Altium Designer

Advanced Training with Altium 365

Using Design Variants

Software, documentation and related materials:

Table of Contents

Using Design Variants

1 Purpose

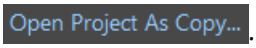
With variants, you create various versions of the base design. You can include or exclude components, adjust their parameters, or replace them with alternatives. These changes are called Assembly Variants because they only affect assembly, while all variants use the same bare board.

2 Shortcuts

Shortcuts used 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

3 Preparation

1. Close all existing projects and documents.
2. Next, create a Copy of the Training Project: `Using Design Variants`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button  .
5. Navigate to the predefined Training Project `Using Design Variants (Top\Projects\Altium Designer Advanced Training Course\...)`.
6. Select **Open Project as Copy...** 

.
7. In the new dialog *Create Project Copy:*
   a) Add your name to the project name: `Using Design Variants – [Your Name]`.
   b) Add a description: `Altium Advanced Training – [Your name]`.
   c) Open the *Advanced* section.
   d) Select the **Ellipsis Button**  from the *Folder* configuration to open the *Choose Folder* dialog.
      i) Select the folder with your name: `Project\For Attendees\[Your name]`.
      ii) Select **OK**.
   e) Change the **Local Storage** path if needed.
   f) Select **OK** to create the copy.
8. Wait until Altium Designer creates the copy of the project and opened the Project for you in the *Projects* panel, this may take up to 1 minute.

Hint: For details how to copy the predefined training project, see module *03 Getting started - Opening a Project*.

4 Variant Management

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

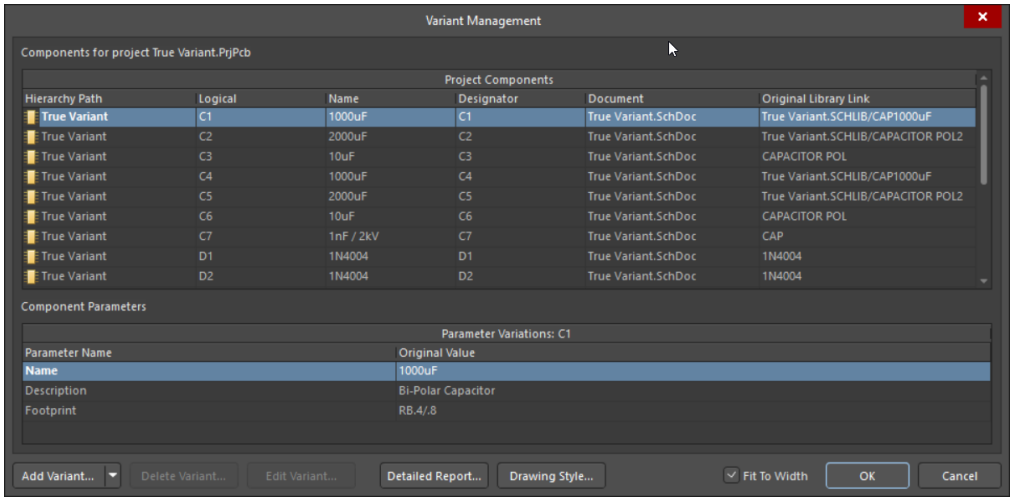


Figure 1. Variant Management Menu

11. You will create two new variants for this project. Use Figure 2 below as a reference.
    a) Select the **Add Variant...** button in the bottom-left corner of the window.
    b) In the *Description* field, name the first variant as `120VAC`, Figure 2.
    c) Select the **Add...** button to add a parameter.
    d) Add the Parameter Name: `ProjectNumber`.
    e) Add the Parameter Value: `ABC123-120VAC`.
    f) Select **OK** to close the *Parameter Properties* dialog.
    g) Select **OK** to see a new variant on the right side of the *Variant Management* window.

