Generating Output for an Assembly Variant

Contents

- Selecting the Variant
- Using an Output Job file
    - Schematic prints
        - Labeling the print with the variant name
    - PCB assembly drawings
- Direct report generation
- Smart PDF support

Altium

Assembly and/or report-based output for a variant is typically generated using an Output Job Configuration file (*.OutJob). Such output includes Schematic Prints, Assembly Drawings and a Bill of Materials. The advantage of using an Output Job file is that the OutJob is a self contained container of outputs - it is easy to see which outputs are required, check their configuration, and regenerate outputs as required. OutJobs can also be easily copied from one project to another, ensuring portability of output settings between projects.

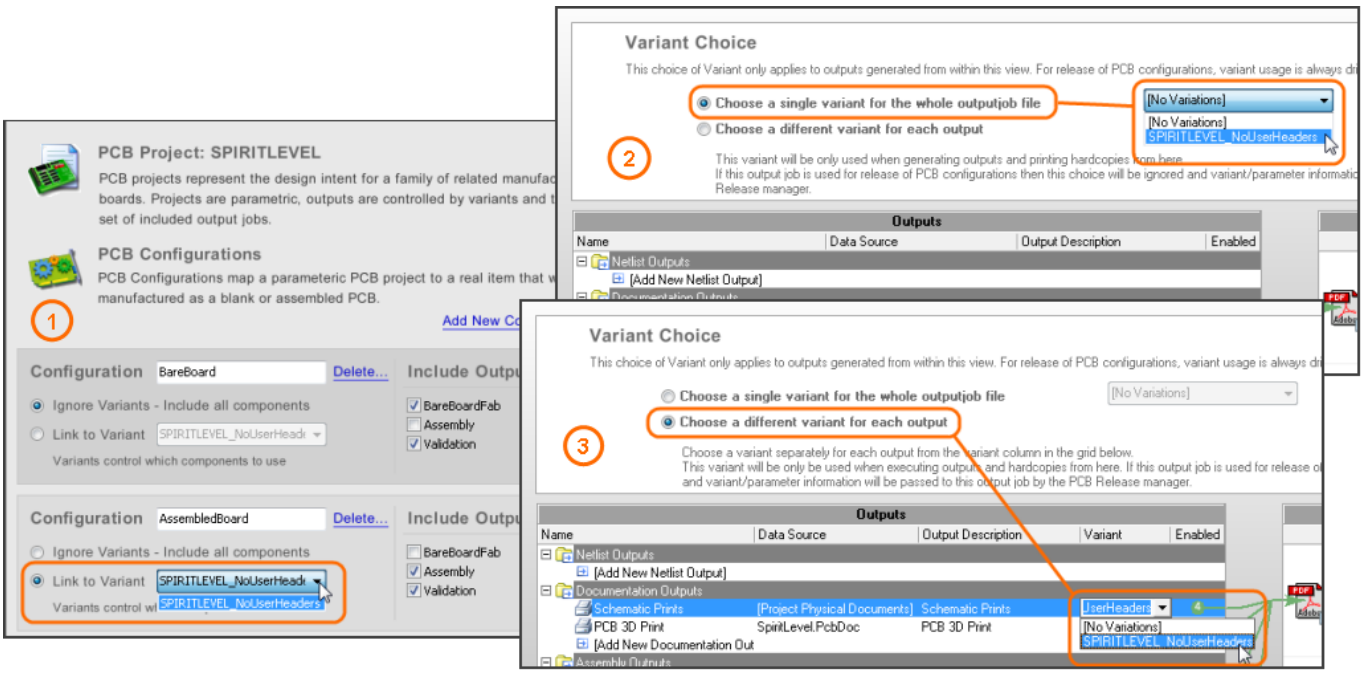
Variant-based output can also be generated using Altium Designer's Smart PDF feature, and also via the direct report generation commands in the Schematic and PCB Editors. The following sections take a look at the support available for documenting assembly variants.

Selecting the Variant

If you are using Altium Designer's variant feature you will need to decide how to best organize your project outputs before generating variant output information. In Altium designer there are 3 different modes of where the variant is selected, each suiting a different style of project management:

1. **As part of the project configuration** - use this mode if your project is using Altium Designer's managed design capabilities, where the variant is selected as part of the Project Configuration.
2. **For all of the outputs in an OutJob** - use this mode if you have a separate OutJob file for each variant, select the variant at the top of the OutJob.
3. **For individual outputs within an OutJob** - use this mode if you have all variant outputs defined within the one OutJob.

For option 1 the variant is chosen in the *Configuration Manager* dialog. For options 2 and 3 the variant is selected in the Output Job file, as shown in the the image below.



Selecting where variant is chosen: 1) in the configuration, 2) for the entire OutJob, or 3) for each output in the OutJob.

Using an Output Job file

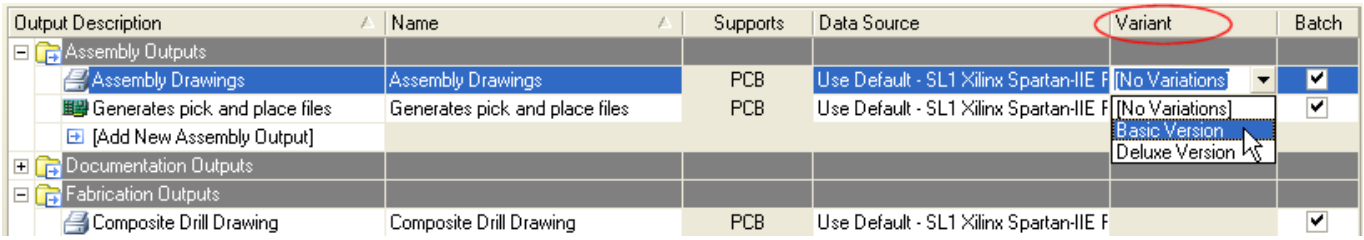
Output Job Configuration files are defined and managed within the OutputJob Editor. An Output Job file enables you to define design output configurations - assembly, reports, etc - exactly as required, all in the one convenient and portable document. The majority of documentation that can be generated for an assembly variant will be configured and output from an Output Job file.

