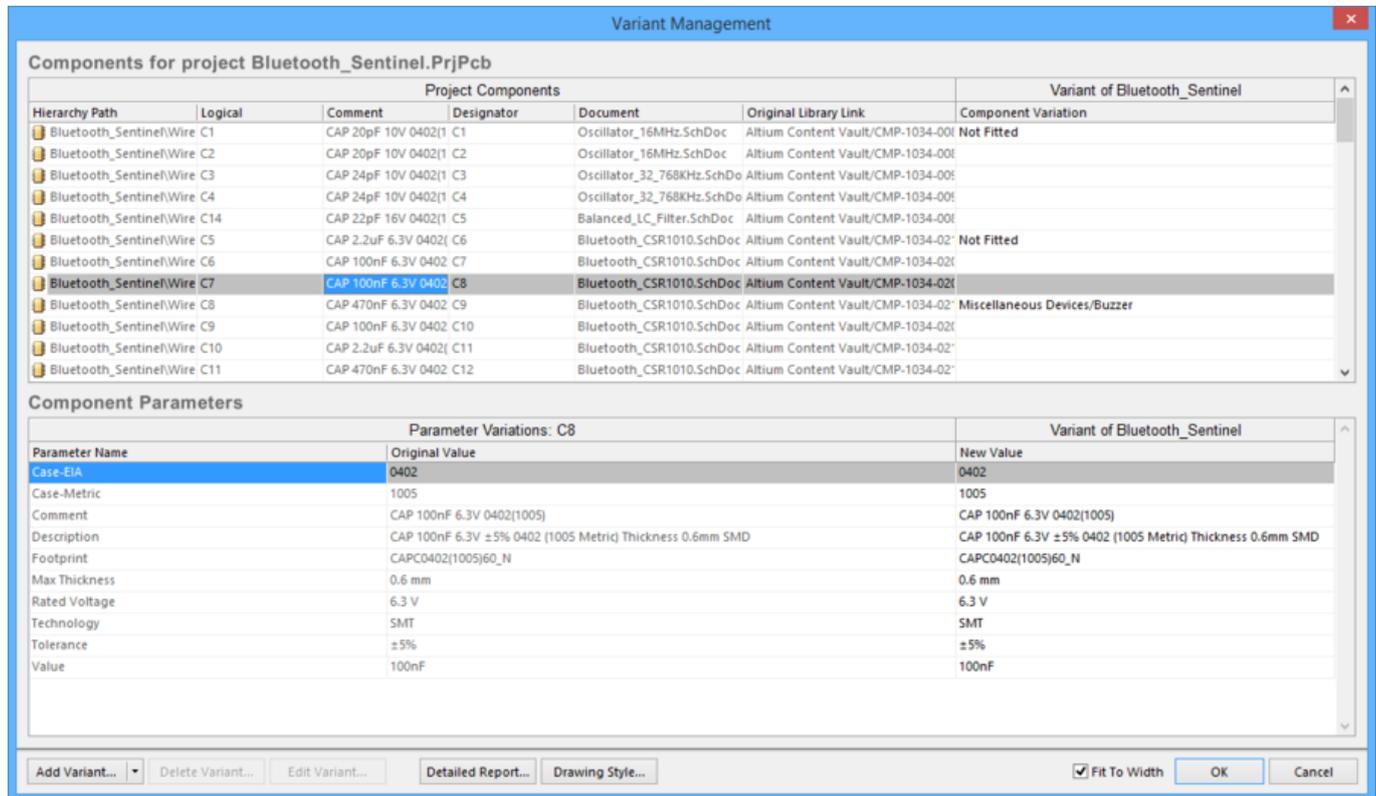


[Home](#) > Variant Management

Using Altium Documentation

Modified by Rob Evans on Jun 19, 2017



The Variant Management dialog

Summary

This dialog enables you to add and configure variants of the main PCB board design. Variants are variations of the main PCB board design and by using variants, any number of variations of the base design can be designed. Each component in the base design can be configured to be:

- *Fitted* – the default setting when a new variant is created.
- *Not Fitted* – the original component as used in the base design is not fitted/used in this variant of that design.
- Fitted with modified component parameters, such as the component's value.
- *Alternate Part* – completely replacing one component with another.