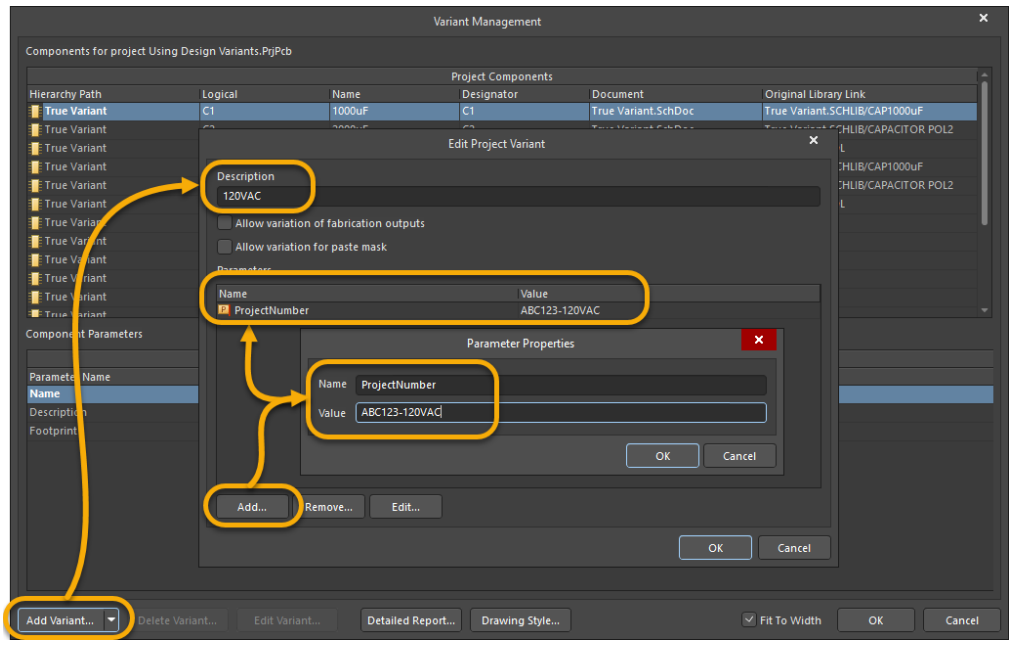


Figure 2. Creating New Variants

12. Add the second variant `240VAC` with the parameter name `ProjectNumber` and the value `ABC123-240VAC` to see the result illustrated by Figure 3.

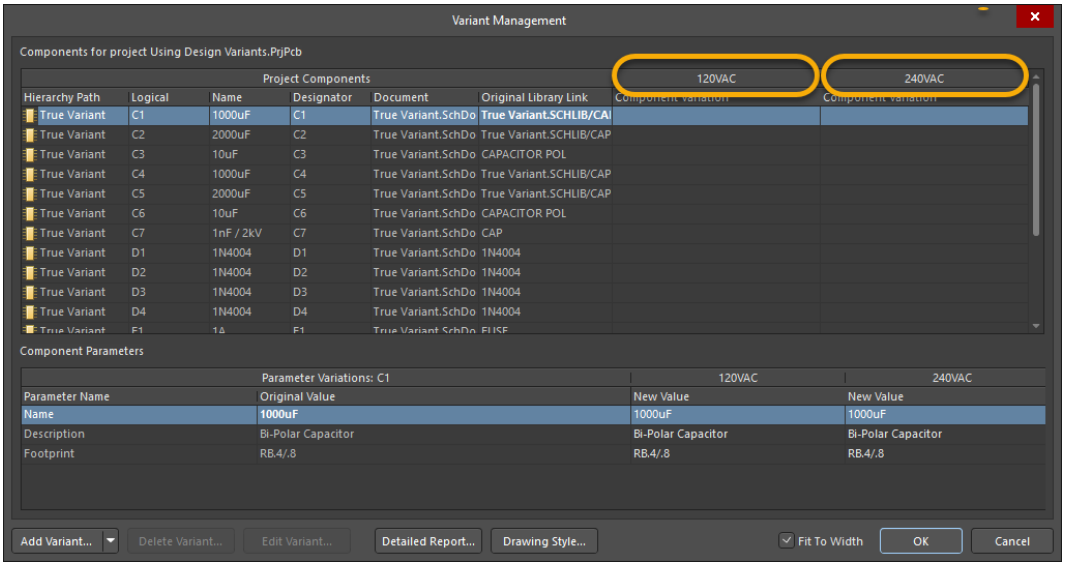
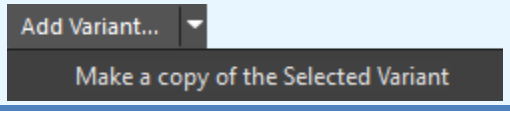


Figure 3. Two new variants

Hint: Instead of creating a new variant from scratch, you can make a copy of the Selected Variant and edit the new Variant.



5 Creating Fitted and Unfitted Variants

13. You will set both C2 and C5 to **Not Fitted** for the 120VAC column. To do this, select the *Component Variation* cell for variant 120VAC. You can use Figure 4 as a reference.
14. Select the ellipsis button ··· in the cell once it appears.
15. Change the *Variation Kind* from **Fitted** to **Not Fitted**.
16. Repeat the steps above to change C1 and C4 to **Not Fitted** for the 240VAC variant.