A new Output Job Configuration file can be created for the active project by:

- Selecting the **File » New » Output Job File** command.
- Right-clicking on the project's entry in the **Projects** panel and choosing **Add New to Project»Output Job File** from the pop-up menu that appears.

The new file will be opened as the active document in the OutputJob Editor. If variant control is being managed at the OutJob level, select the mode using the options at the top of the OutJob. If you select the option **Choose a different variant for each output**, then for each documentation type that can be generated for assembly variants, an associated Variant field will become available. The remainder of this article will assume this mode has been selected, to give the most detailed coverage of possible options.

For Schematic Prints, the Data Source field must be set to [Project Physical Documents] in order to access the Variant field. (See the next section for further information).
Use the Variant field to determine whether a particular variant of the design is used as the source when generating the output, or whether the full, base design is used. In the latter case, the entry should be set to the default [No Variations].



Specifying an assembly variant as the source for document generation.

Assembly variants can be used as the source when generating the following documentation types:

• Assembly Drawings	• Pick and Place Files	• Bill of Materials	• Component Cross Reference Report
• Schematic Prints	• Report Project Hierarchy	• Report Single Pin Nets	• Simple BOM

It is important to remember that although such documentation types can be generated directly using commands accessible from within the respective document editor (from the File and Reports menus), these commands relate to the full base design and not to any variant of that design. The exception to this is documentation generated involving the Report Manager dialog (Bill of Materials, Component Cross Reference, etc), whereby you can select which variant to use directly within the dialog (see Direct report generation).

For further information on Output Job Configuration files, including output definitions and generation, see Design to Manufacturing

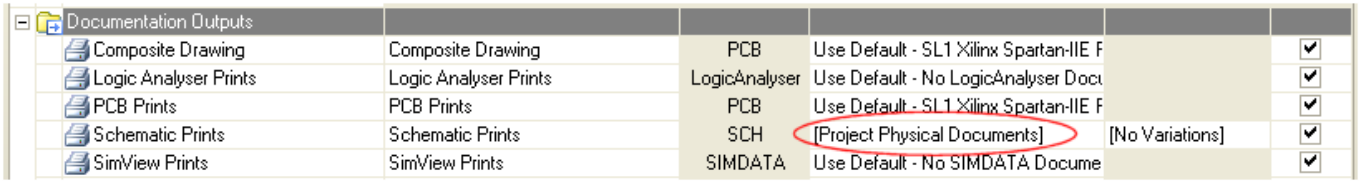
Schematic prints

What are the Project Physical Documents?

In case you're wondering, the term Project Physical Documents is used to distinguish the documents for the thing you are actually going to make (the physical thing), from the logical design documents you captured as you designed that thing.
So just why are the logical design documents different from the Project Physical Documents, you ask? That's because the logical source documents might cover a number of physical variants that you make from the one project. Or the design might use Altium Designer's multichannel capabilities, where the logical channel is captured once and then instantiated X times in the product - in both of these cases printing the Project Physical Documents will give a different result to printing the logical source design documents.

As mentioned in the previous section, an Output Job file can be used to generate schematic prints, the source of which can be an assembly variant. In reality, what this means is that the full design appearing on the source schematic document(s) is printed, with the ability to determine how components that are not fitted for that variant are drawn.

