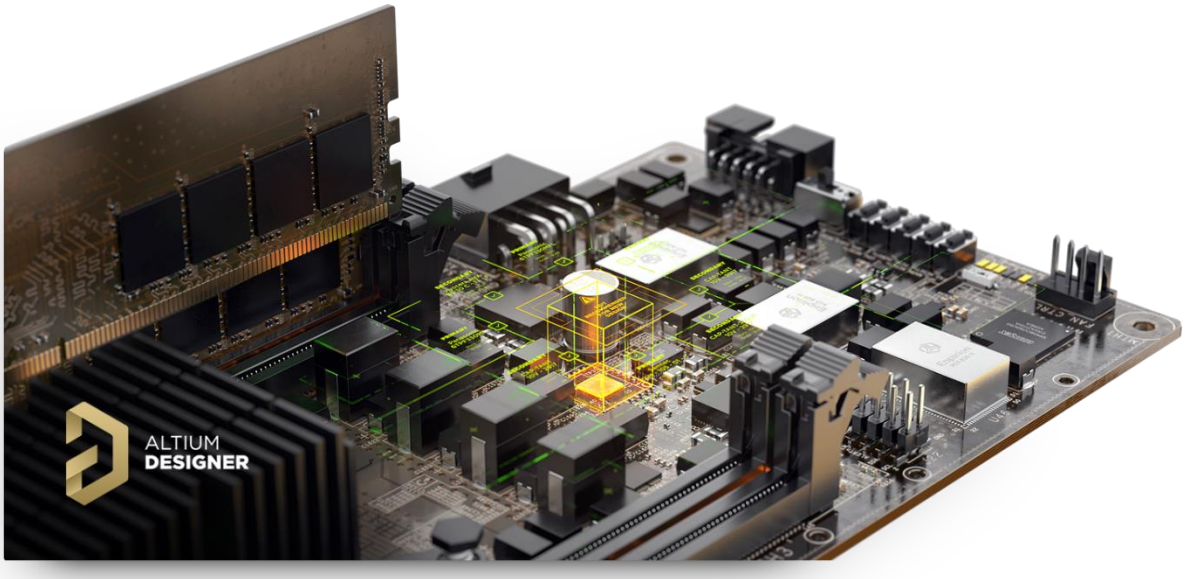

Altium Designer

Advanced Course

Module: Using Design Variants

Table of Contents

Using Design Variants

1.1 Purpose

Using variants allows you to define any number of variations of the base design. This means we can have component to be fitted, not fitted, fitted with modified component parameters, such as the component's value, or completely replace a component with an alternative part. Variants that use any of these types of variations are all referred to as Assembly Variants, as they only impact of the assembly process. All variants share the same fabricated bare board.

1.2 Shortcuts

Shortcuts when working with Using Design Variants

F1:	**Help**
C-V:	**Variant Manager**
2:	**2D Mode**
3:	**3D Mode**
G:	**Grid**
V-F	**View Fit Board**
Num 0:	**PCB 3D View – Isometric View**
Num 1:	**PCB 3D View – Top View**
CTRL+S:	**Save Document**

1.3 Preparation

1. **Close all existing projects and documents.**
2. Open the `Using Design Variants.PrjPCB` project found in its respective folder of the Advanced Training.

1.4 Variant Management

3. Open the `True Variant.SchDoc` schematic document from the *Projects* panel.
4. From the **Project** menu, select **Variants…** to open the *Variant Management* dialog.

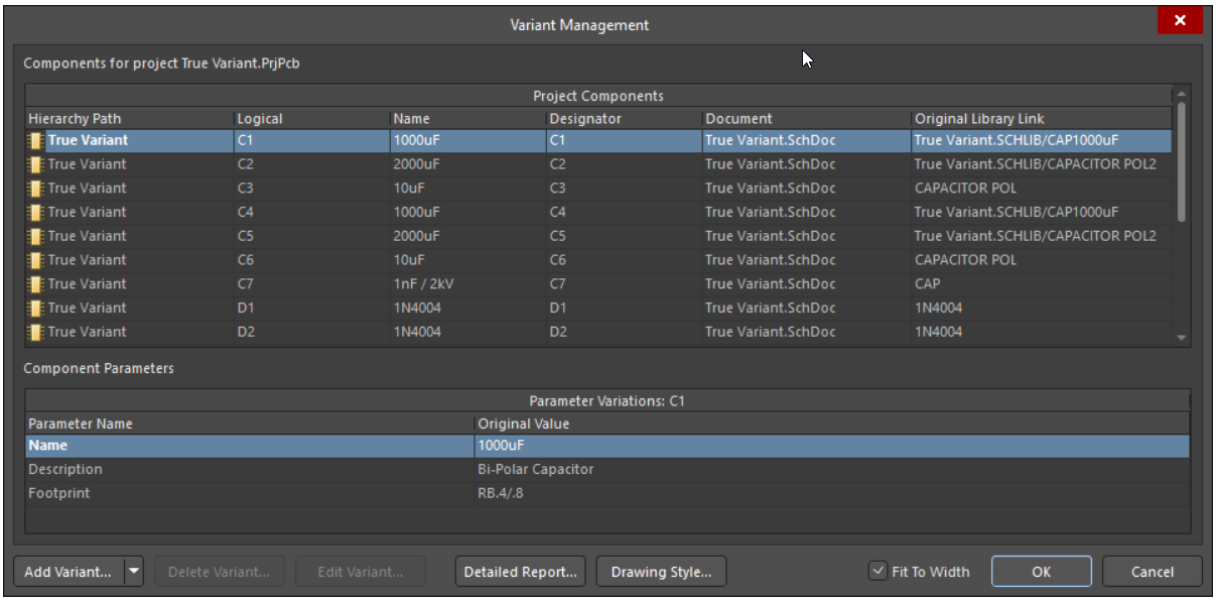

Figure 1. Variant Management Menu

5. We will be creating two new variants for this project. Use Figure 2 below as a reference.
   a) Click the **Add Variant…** button in the bottom-left corner of the window.
   b) In the *Description* field, we will name the first variant as `120VAC` as shown in Figure 2.
   c) Click the **Add…** button to add a parameter as shown in Figure 2.
   d) Add the Parameter Name: `ProjectNumber`
   e) Add the Parameter Value: `ABC123-120VAC`
   f) Click **OK** to close the *Parameter Properties* dialog.
   g) Click **OK** to create this variant. You'll notice it on the right side of the *Variant Management* window.
6. Add a second variant named `240VAC` with the parameter `ABC123-240VAC` as described in the last step, Figure 3.