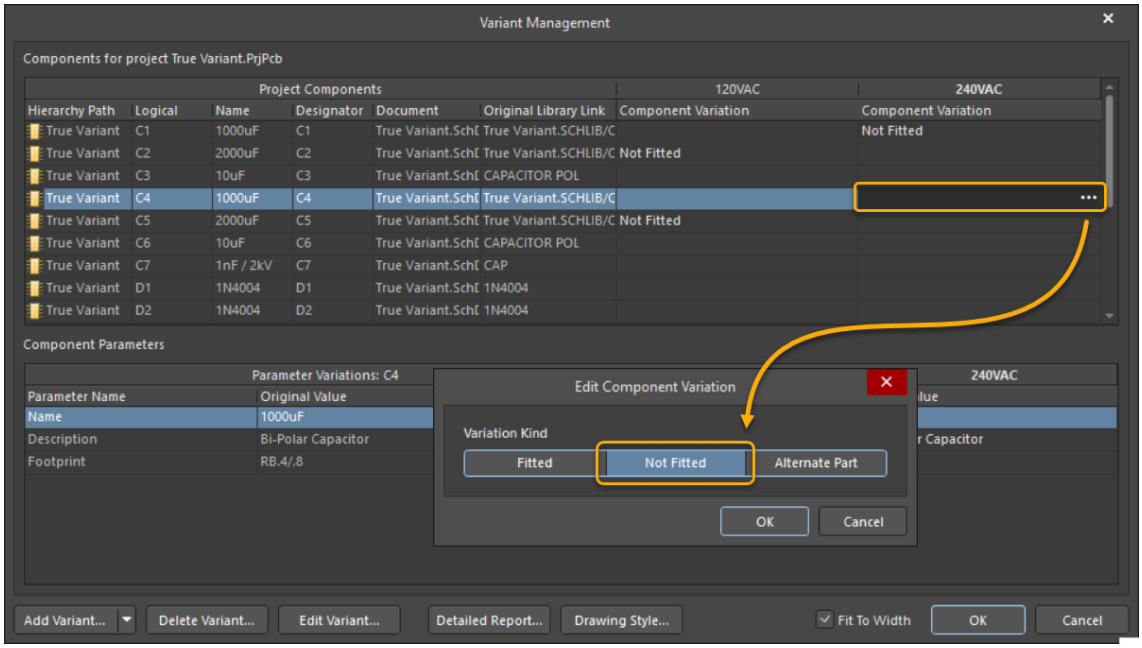


Figure 4. Setting up fitted and not fitted parts

Note: If a component is marked as **Not Fitted**, it remains on the schematic and is included in the PCB but is excluded from output documentation like the BOM. You can adjust how Not Fitted components appear in the documentation.
Alternatively, you can choose a different component as an **Alternate Part**, which must have the same pin configuration as the original to ensure proper connectivity in the design.

17. At the bottom of the *Variant Management* interface, select the **Drawing Style...** button to open the *Variant Options* dialog.

18. Make sure the *Schematic Drawing Options* has the **Use Graphics** checkbox enabled, Figure 5.
19. Under the **Use Graphics** checkbox, ensure that **Use Red Cross** is selected to indicate which components aren't fitted for the variant display mode.
20. Under the *PCB Drawing Options* section, ensure that **Display Not Fitted Components in Draft Mode** is selected, Figure 5.
21. In the same section, check the **Use Graphics** box and select the **Use Cross** option to show all not fitted components in the output. A preview will update as you adjust the display settings.