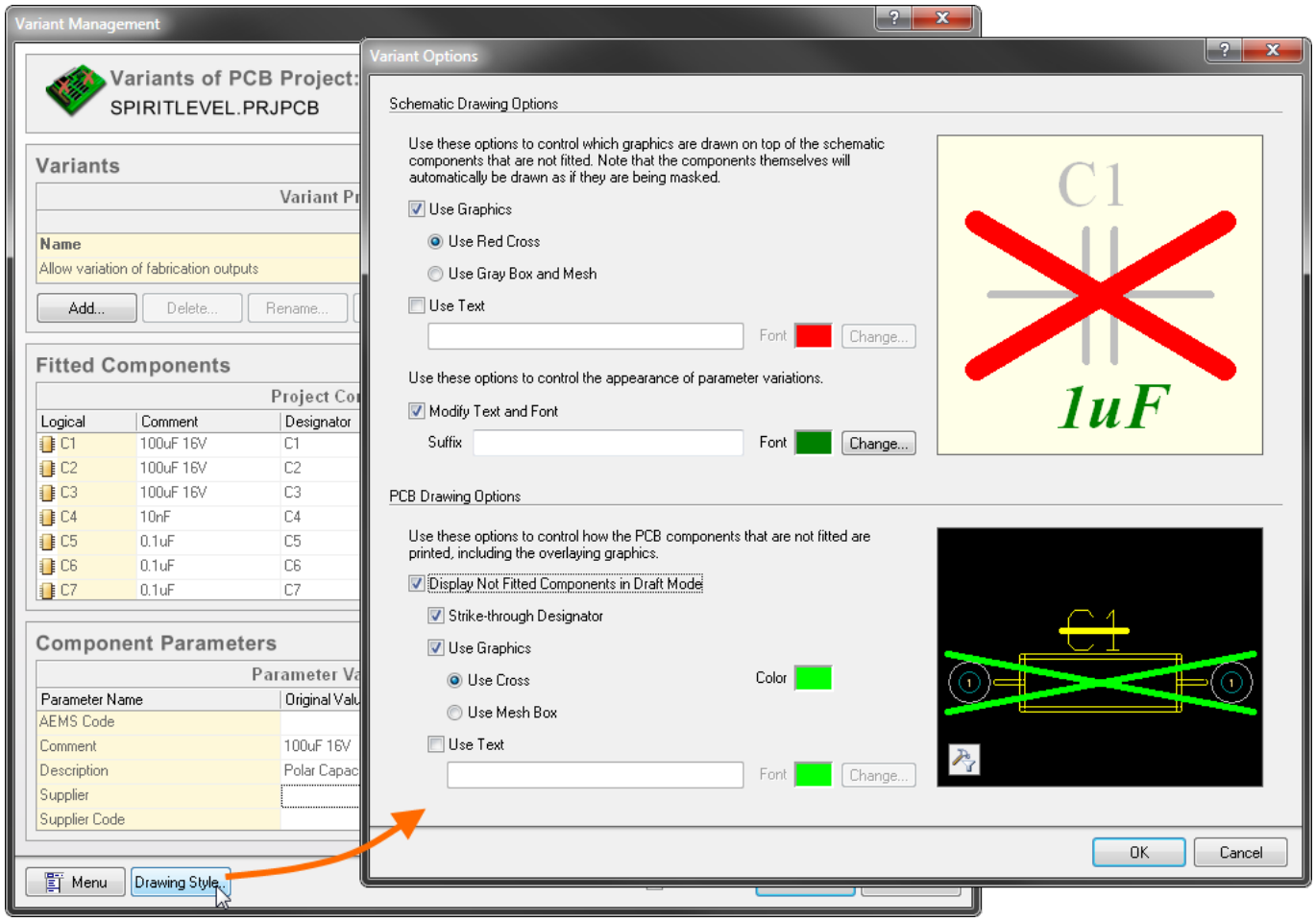
In order to access and nominate a source assembly variant for use in generating the prints, you must set the **Data Source** field for the corresponding **Schematic Prints** output generator to [Project Physical Documents]. Use of the project's physical structure means that schematics will be expanded from logical sheets to physical sheets - obtaining separate prints for each channel in a multi-channel design.



Enabling use of variants by setting the data source to use the physical structure.

Once you have configured the output generator to use the required assembly variant as its source, you will need to specify how you want the non-fitted components handled 'graphically' on the printed sheets. Options for doing this

are available in the Schematic Drawing Options region of the Variant Options dialog, as shown in the image below. Access to this dialog is made through the *Variant Management* dialog, click the **Drawing Style** button down the bottom.



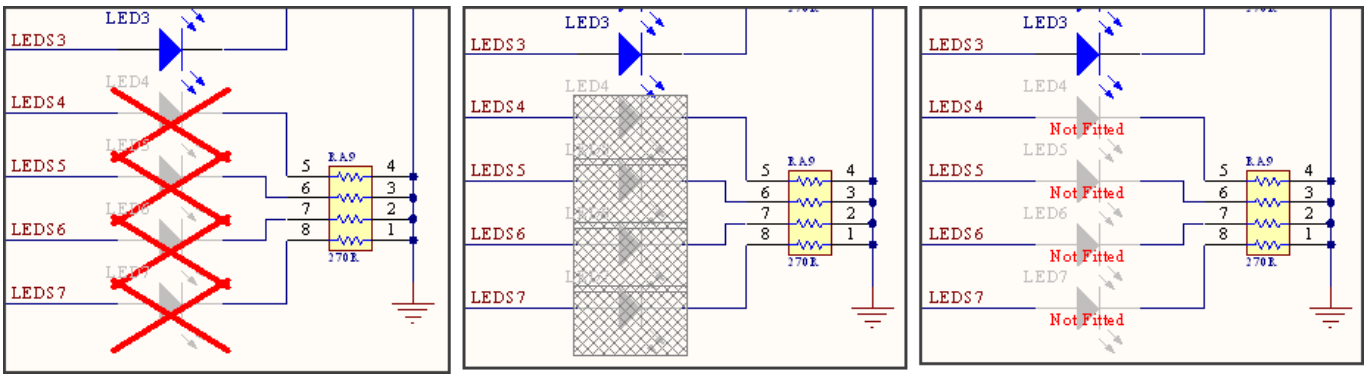
Accessing variant drawing style options.

The components that are not fitted for the source variant - as determined in the Assembly Variant Management dialog - are still drawn on the generated printouts, but drawn as though they were masked. The Variant Options dialog provides you with the ability to overlay more 'obvious' graphics or text so that you can better distinguish those components.

Use the available schematic drawing options in the dialog to define the required drawing overlay as required. The available options are:

- **Graphics** - a red cross or gray meshed box
- **Text** - enter the string that is to appear on the schematic, overlaying each component. Note the appearance of the text can be changed.