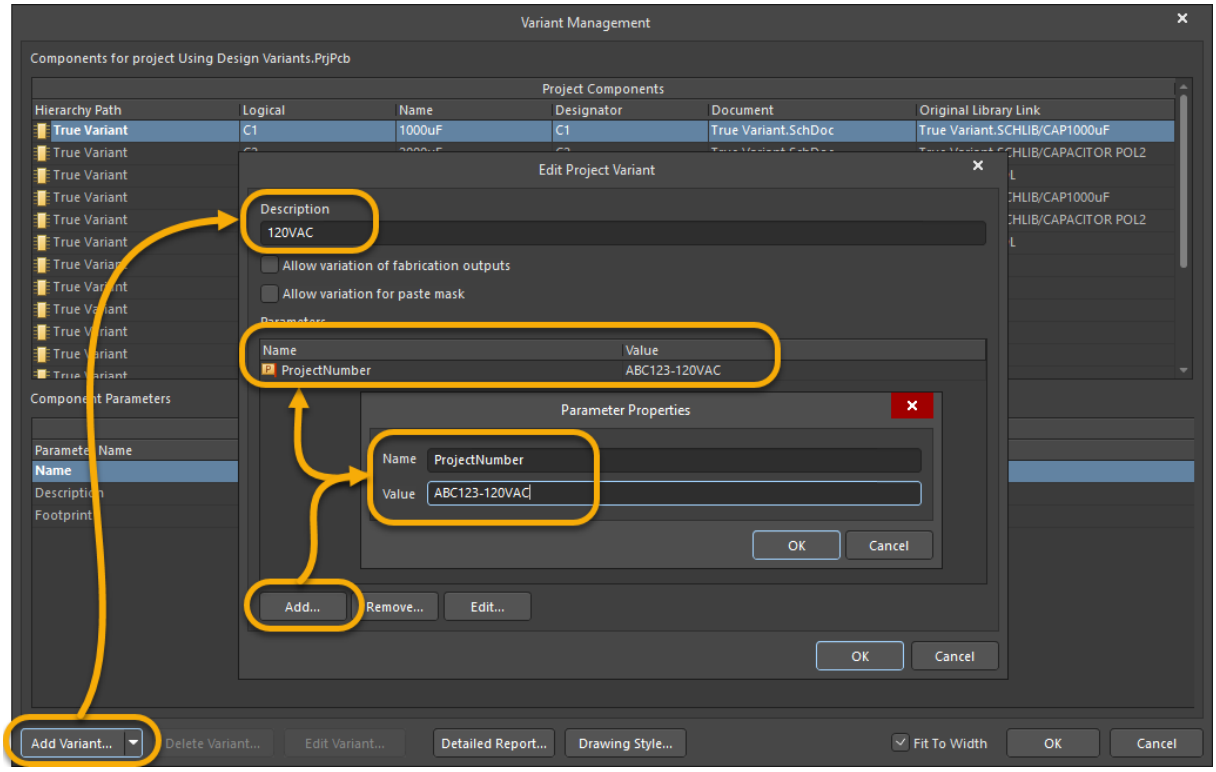

Figure 2. Creating New Variants

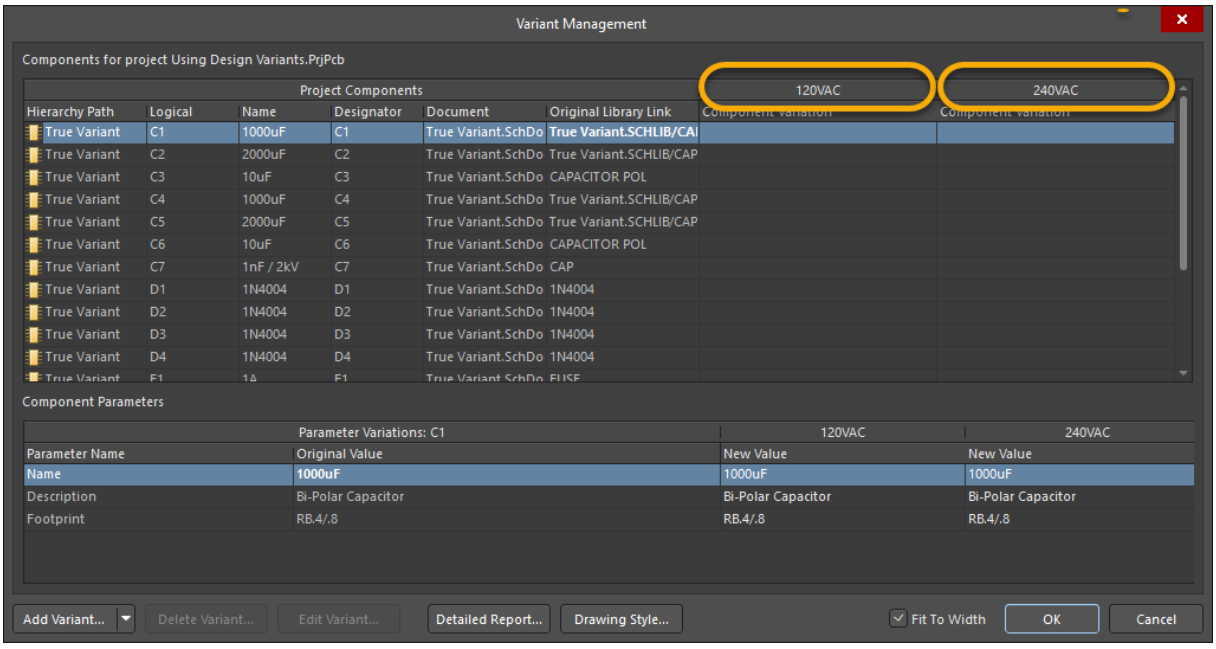

Figure 3. Two new variants

1.5 Creating Fitted and Unfitted Variants

7. We will set both `C2` and `C5` to **Not Fitted** for the `120VAC` column. To do this, click on the *Component Variation* cell for variant 120VAC. You can use Figure 4 as a reference.
8. Click on the ··· button in the cell once it appears.
9. Change the *Variation Kind* from **Fitted** to **Not Fitted**.
10. Repeat the steps above to change `C1` and `C4` to **Not Fitted** for the `240VAC` variant.

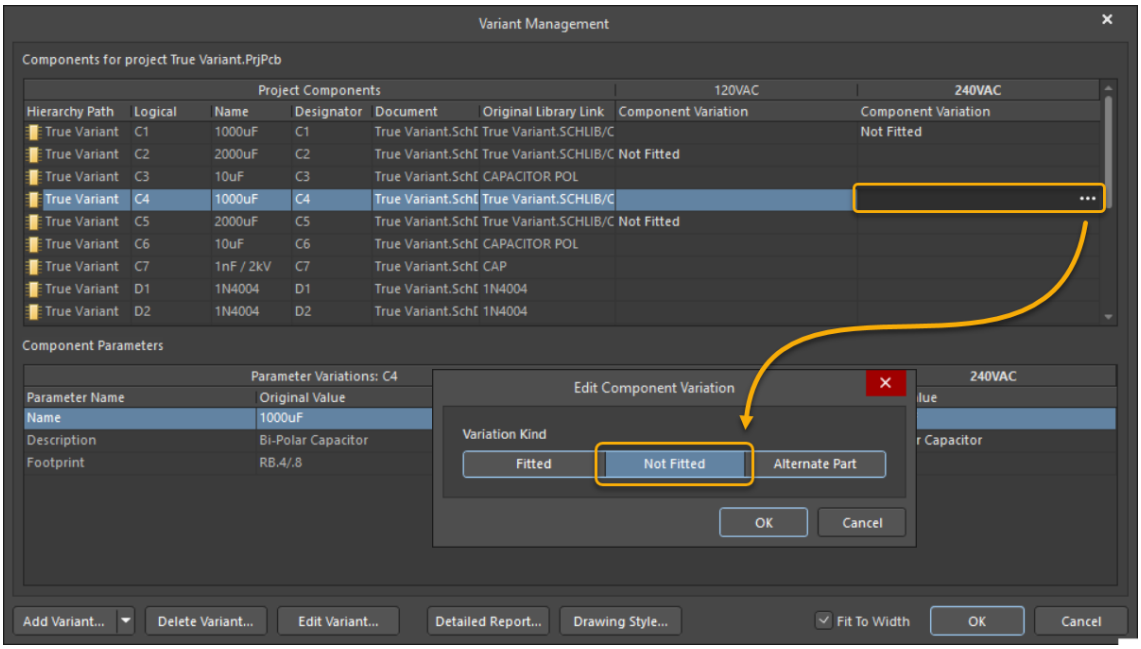

Figure 4. Setting up fitted and not fitted parts

If a component is set to **Not Fitted**, it still exists on the schematic and is transferred to the PCB, but it is removed from the appropriate output documentation, such as the BOM. You can configure how Not Fitted components are presented in the documentation.

It is also possible to select an entirely different component, as an **Alternate Part**. Since it is a different component, only one component is represented in the compiled tab of the schematic sheet. There is also the requirement that the alternate part shares the same set of pins as the base part. This is an essential requirement to ensure the connectivity remains valid when the design is compiled.

11. At the bottom of the *Variant Management* interface, select the **Drawing Style…** button to open the *Variant Options* dialog.
12. Make sure the *Schematic Drawing Options* has the **Use Graphics** checkbox enabled as shown in Figure 5 below.
13. Under the **Use Graphics** checkbox, ensure that **Use Red Cross** is selected to indicate which components are not fitted for the variant display mode.
14. Under the *PCB Drawing Options* section, ensure that **Display Not Fitted Components in Draft Mode** is selected as shown in Figure 5.
15. In the same section, enable the **Use Graphics** checkbox and select the **Use Cross** option to display all not fitted components during output generation. Notice that a preview will be shown as you make changes to the display settings.