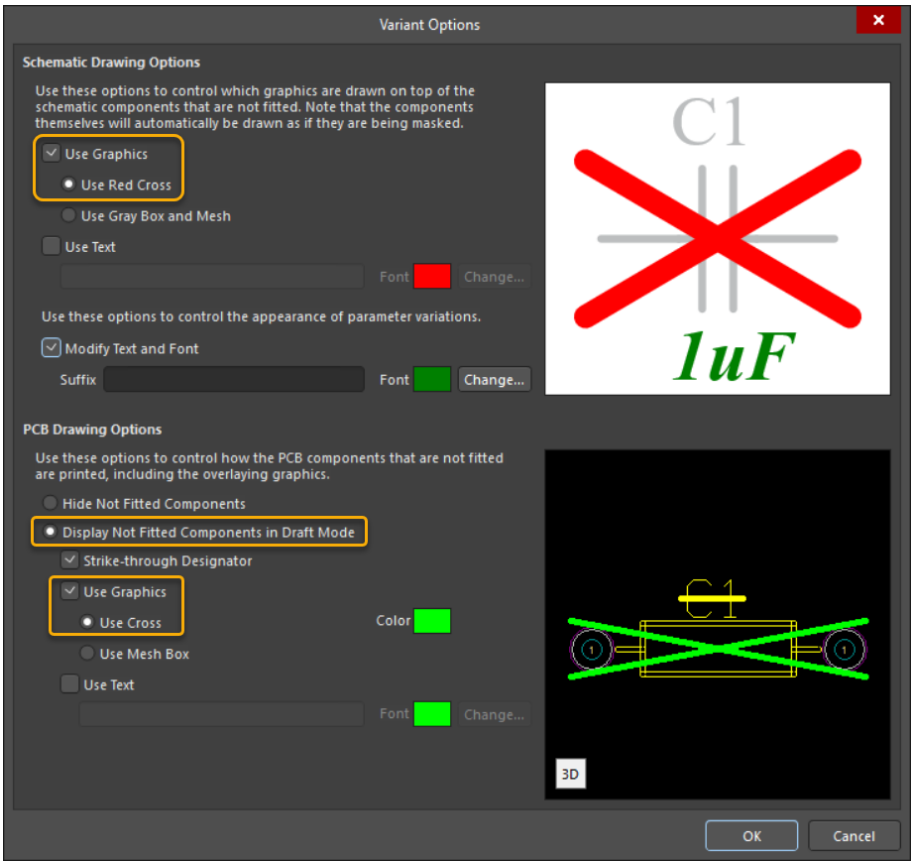


Figure 5. Variant Drawing Options

22. Select **OK** to accept the variant drawing options.
23. Select **OK** to close the *Variant Management interface.*

5.1 Displaying Preview for Variants

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

24. From the schematic, select the *Compiled tab* named `True Variant`, outlined in Figure 6.

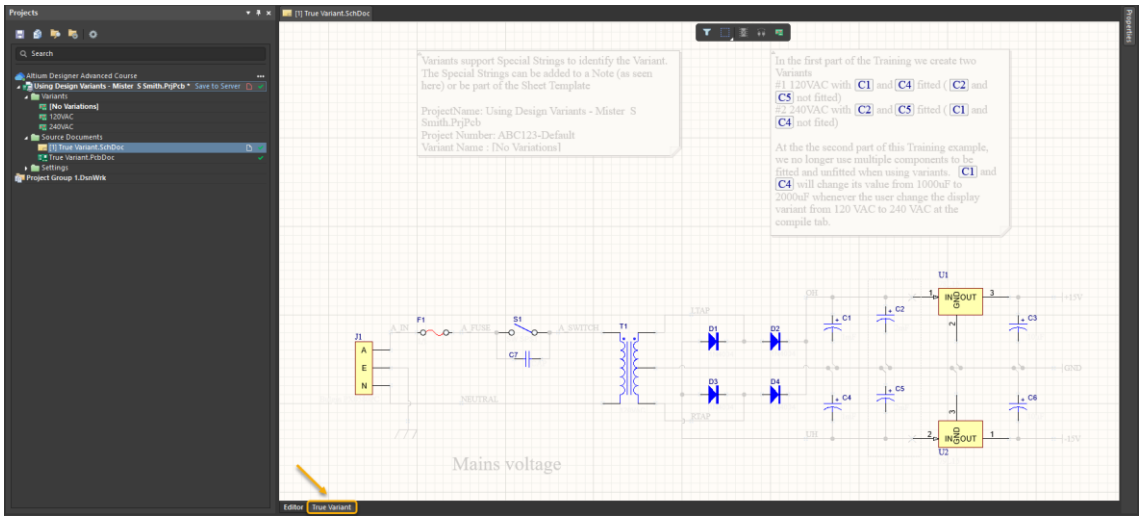


Figure 6. The Compiled Tab selected

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

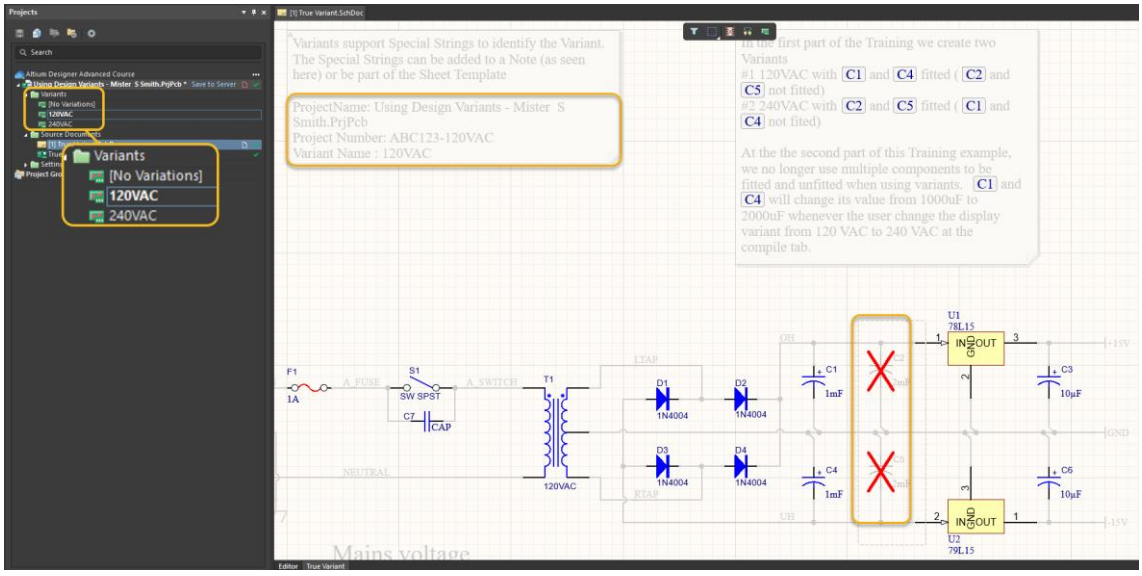


Figure 7. Selecting the Variant Menu

26. To view each variant of your design, double-click on the variant from the *Projects* panel, or right-click on one and select **Set as current.**
27. Go back and forth between the **120VAC** and **240VAC** variants to see what components you've set to **Fitted** or **Not Fitted** from the *Variant Manager*.
28. 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.
29. Select the **120VAC** variant.

> Note: Variant information isn't saved in the *.SCHDOC files, but in the *.PRJPCB project.

5.2 Updating the PCB

Next, you will update the PCB with the modifications. The PCB is preconfigured with the four Capacitors missing: `C1`, `C2`, `C4`, and `C5`.

30. Open the PCB file `True Variant.PcbDoc`.
31. From the **Design** menu, select **Import Changes from Using Design Variants.PRJPCB**.
32. In the *ECO*, select **Execute Changes** and **Close.**

> Hint: Don't save the PCB. In section **6 Alternative Parts**, you will update and save the PCB with the alternative part variant information.

33. The four capacitors are added to the PCB, as seen at the lower right corner of the PCB. You will place the capacitors later in this module.
34. Change to the 3D View to see the 120VAC variant, Figure 8.