Variants that use any of these types of variations are all referred to as *Assembly Variants* since they only impact the assembly process. All variants share the same fabricated base board.

Access

The *Variant Management* dialog can be accessed in the following ways:

- Click **Project** » **Variants** from the main menu of any document in the project.

- In the *Projects* panel, right-click on project name and select **Variants**.
- In the Schematic Editor, right-click a placed component then select **Part Actions » Variants**.
- In the Schematic Editor, click the  icon in Variants Toolbar.

 Use the **Part Actions » Variants** option to populate the *Variant Management* dialog with the parts that are currently selected in the Schematic editor. If multiple parts are selected in the schematic, then only those parts will be listed for variant editing. See [Defining a Variation from the Schematic](#) for more information.

Options/Controls

The dialog is divided into two main regions:

- The upper region, titled **Components**, lists all of the components in the base design.
- The lower region, titled **Component Parameters**, details all of the parameter variations of the component(s) currently selected in the upper region.

Components

- **Project Components** - this region of the dialog lists each component in the project. For each component, the **Hierarchy Path** and **Logical** designator are listed, as well as **Comment** and physical **Designator**. The **Document** and **Original Library Link** list where source documents are located.

 Double-click on a component, or right click and select the **Cross Probe** option, to jump to that component on the schematic.

- **Component Variation** - After a new variant is added, a dedicated column appears to the right of **Project Components**. The name given the new variant appears as the title and the **Component Variation** column lists any variations for each component. Once the variant has been created, you can configure the state of each component. This is done by clicking the component cell in the new column to reveal the  button, or by right-clicking to access the context menu commands. When the  button is clicked, the [Edit Component Variation](#) dialog opens and presents three choices:
 - **Fitted** - the original component as used in the base design is also fitted/used in this variant of that design. For a newly added variant, all components are **Fitted** by default and the cell is empty. Note that individual parameters can also be varied for a **Fitted** component - simply type in the new parameter value. Varied parameters are shown in bold.
 - **Not Fitted** - the original component as used in the base design is not fitted/used in this variant of that design. For a **Not Fitted** component, the cell displays the text **Not Fitted**.

- **Alternate Part** - this option gives access to browsing and selecting the alternate part. Once chosen, the cell displays the alternate part's Library Link. The lower region of the dialog will display all of the parameters in the alternate part.

To make a variant active in the dialog, either click on that variant's name in the column header, or click in any of that variant's cells.

When you configure variations in the *Variant Management* dialog, the settings are saved in the project file. This includes the **Not Fitted** state, local parameter variations to a **Fitted** component, and the parameter values of **Alternate Parts**. The **Alternate Parts** are stored in the file <ProjectName>.PrjPcbVariants.



To change the order the variants are listed, click and hold on the column heading, then drag that column to a new location. Use this in combination with the **Fit to Width** checkbox to position and size the variant of interest in your preferred working location.

- **Right-click Commands** - the following commands are available on the Components region's right-click menu:

- **Columns** - click to see/hide columns that are not needed. Toggle the visibility of any column.
- **Edit Selected** - make changes to the selected component.

Edit Selected is only available when right-click is done in a **Component Variation** column.

- **Set Selected As** - click to change to **Fitted**, **Not Fitted**, or **Alternate**.
- **Only Show Varied Components** - click to show varied components in the listing.
- **Filter** - click to filter components listed by **Show Fitted with Varied Parameters**, **Show Not Fitted**, or **Show Alternate**.

Filter is only available when **Only Show Varied Components** is checked.