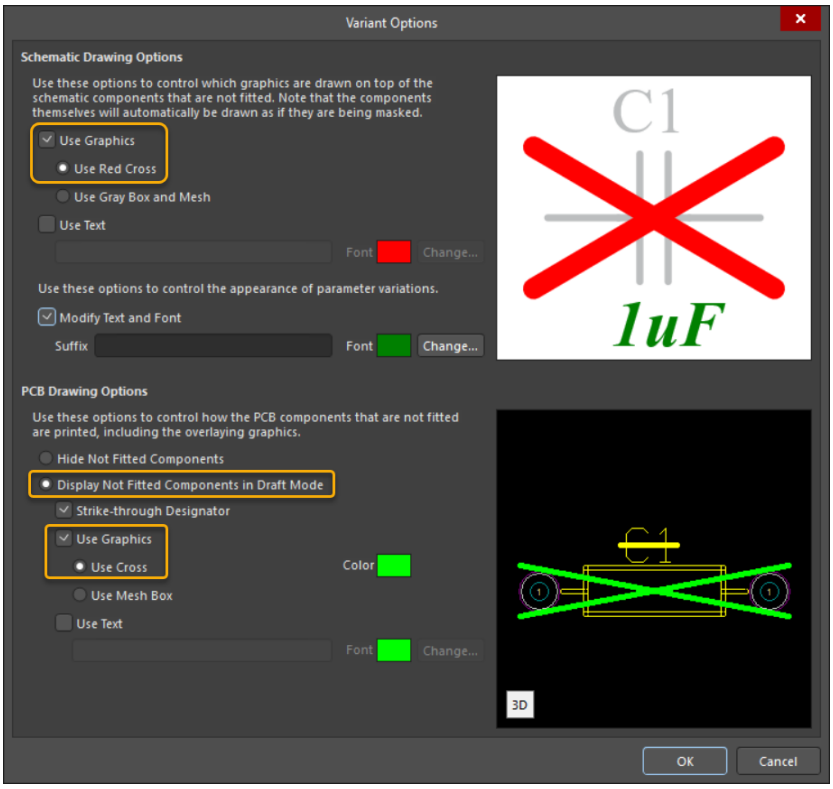

Figure 5. *Variant Drawing Options*

16. Click **OK** to accept these variant drawing options.
17. Click **OK** to close the *Variant Management* interface.

1.5.1 **Display Preview for Variants**

We can visually see the variant changes in the compiled tab of the schematic sheet.