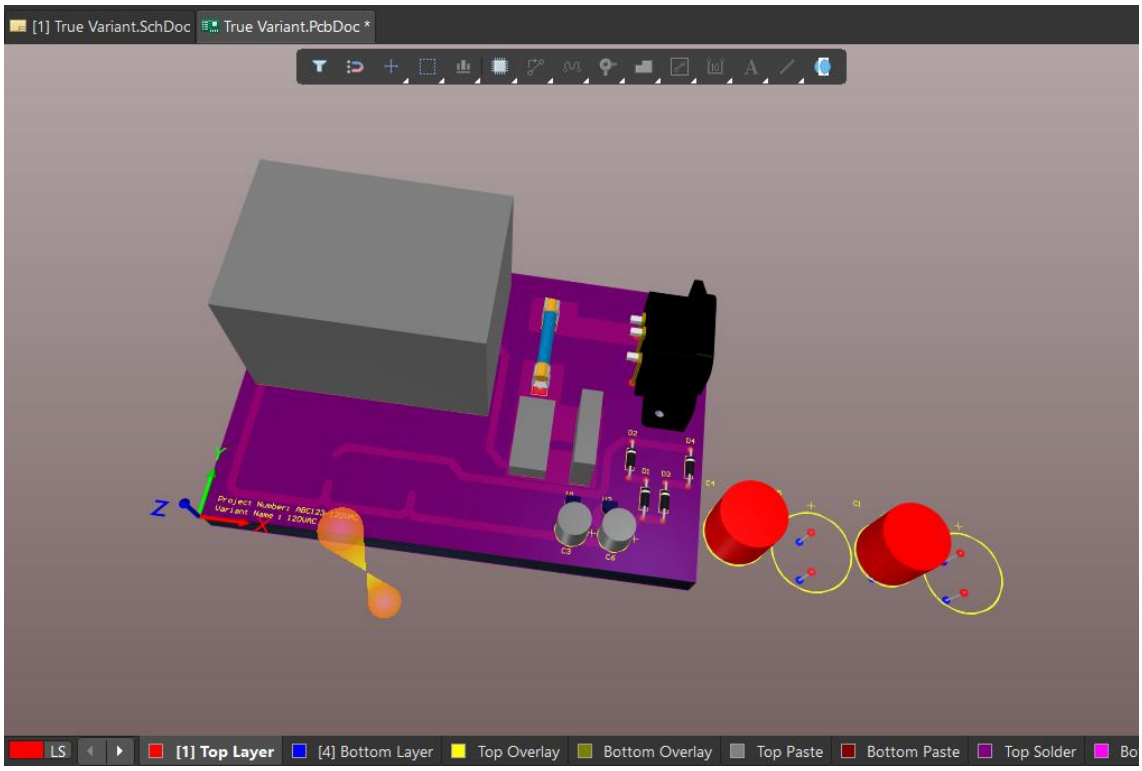


Figure 8. 120VAC Variant

5.3 Generating Variant Outputs

35. From the *Projects* panel, open the `True Variant.Outjob` file that is found in the *Settings* folder, as shown in Figure 9.

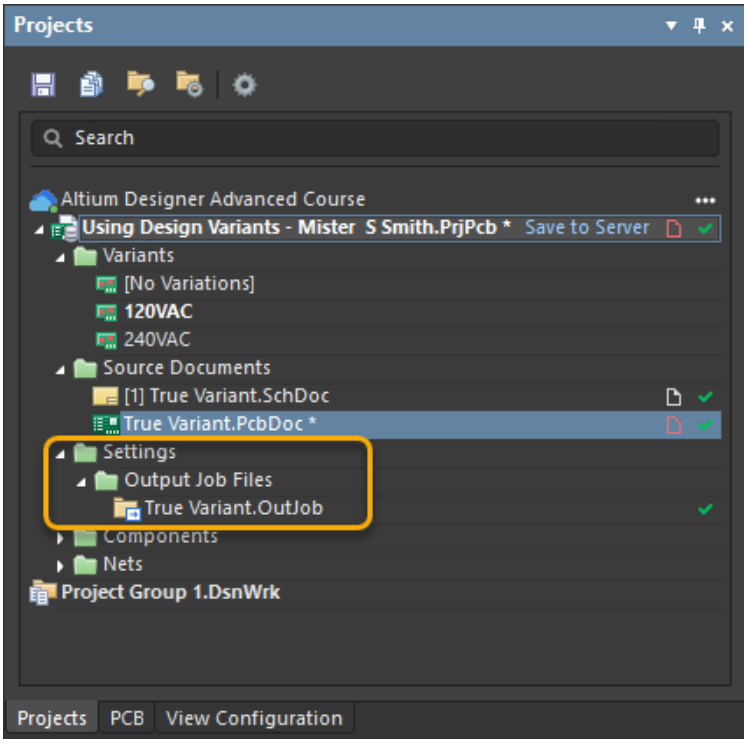


Figure 9. Output Job file from the Projects panel

36. At the top of the Output Job interface, in the Variant Choice section, select **Choose a single variant for the whole output job file**, as shown in Figure 10. For now, you will use the `120VAC` variant.

