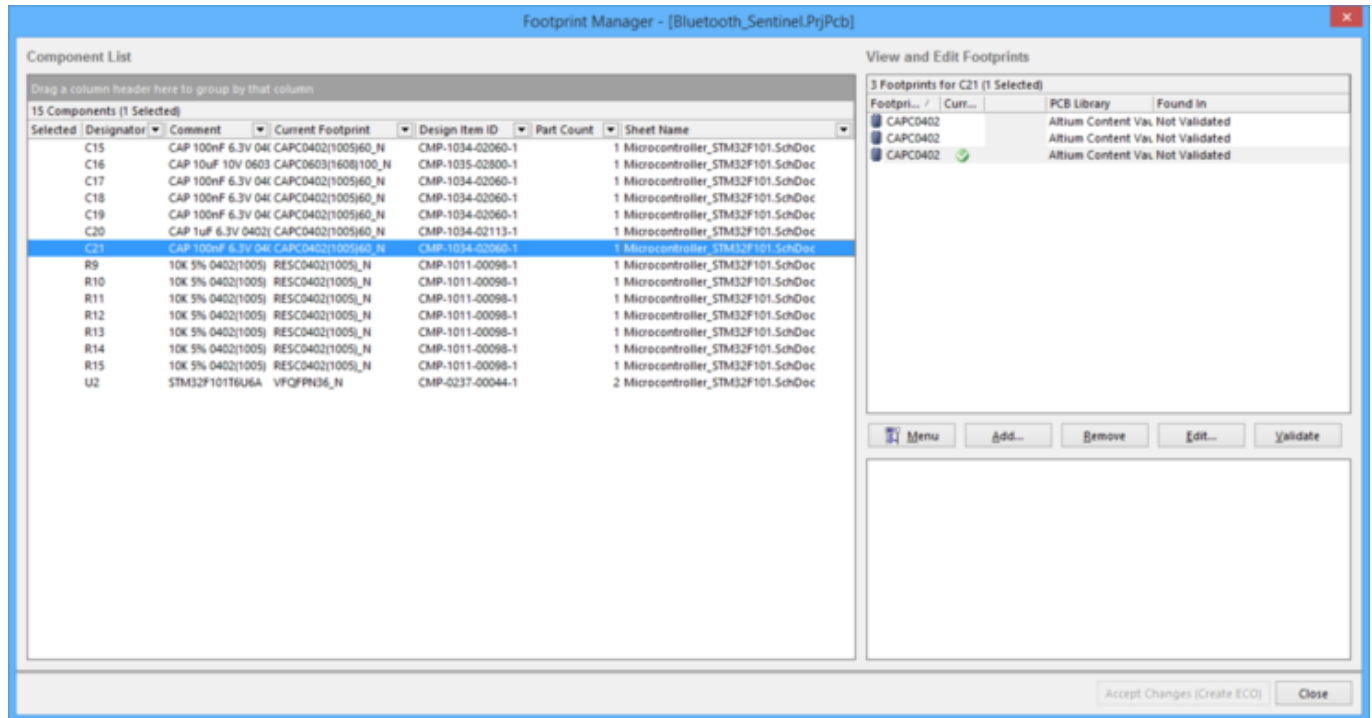# Altium ®

Using Altium Documentation
Modified by Phil Loughhead on Jun 17, 2017

The *Footprint Manager* dialog

Summary

This dialog allows the user to preview, validate, add, remove, or edit footprint associations across the entire project.

Access

The *Footprint Manager* dialog is accessed by clicking **Tools » Footprint Manager** in the schematic editor.

Options/Controls

Component List

This is a list of all components from the schematic sheets of the currently focused PCB design project. Information includes:

- Designator
- Comment
- Current Footprint
- Design Item ID
- Part Count

- **Sheet Name**

> Click a component in this list to review its footprints in the list on the right. You can add, remove or edit footprints for the selected component.

> ✅ To group the list, drag a column header to the area above the list where specified. The information will be grouped into a concise listing for easier readability. Click +/- to expand/contract grouped items.

> ✅ **Select All** is available from the right-click menu to quickly select all components in the list.

View and Edit Footprints

This is a list of footprints associated with the selected component on the **Component List**. Use the controls at the bottom of the region or the right-click menu commands to manipulate the footprints in the list.

> ✅ Click on a footprint to see a graphical view of that footprint in the region below.

Right-click Menu

- **Add** – click to open the [PCB Model dialog](#) to choose a new footprint model to add for the highlighted schematic component.
- **Remove** – click to remove the selected footprint.
- **Edit Footprint** – click to open the *PCB Model* dialog to review the highlighted footprint's attributes such as the name and the location of the PCB library from which the footprint comes.
- **Validate Footprint Paths** – click to validate the full path to this footprint.
- **Change PCB Library** – click to open the *Edit PCB Library* dialog to change the PCB library of the selected footprint.
- **Copy** – click to copy the selected footprint.
- **Paste** – click to paste the selected footprint.
- **Select All** – click to select all footprints on the list.
- **Set As Current** – click to set the selected footprint as the current footprint (denoted by a ✅ in the **Current** column).
- **Add to All Parts** – click to add the selected footprint to all in the list.

Additional Controls

- **Menu** – click to view a menu with the same controls as the right-click menu above.
- **Add** – click to open the *PCB Model* dialog to choose a new footprint model to add for the highlighted schematic component.
- **Remove** – click to remove the selected footprint.
- **Edit** – click to open the *PCB Model* dialog to review the highlighted footprint's attributes such as the name and the location of the PCB library from which the footprint comes.
- **Validate** – click to validate the existence of the highlighted footprint. If the footprint cannot be found, **Not Validated** appears in the **Found In** field. If the footprint is validated, a full path to this footprint is displayed.
- **Accept Changes (Create ECO)** – click to accept the changes to schematic components and their linked footprint models and execute the [Engineering Change Order dialog](#). With the ECO dialog, you can validate and execute the changes after reviewing the

modifications made to the footprint values for the corresponding schematic components.