18. From the schematic, select the *Compiled tab* named `True Variant` which is outlined in Figure 6 below.

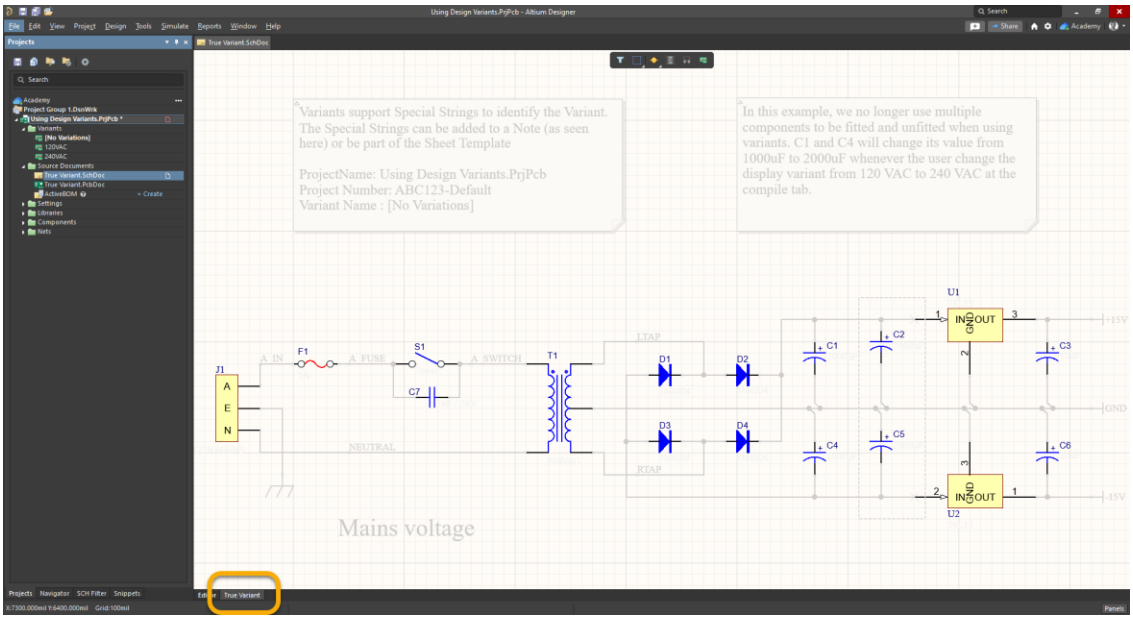

Figure 6. The Compiled Tab

19. From the *Projects* panel, you will find a *Variants* folder as shown in Figure 7.

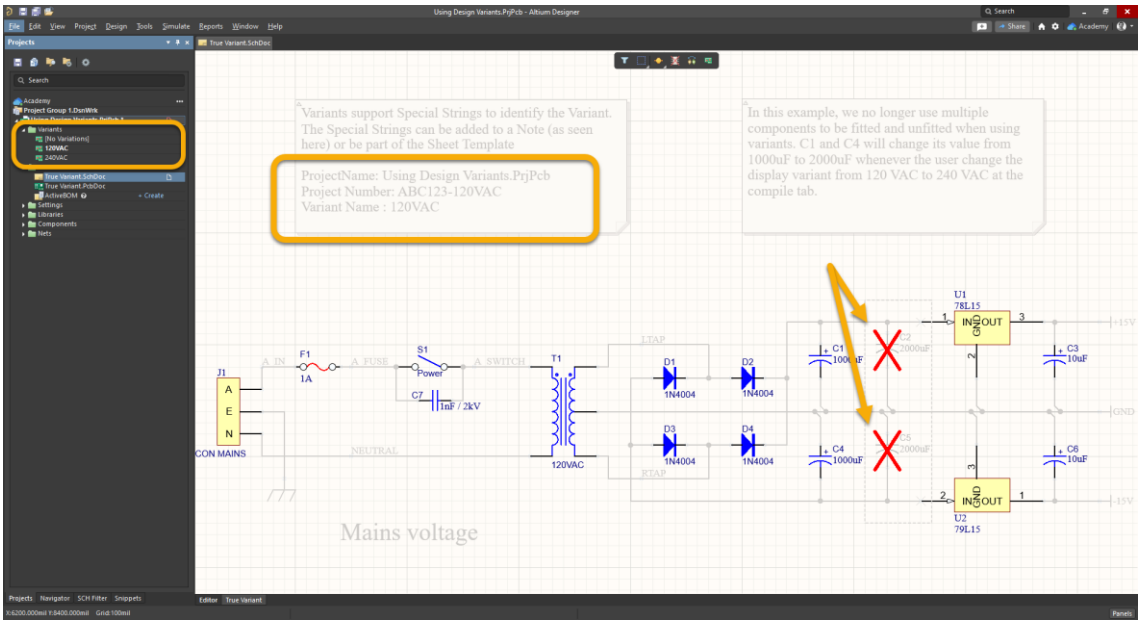

Figure 7. Selecting the Variant Menu

20. To view each variant of our design, double-click on the variant from the *Projects* panel, or right-click on one and select **Set as current.**
21. Go back and forth between the **120VAC** and **240VAC** variants to see what components we have set to **Fitted** or **Not Fitted** from the *Variant Manager*.
22. Go back and forth between the **120VAC** and **240VAC** variants to see the updated Special Strings `Variant Name` and `Project Number` in the Note object.

Variant information is not saved in the *.SCHDOC files, but directly in the *.PRJPCB project.

1.5.2 **Generating Variant Outputs**

23. From the *Projects* panel, open the `True Variant.Outjob` file that is found in the *Settings* folder as shown in Figure 8.

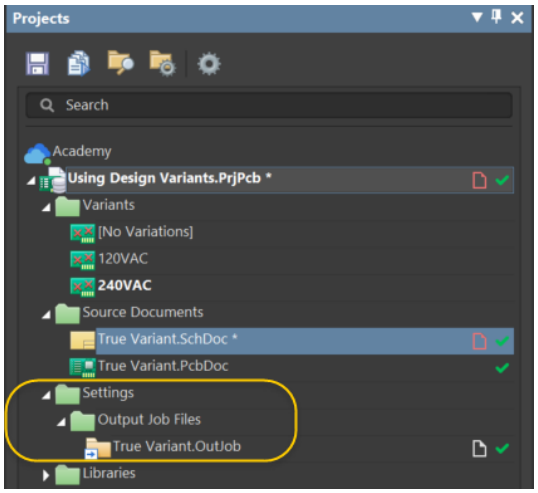

Figure 8. Output Job file from the Projects panel

24. At the top of the Output Job interface, in the *Variant Choice* section, select the **Choose a single variant for the whole output job file** option as shown in Figure 9**.** We will work with the `120VAC` variant for now.

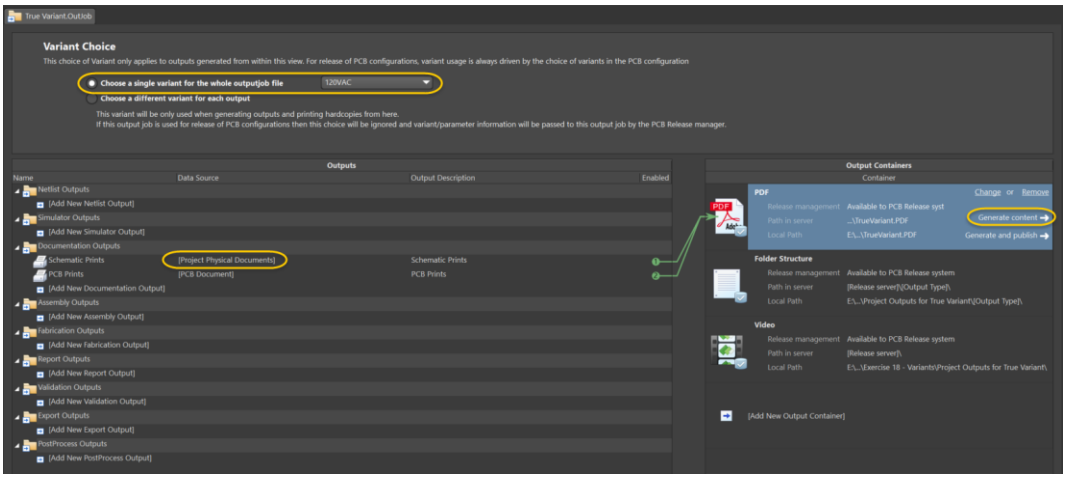

Figure 9. Generating PDF for Schematic and Assembly Drawing

25. Under the *Documentation Outputs* section, note that the *Schematic Prints Data Source* is set to **[Project Physical Documents].** This setting is required for the variations to show in the generated print.
26. Also note that the schematic and PCB prints are going to a PDF output job container, represented by the green arrow. Next, we will generate the PDF output of the selected variant.
27. Select the **Generate Content** button as shown in Figure 9 to generate a PDF printout of the Schematic and PCB Assembly Drawings.
28. Close the PDF after you've reviewed it.

For PDF documents, when multiple jobs are configured, these will automatically get concatenated into a single PDF document.

1.6 Alternative Parts

1.6.1 Creating Alternative Parts

29. Return to the `True Variant.SchDoc` schematic document.
30. From the *Editor* tab of the schematic, delete components `C2` and `C5` in the red dashed enclosure as shown in Figure 10. We will be looking at Alternate Components moving forward and we will no longer need these components.