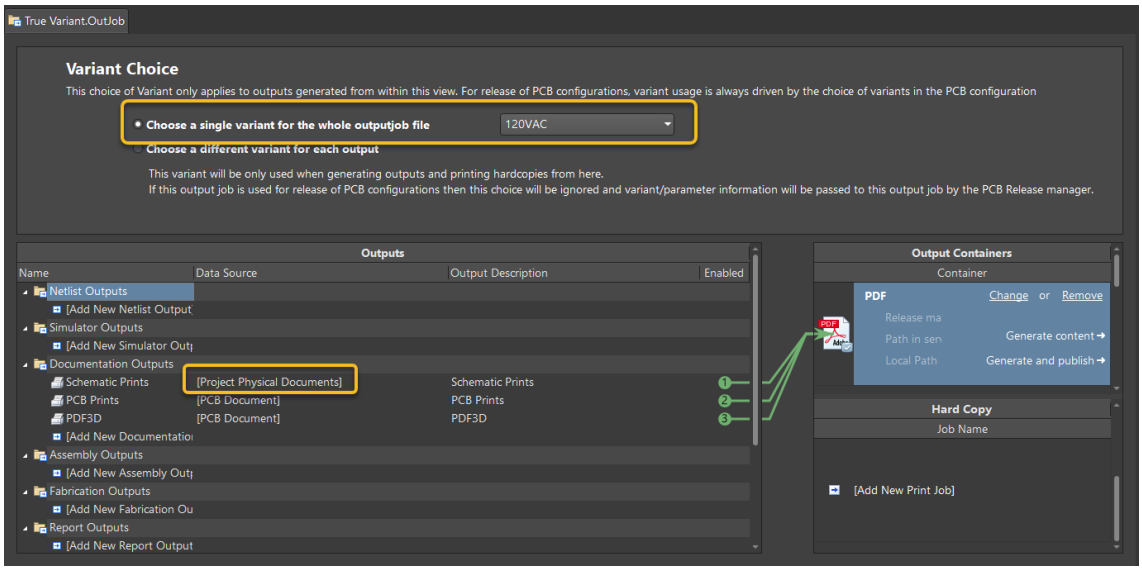


Figure 10. Generating PDF for Schematic and Assembly Drawing

37. In the Documentation Outputs section, ensure the Schematic Prints Data Source is set to **[Project Physical Documents]** for the variations to appear in the generated print.
38. The schematic and PCB prints are directed to a PDF output job container, shown by the green arrow. Next, you will generate the PDF for the selected variant.

39. Select the `PDF` container area and click the **Generate Content** button, as shown in Figure 11, to create a PDF of the Schematic, PCB Assembly Drawings, and 3D View.

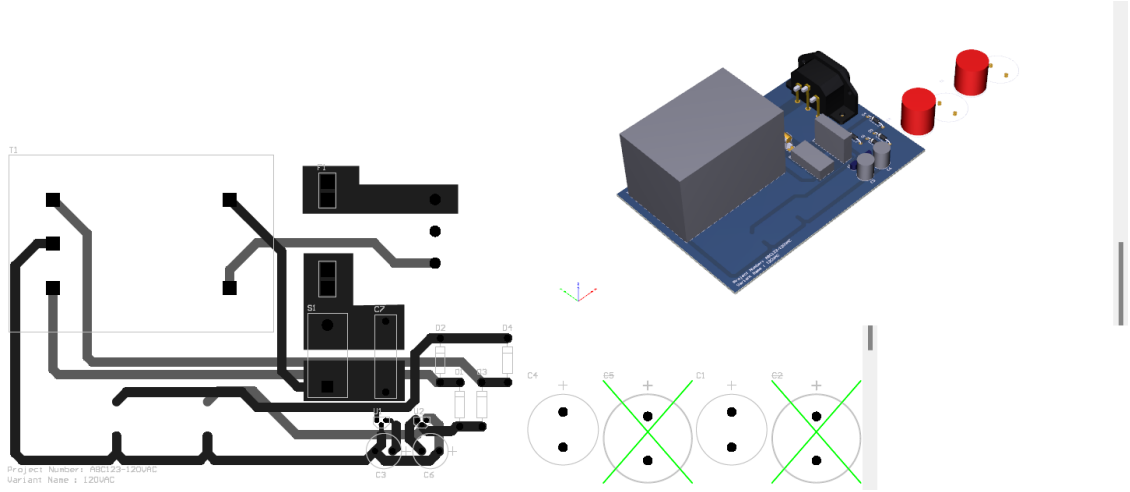


Figure 11. PCB documentation with Variant

40. Close the PDF after reviewing it.
41. Close the `True Variant.OutJob` document.
42. Close the PCB without saving the PCB document.

> Note: For PDFs, multiple jobs are automatically combined into a single document.

6 Alternative Parts

6.1 Creating Alternative Parts

43. Return to the `True Variant.SchDoc` schematic document.
44. From the **Project** menu, select **Variants...** to open the *Variant Management* interface.
45. In the *Variant Management* interface, change `C1`, `C4` and `C2`, `C5` back fitted.
46. Close the *Variant Management* interface with **OK**.
47. In the *Editor* tab of the schematic, delete components C2 and C5 from the red dashed area, Figure 12. You will focus on Alternate Components next and no longer need these.