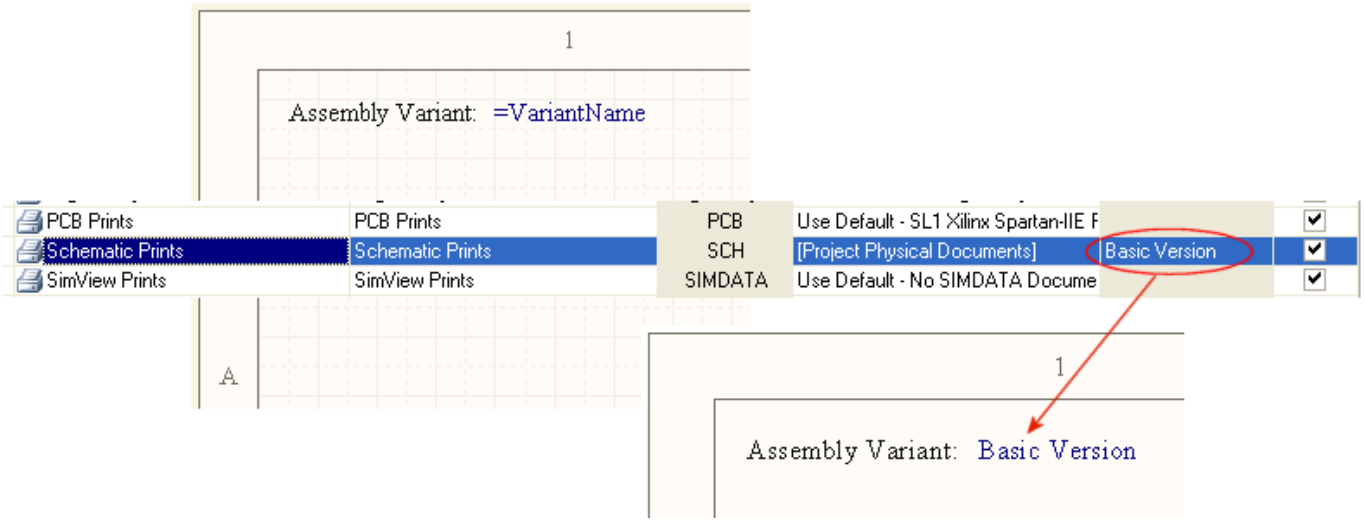
You can also combine one of the graphic options with text. As you make a choice, an example of the result will appear in the associated preview window. The image below illustrates different overlays available.



Appearance of non-fitted components using different overlay options.

Labeling the print with the variant name

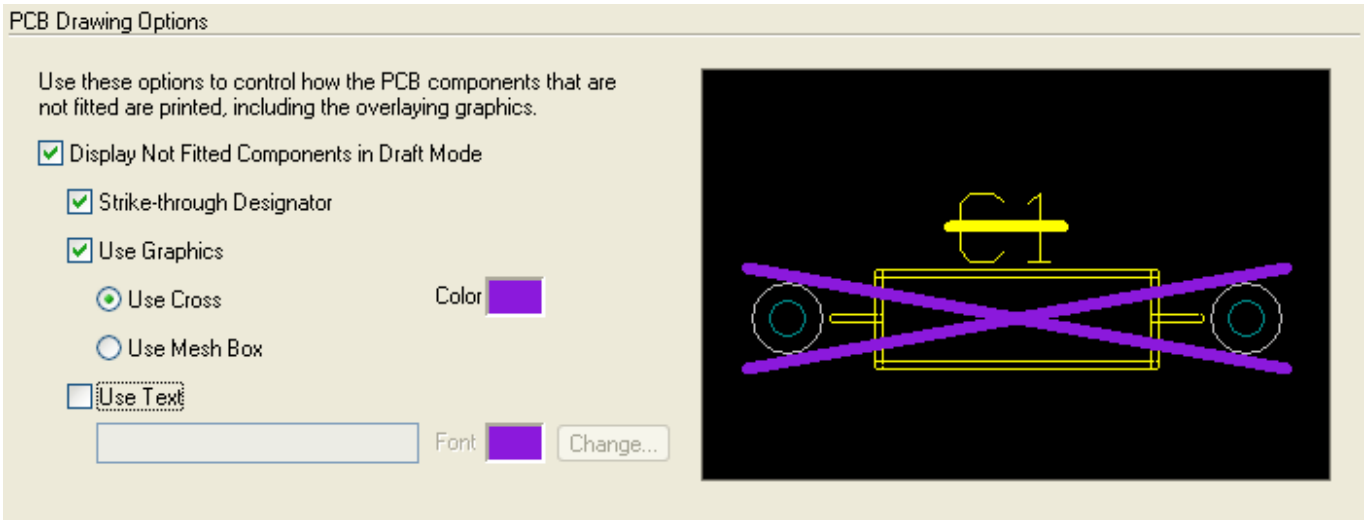
Schematic prints can be generated for a number of assembly variants, in addition to the base design itself. To identify which prints belong to which variant, a special string can be placed on the source schematic documents. Simply add the special string =VariantName to a schematic document. When the output is generated, the string will be interpreted, using the name of the variant currently assigned in the Variant field associated to the Schematic Prints output generator (in the OutputJob Editor).



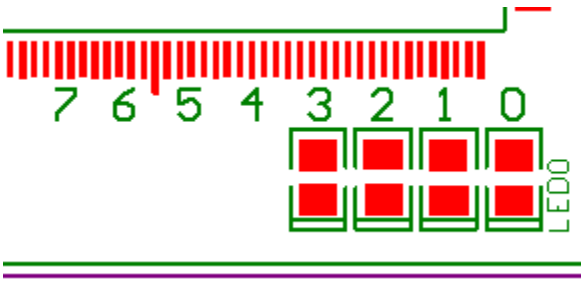
Using a special string to display variant naming on a printed schematic document.

PCB assembly drawings

You can also control the way in which non-fitted components are handled graphically on generated PCB Assembly drawings. Options for doing this are available in the PCB Drawing Options region of the Variant Options dialog, as shown below.