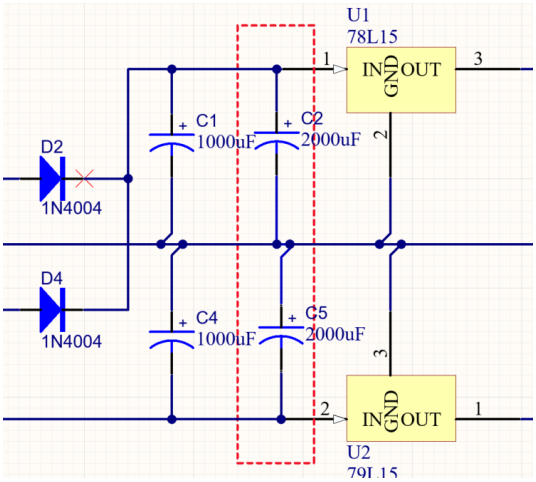

Figure 10. Delete components C2 and C5 on the schematic

31. From the **Project** menu, select **Variants…** to open the *Variant Management* interface.
32. In the `240VAC` column, select `C1` and click the ··· button to set it to **Alternate Part**.
33. Select **Replace Component…** to begin selecting the appropriate part as shown in Figure 11.

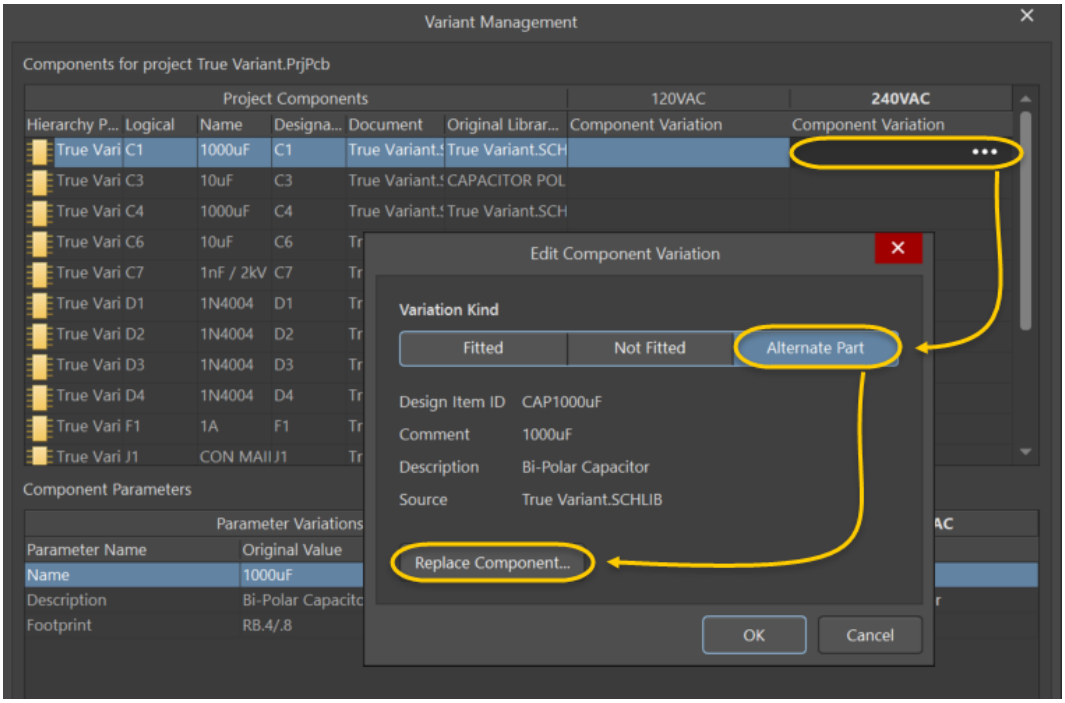

Figure 11. Selecting an Alternative Part

34. To see the libraries and components available to you, ensure to enable the triple line icon to expand the panel as shown in Figure 12.
35. Select the local `True Variant.SchLib`.
36. Select **CAP2000uF**.

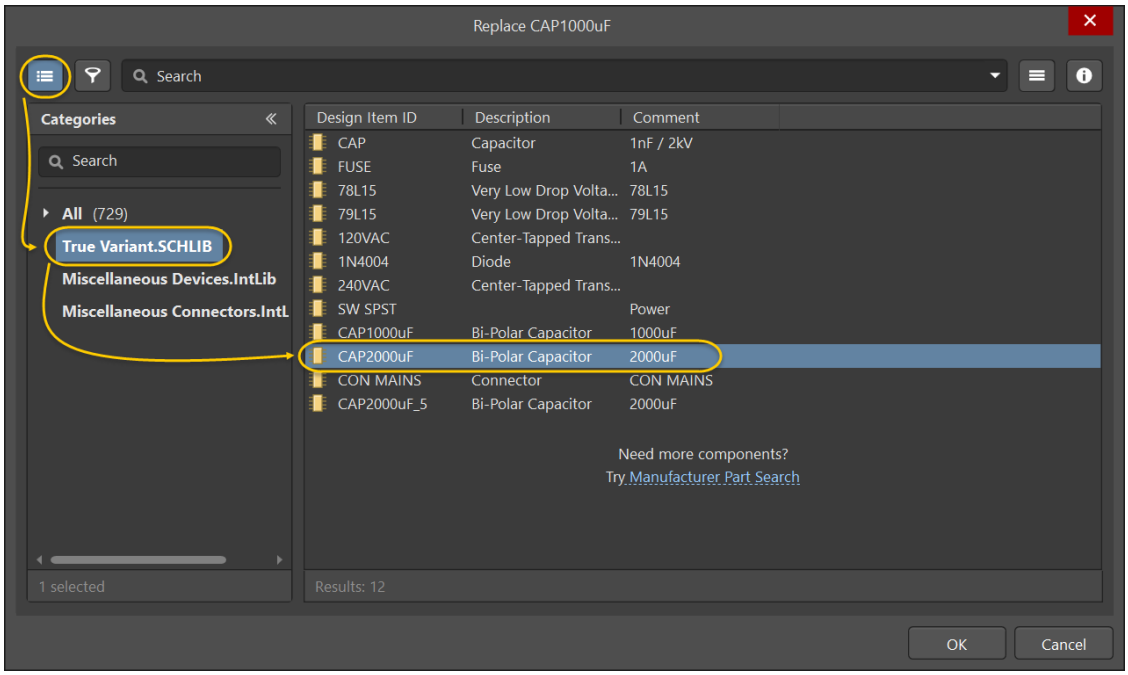

Figure 12. Alternate part selection

37. Click **OK** twice to return to the Variant Dialog.
38. Repeat the steps above to assign **CAP2000uF** as the alternate par for `C4` for `240VAC`.
39. Ensure that the `120VAC` column still has both the original `C1` and `C4` assigned to `1000uF` capacitors and the `240VAC` column has both `C1` and `C4` assigned to `2000uF` as shown in Figure 13.