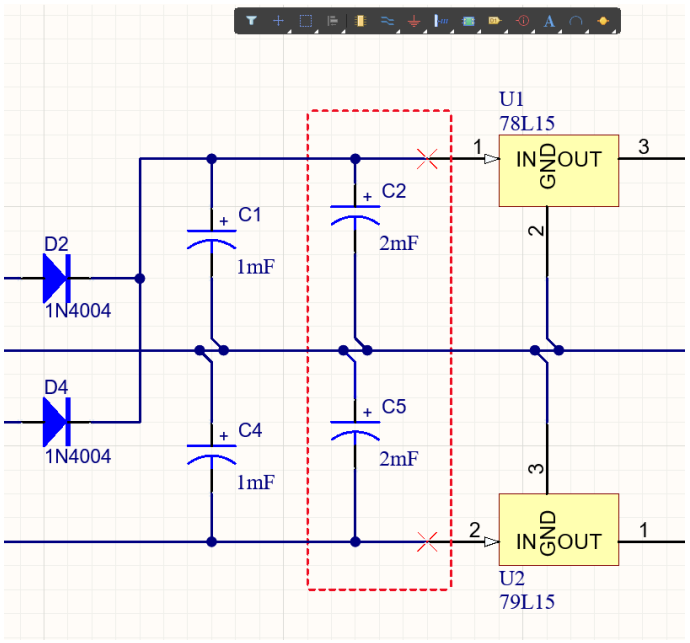


Figure 12. Delete components C2 and C5 on the schematic

48. From the **Project** menu, select **Variants...** to open the *Variant Management* interface.
49. In the `240VAC` column, select `C1` and select the ellipsis button ... to set it to **Alternate Part**.

50. Select **Replace Component...** to begin selecting the appropriate part as shown in Figure 13.

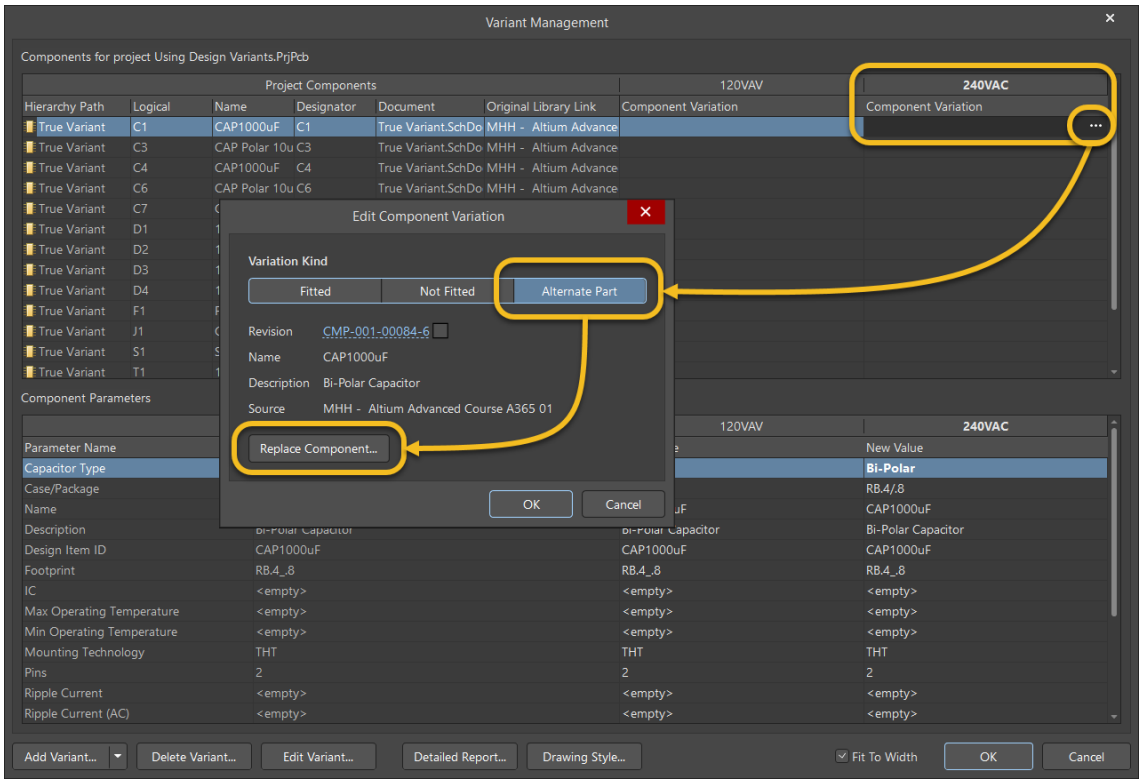


Figure 13. Selecting an Alternative Part

51. To view the available libraries and components, click the triple line icon to expand the panel, as shown in Figure 14.
52. Select the branch `Capacitors`. Feel free to use the search to filter.
53. Select `CAP2000uF`.