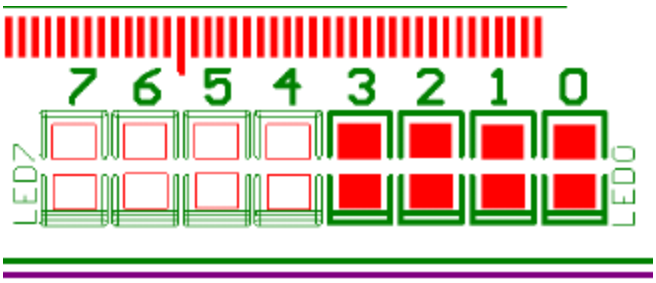


Specifying PCB drawing options in the Variant Options dialog.



Default appearance of non-fitted components.

By default, components that are not fitted for a particular assembly variant are simply not drawn on the printout. The image above illustrates this for the four LED components shown previously in the schematic images. To add these components onto the printout (in draft mode) enable the **Display Not Fitted Components in Draft Mode** option. The result is shown in the image below.

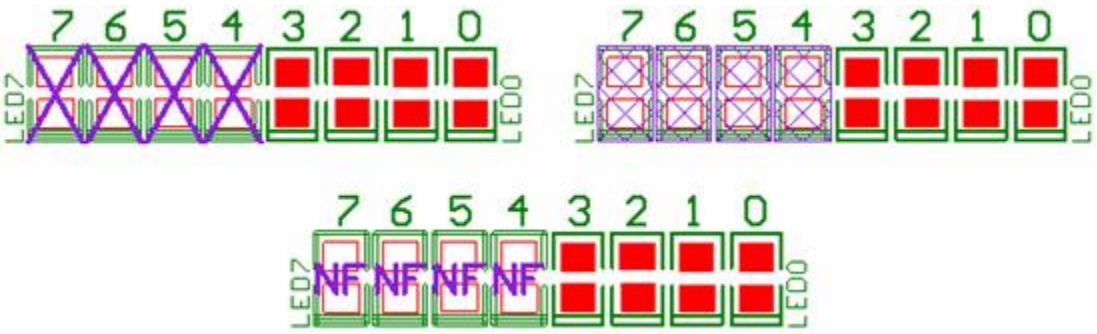


Adding non-fitted components to prints - displayed in draft mode.

Use the available PCB drawing options in the dialog to overlay more obvious graphics or text so that you can better distinguish those components. The available options include:

- **Line through the designator** - the designator is shown with a line through it.
- **Graphics** - a cross or a mesh box is drawn on top of the component overlay.
- **Text string** - a text string is placed within the component overlay, displayed the user-defined text string (eg,

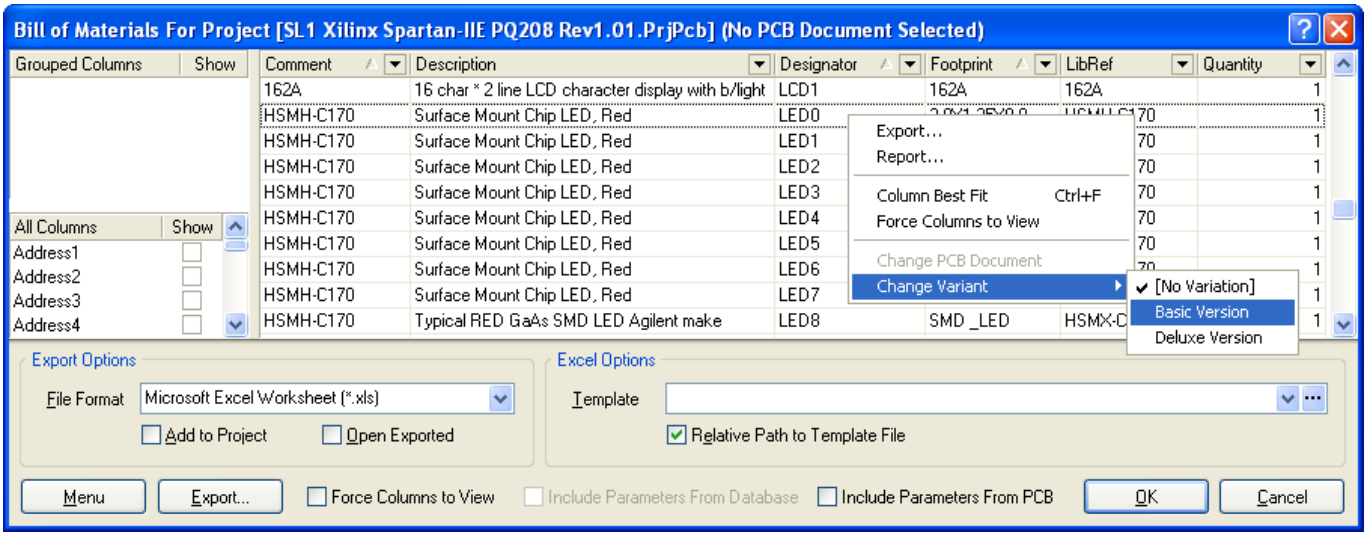
NF)



Different ways of displaying not-fitted PCB components.