If you have been experimenting with the **Filter** options and have applied different filters, you may need to reset the filter before attempting another update. To do this, disable the **Only Show Varied Components** option to clear the filter system, then re-apply it and clear the **Filter** options as required.

- **Cut** - click to delete the selected variant.
- **Copy** - click to make a copy of the selected item.
- **Report** - click to open the [Report Preview](#) dialog to create a printout of the **Components** region.
- **Save All** - click to open the *Save Grid Contents to File* system dialog to save a listing of all project components to another location, as a tab formatted text file.
- **Save Selected** - click to open the *Save Grid Contents to File* system dialog to save a listing of all selected project components to another location, as a tab formatted text file.
- **Select All** - click to select all cells.

- **Select Column** – click to select the current column.
- **Invert Selection** – click to select all components other than those that are currently selected.
- **Cross Probe** – click to show, on the schematic, the selected component's location.

Standard Windows multi-select techniques are supported; use these to select and configure multiple components simultaneously.

Component Parameters

Each component in the *Variant Management* dialog can have **Parameter Variations**. The **Parameter Name** and its **Original Value** are listed, along with **New Value** listings for columns representing Variants that have been added. The **New Value** that appears is a copy of the base design component value until changes are made. Right-click in a cell to access the range of parameter commands from the pop up menu. The menu offers the following options:

- **Edit Selected** – click to change **New Value** of the selected variant cell.
- **Reset Selected** – click to restore **New Value** back to original value.
- **Reset All** – click to restore all **New Value** cells to their original value, regardless of current selection.

Edit Selected, **Reset Selected**, and **Reset All** are available only when right-clicking in an added variant column.

- **Update Values From Library** – click to bring any parameter changes made to a library component that has been used as an Alternate Part into the variant definitions. Note that this updates the parameters only, and not the component itself.
- **Only Show Varied Values** – click to show only those values that have been varied from the original values.
- **Cut** – click to delete cell contents. A **Confirm** window opens for confirmation before deletion occurs.

Cut is available only when right-click is done in the **Component Variation** and **New Value** columns.

- **Copy** – click to make a copy of the selected item.
- **Report** – click to open the [Report Preview](#) dialog to create a printout of the **Component Parameters** region.
- **Save All** – click to open the *Save Grid Contents to File* system dialog to save all parameters for the selected component to a formatted text file in another location.
- **Save Selected** – click to open the *Save Grid Contents to File* system dialog to save all selected parameters to a formatted text file in another location.
- **Select All** – click to select all cells.
- **Select Column** – click to select the current column.
- **Invert Selection** – click to select all parameters other than those that are currently selected.

Additional Controls

- **Add Variant** button – click to create a new variant of the base design, via the the [Edit Project](#)

[Variant](#) dialog. Use this dialog to add, edit or remove a variant, and specify any required variant-level parametric data. This button includes a dropdown that is used to access the **Clone Selected Variant** command. Cloning is particularly useful when you need to define a number of variants that are very similar.

- **Delete Variant** - click to delete selected Variant(s). A *Confirm Delete Variant* dialog opens for confirmation before deletion occurs.
- **Edit Variant** - click to open *Edit Project Variant* dialog to edit a variant.
- **Detailed Report** - click this button to open the [Variant Report](#) dialog to generate a detailed variant report in HTML format.
- **Drawing Style** - click this button to access the [Variant Options](#) dialog from where you can define how non-fitted components and varied parameters will appear, both in the compiled document view of the schematic and in schematic prints. You can also define appearance for these components in PCB assembly drawings.
- **Fit to Width** - check this option to automatically adjust the width of the columns, based on the content.

Source URL:

[https://www.altium.com/documentation/display/ADES/WorkspaceManager_Dlg-VariantManagementForm\(\(Variant+Management\)\)_AD](https://www.altium.com/documentation/display/ADES/WorkspaceManager_Dlg-VariantManagementForm((Variant+Management))_AD)