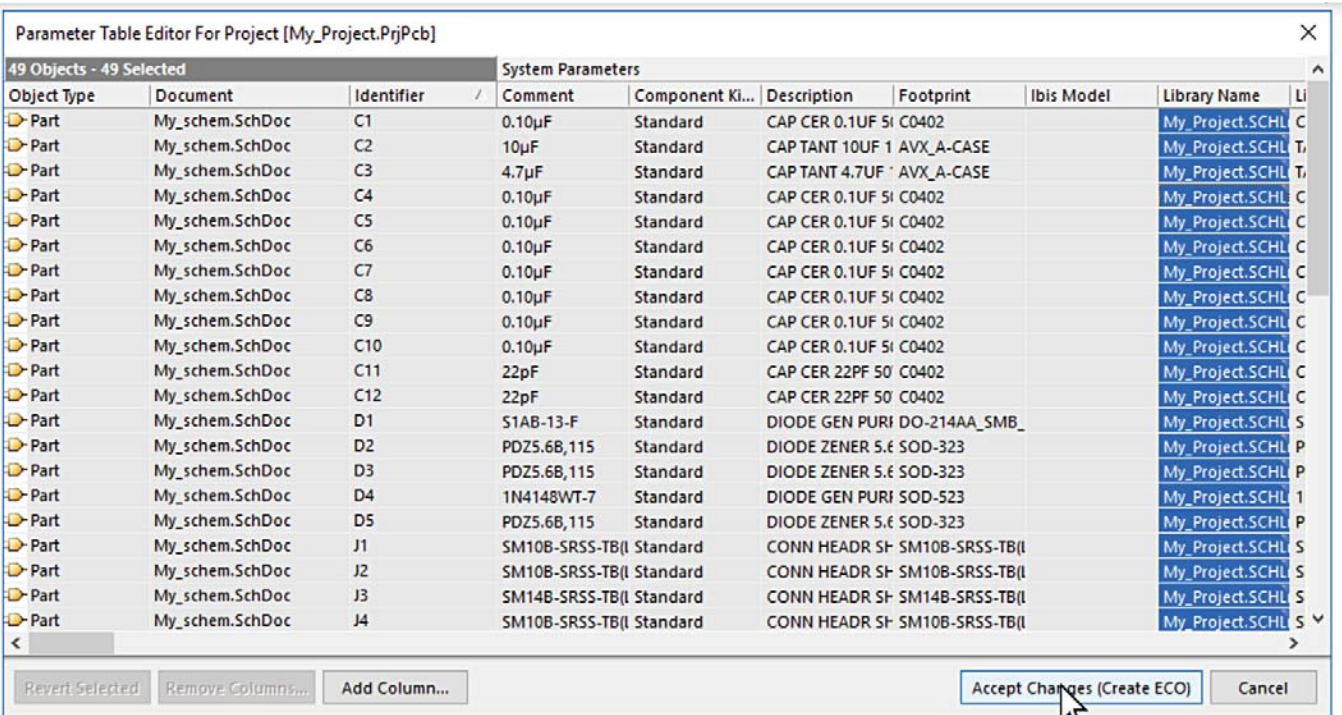

*Figure 9: Column updated: click **Accept Changes**.*

QUICKLY CREATE LIBRARIES FROM THE SCHEMATIC

Once you see that the new library name appears in the entire column, you must click **Accept Changes (Create ECO)** shown in Figure 9 (previous page). This will open the Engineering Change Order dialog where you will be able to review the changes before selecting the **Execute Changes** button, shown in Figure 10.
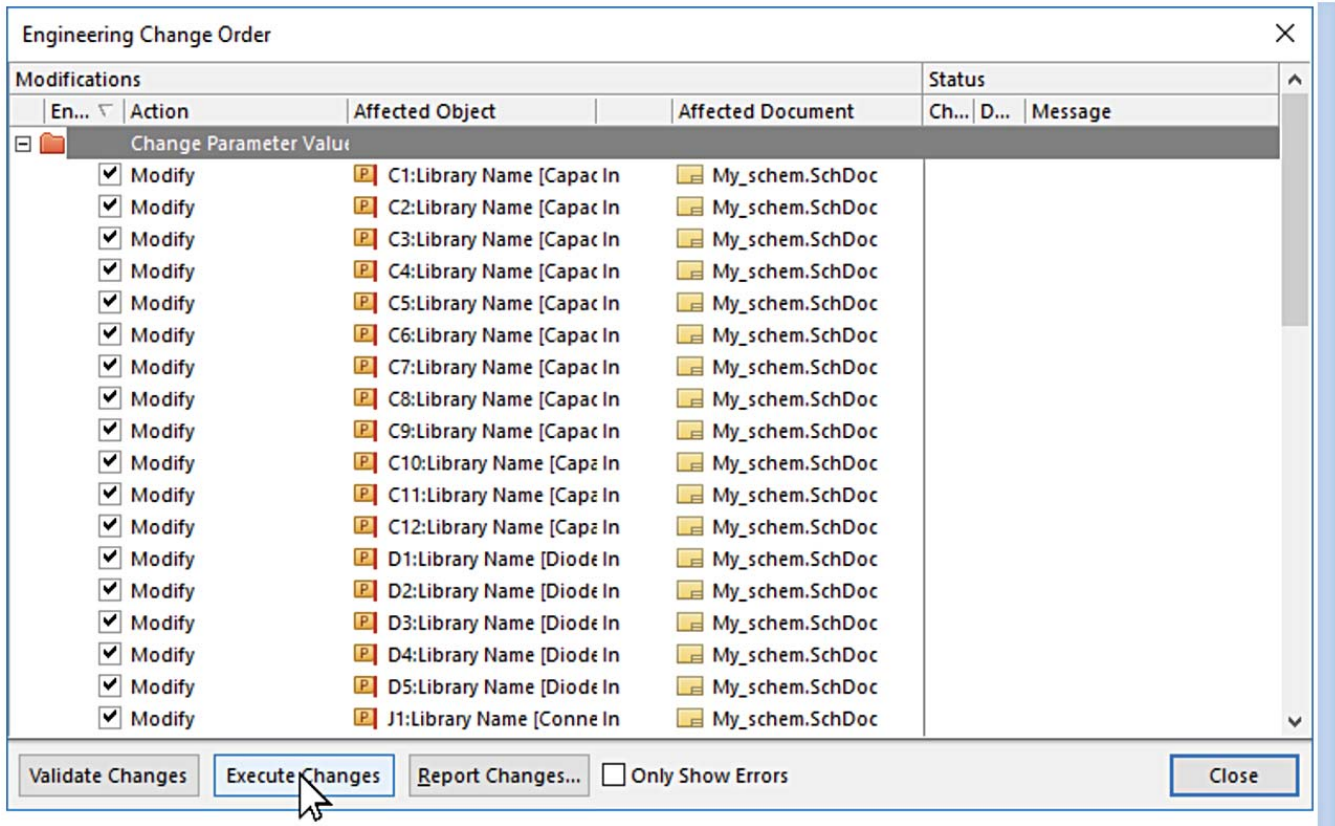

Figure 10: Engineering Change Order dialog.

Once the changes are executed, you have localized your library to the project and severed any dependencies on external library resources.

CONCLUSION

The Parameter Table Editor dialog allows the designer to edit and update parameters across the entire project. The flexibility of the functions available within the parameter manager also allows it to be used as a tool to manipulate system level parameters such as Library Links that can simplify the management of libraries within a project.

Altium.