Direct report generation

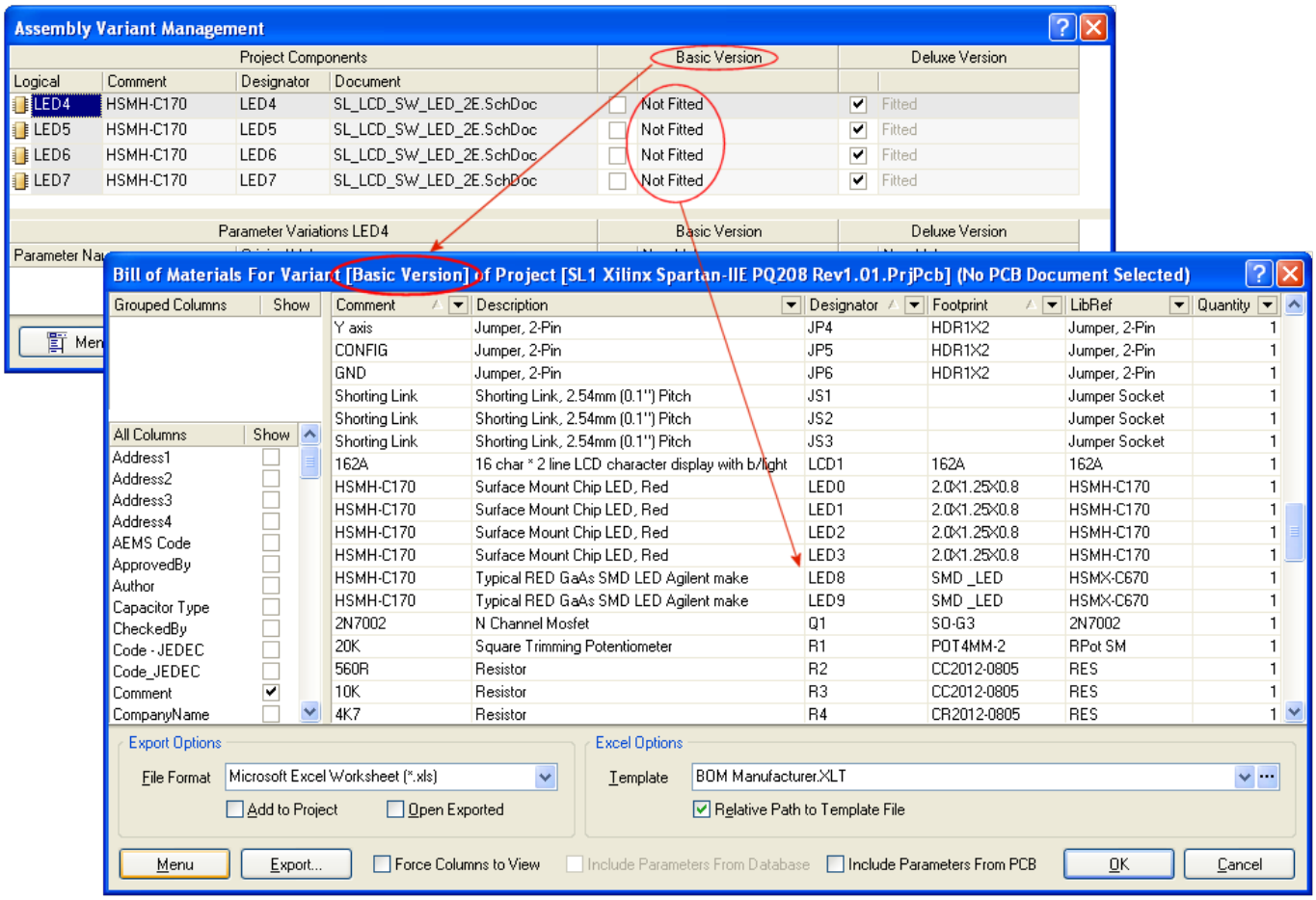
Report-based documentation that is configured using Altium Designer's Report Manager - such as a Bill of Materials or a Component Cross Reference Report - is typically defined as part of the Output Job Configuration file. As with all assembly and report based documents, you can generate them directly from within the schematic and/or PCB Editor. Whereas the majority of documents generated in this way use the base design only, with these particular documents you can choose which assembly variant you want to base the report on. **Right-click** in the *Report Manager* dialog and select the required variant from the **Change Variant** sub-menu. Set the report to use the original base design by choosing [No Variations] from the menu.



Accessing the Change Variant sub-menu.

The dialog name will change to reflect the assembly variant chosen. The data in the main grid will be updated with:

- Components defined as Not Fitted for the selected variant removed
- Parameter values updated with those from the source variant, where they vary from the base design.