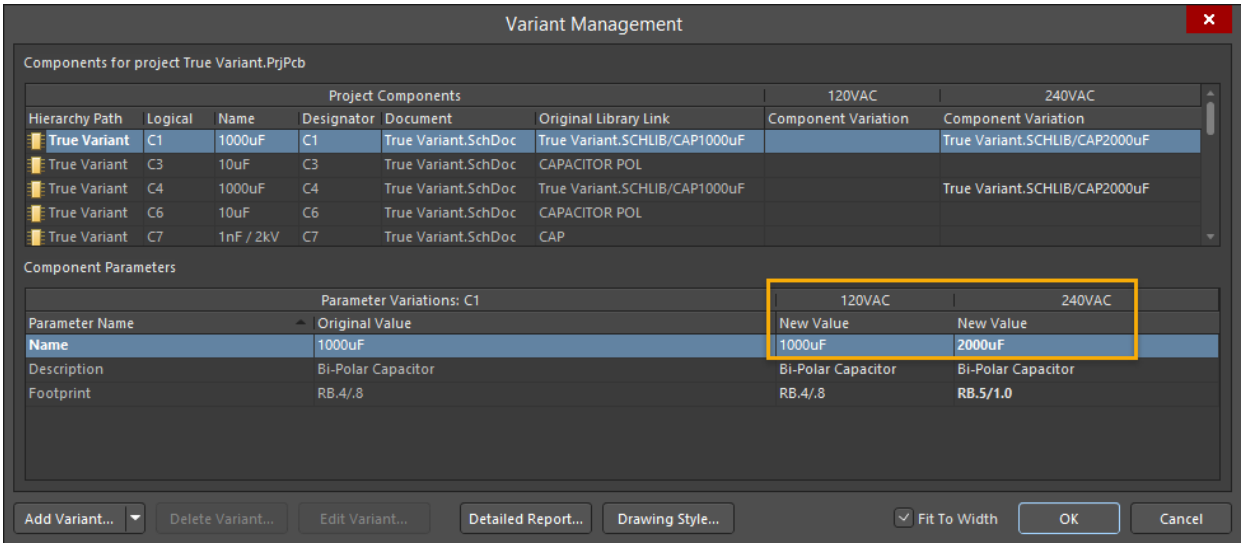

Figure 13. Alternate Part Assignment

40. Click **OK** to close the *Variant Management* dialog.

1.6.2 **Viewing Alternative Parts**

41. Validate the project by going to the **Project** menu and select **Validate PCB Project Using Design Variants.PrjPCB.**
42. Select the *Compiled tab* named `True Variant` located at the bottom left of your workspace.
43. Switch between the `120VAC` and the `240VAC` variants in the *Projects* panel. Observe the values of `C1` and `C4` depending on which variant is selected. You can see the difference in Figure 14.

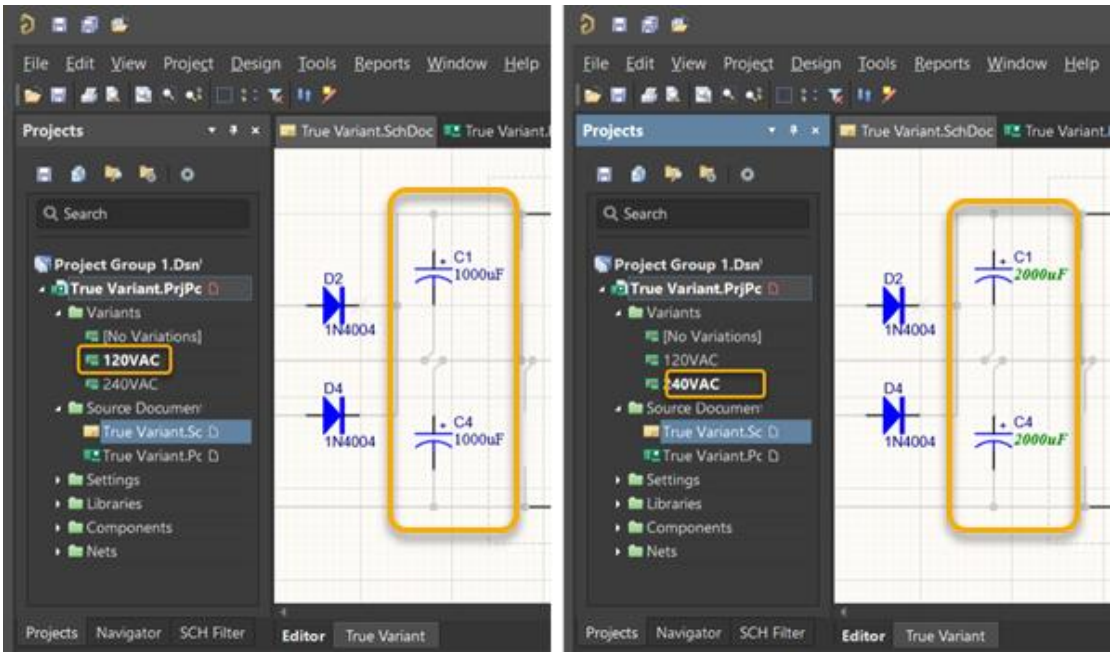

Figure 14. 120VAC and 240VAC Alternate Part Variant

1.7 How Design Variants affect the PCB

44. Open the PCB file `True Variant.PcbDoc`.
45. From the **Design** menu, select **Import Changes from Using Design Variants.PRJPCB**. This will import the alternate components into the PCB. In the *ECO* dialog, you will see the original component and the alternate part both added to the PCB a shown in Figure 15. This can be confusing at first, but this is the intended function.