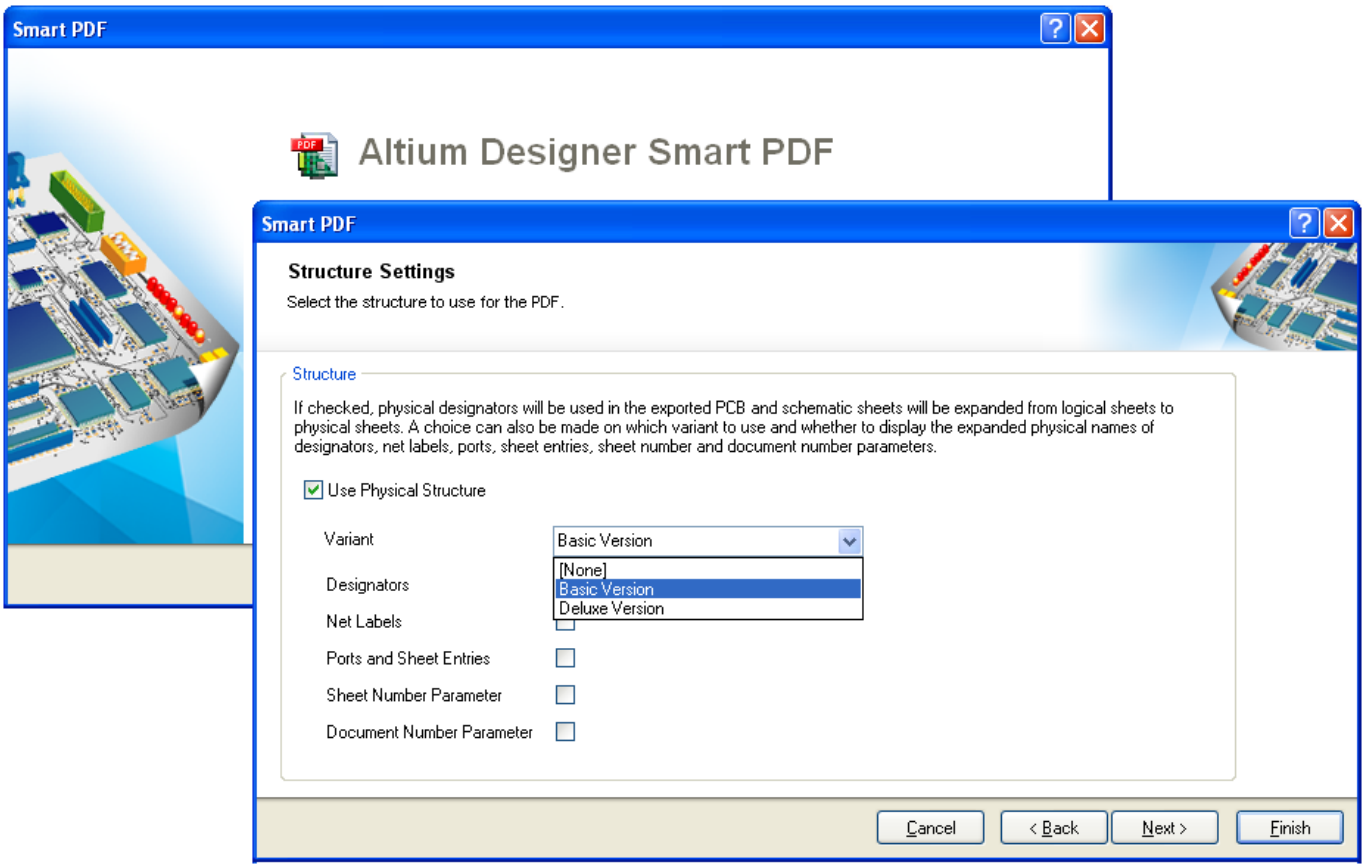


Resulting Bill of Materials based on a chosen assembly variant.

For more information on creating a Bill of Materials, refer to the Generating a Custom Bill of Materials tutorial.

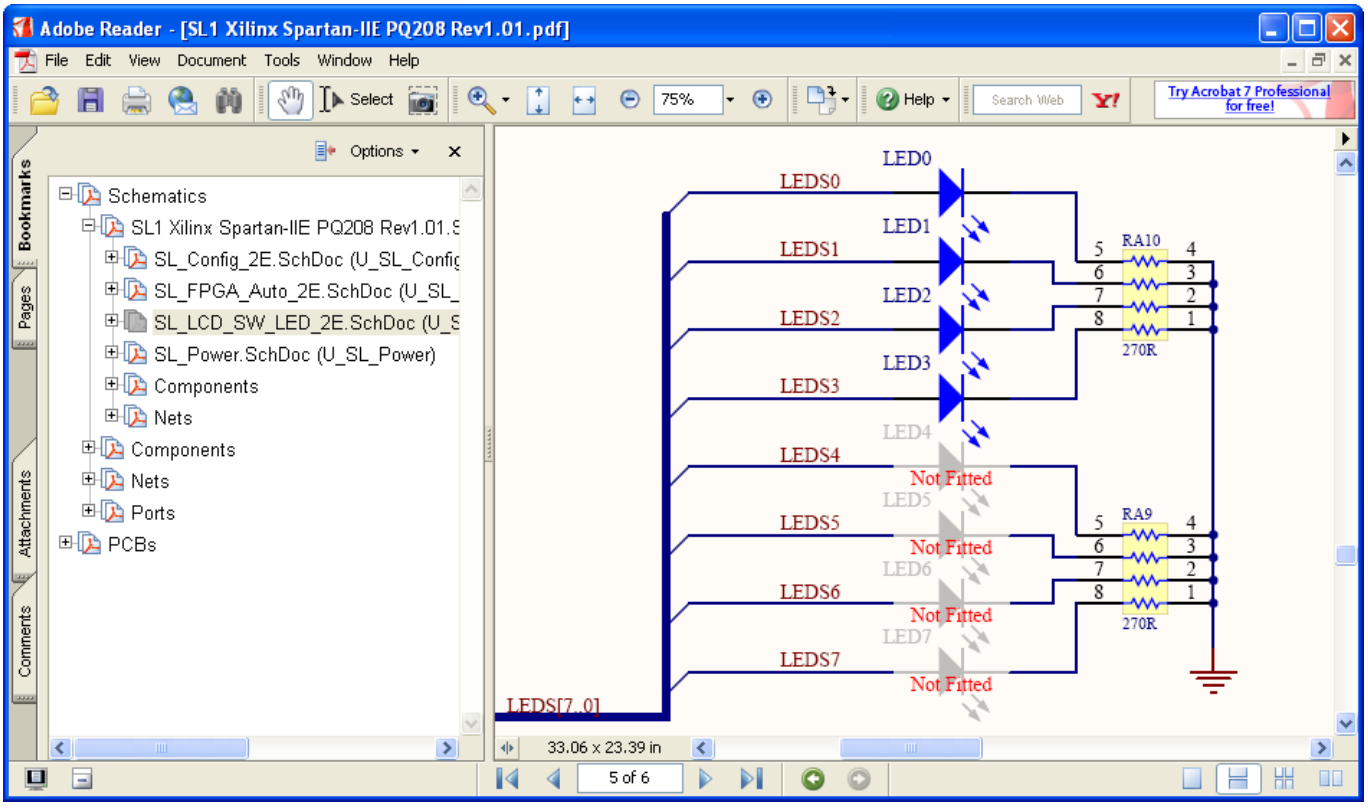
Smart PDF support

Altium Designer's Smart PDF feature also supports generation of schematic and PCB documentation based on the use of an assembly variant as the source. Use the Structure Settings page of the Smart PDF Wizard (Figure 51) to nominate the assembly variant that you wish to use. Note that the option to choose a variant will only be available when the Use Physical Structure option is enabled.

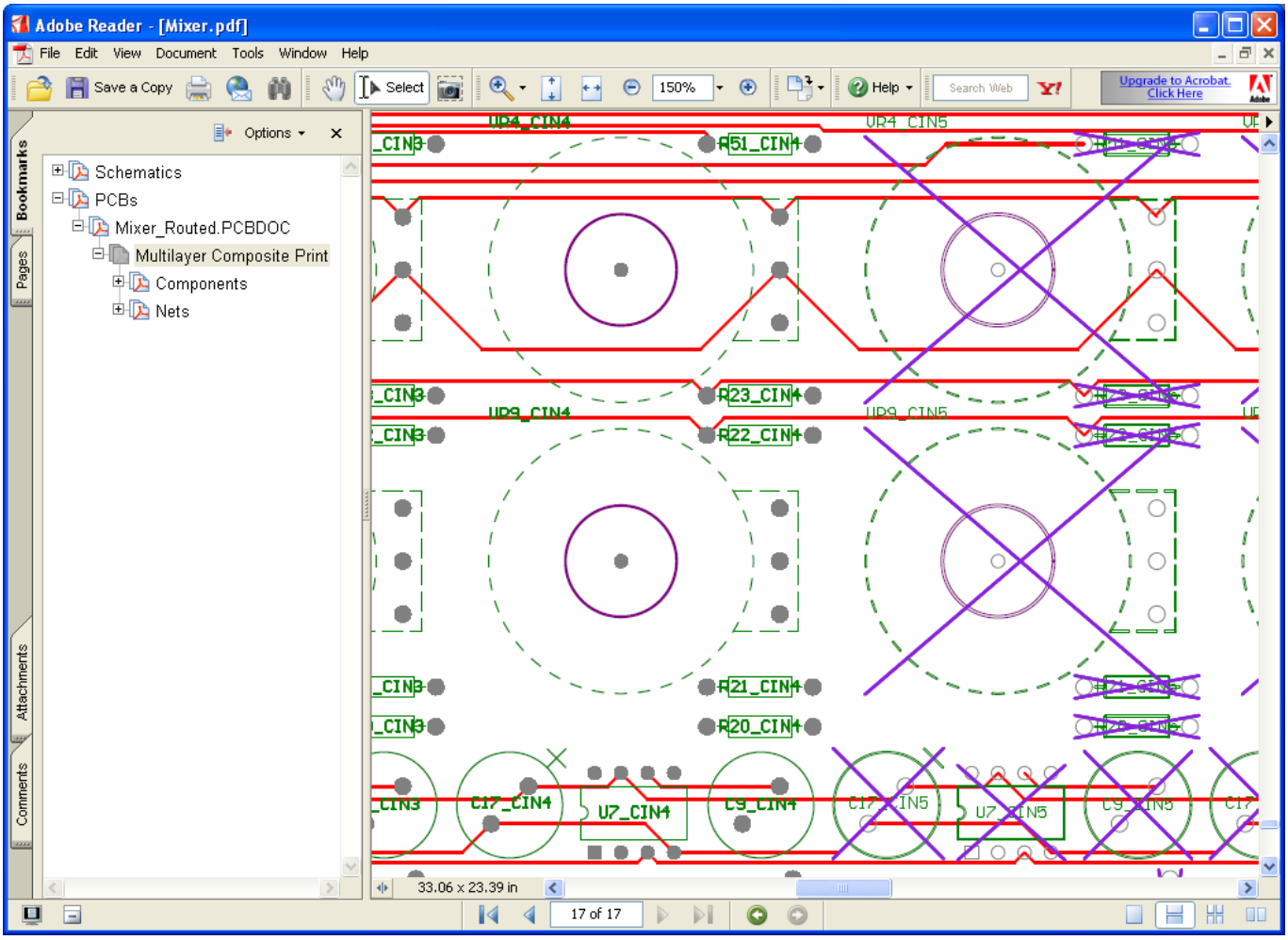


Specifying use of an assembly variant as part of the Smart PDF setup options.

As with schematic prints and assembly drawings generated from an Output Job file, you have the ability to control the graphical representation of non-fitted components on the schematic and PCB prints within the generated PDF. For more information on the options available and where to define them, refer back to the sections on Schematic prints and PCB assembly drawings, respectively.



Schematic sheet within a generated PDF with graphical overlay for non-fitted components.



PCB composite print within a generated PDF with graphical overlay for non-fitted components.

Likewise, the use of the =VariantName special string is also supported by the schematic prints in the generated PDF file. For more information, refer back to the section Labeling the print with the variant name.

For more information on generating documentation using the Smart PDF feature, refer to the document Smart PDF Generation.