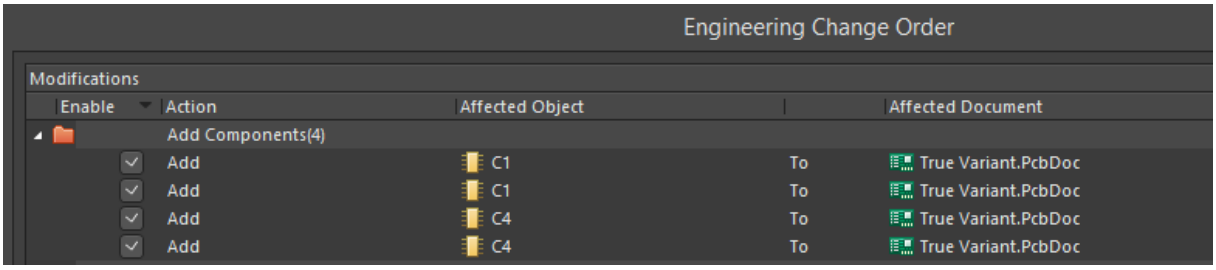

Figure 15. Both the base part and alternate parts will be imported into the PCB

46. Select the **Execute Changes** button to begin importing all changes to the PCB.
47. Click **Close** to exit the *Engineering Change Order* window.

You will have duplicate footprints of C1 and C4 in this PCB project. These components will be enabled or disabled depending on which variant mode you have selected.

48. Go to the 3D View mode by pressing the **3** key.
    a) Switch between the 120VAC and 240VAC modes in the *Project* panel.
    b) Confirm that both 120VAC and 240VAC variant modes will display the 3D body of its respective variant as shown in Figure 16

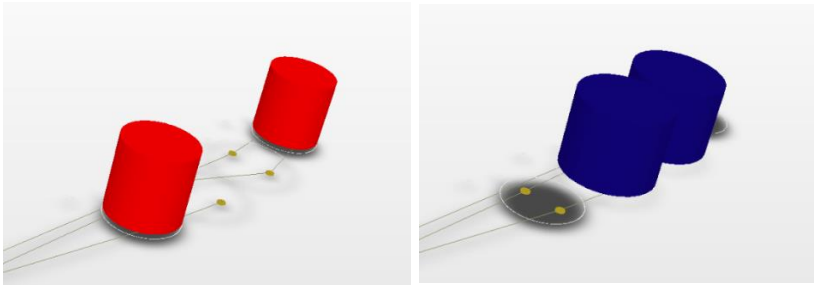

Figure 16. 3D View of Alternate Parts 120VAC and 240VAC

49. Switch back to 2D mode by hitting the **2** key.
50. Position the new components into the PCB, overlapping the duplicate footprints.

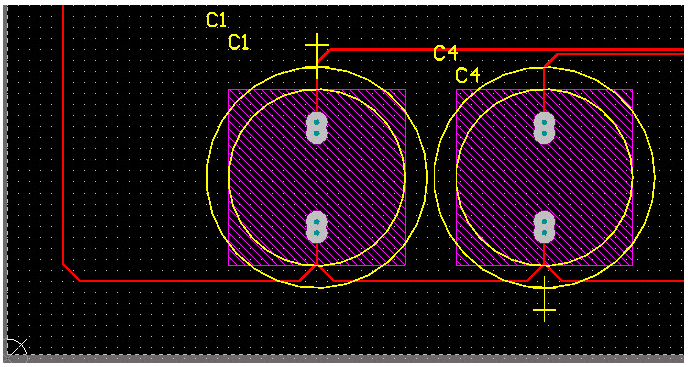

Figure 17. Overlapped Alternate Part Footprints in 2D View

51. Go back to the 3D View by pressing the **3** key.
52. Switch between the 120VAC and 240VAC modes in the *Projects* panel. Confirm that both 120VAC and 240VAC variant modes will display the 3D as shown in Figure 18.

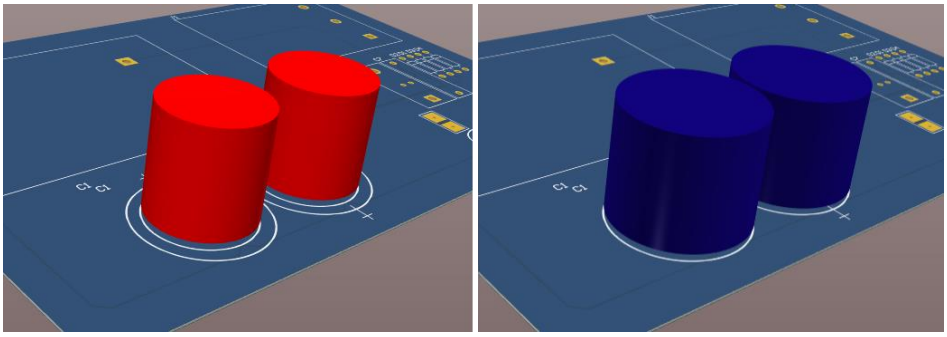

Figure 18. 3D View of the two Variants

53. Feel free to save your modifications.
54. **Close the project and any open documents.**

Congratulations on completing module

Using Design Variants

from the

Altium Designer Advanced Course

Thank you for choosing Altium Designer