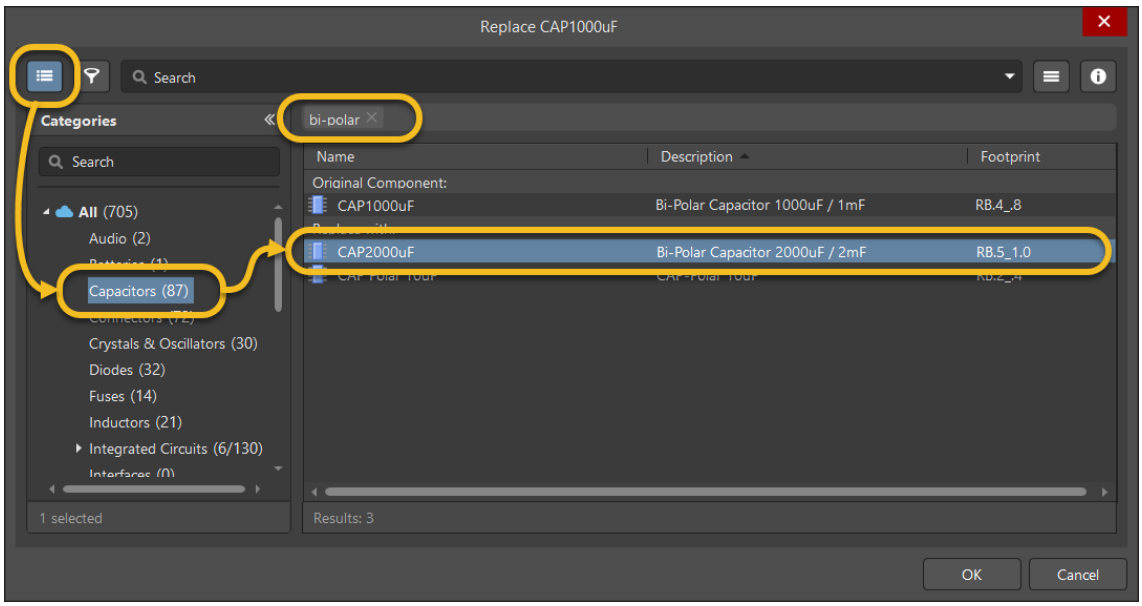


Figure 14. Alternate part selection

54. Select **OK** twice to return to the *Variant Management* dialog.
55. Repeat the steps above to assign `CAP2000uF` as the alternate part for `C4` for `240VAC`.

56. Make sure the 120VAC column has C1 and C4 assigned to 1000uF capacitors, and the 240VAC column has them assigned to 2000uF, as shown in Figure 15.

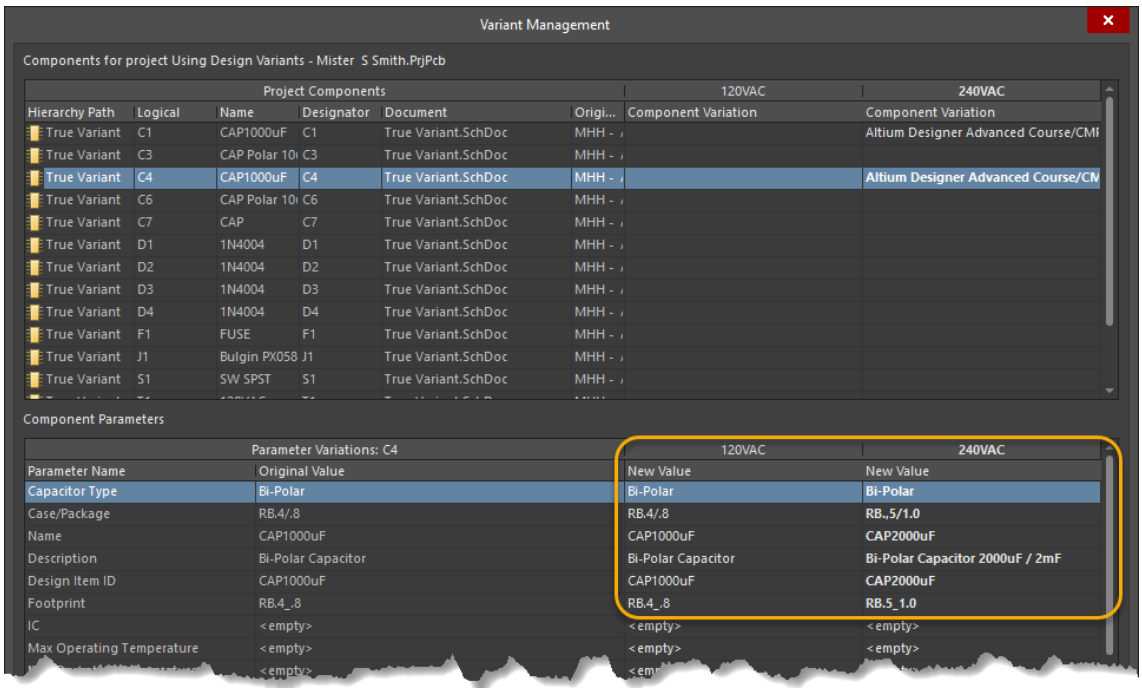


Figure 15. Alternate Part Assignment

57. Select **OK** to close the *Variant Management* dialog.

6.2 Viewing Alternative Parts

58. Validate the project by going to the Project menu and selecting **Validate PCB Project Using Design Variants.PrjPCB.**
59. Select the *Compiled tab named* `True Variant` located at the bottom left of your workspace.
60. 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. The difference is illustrated in Figure 16.

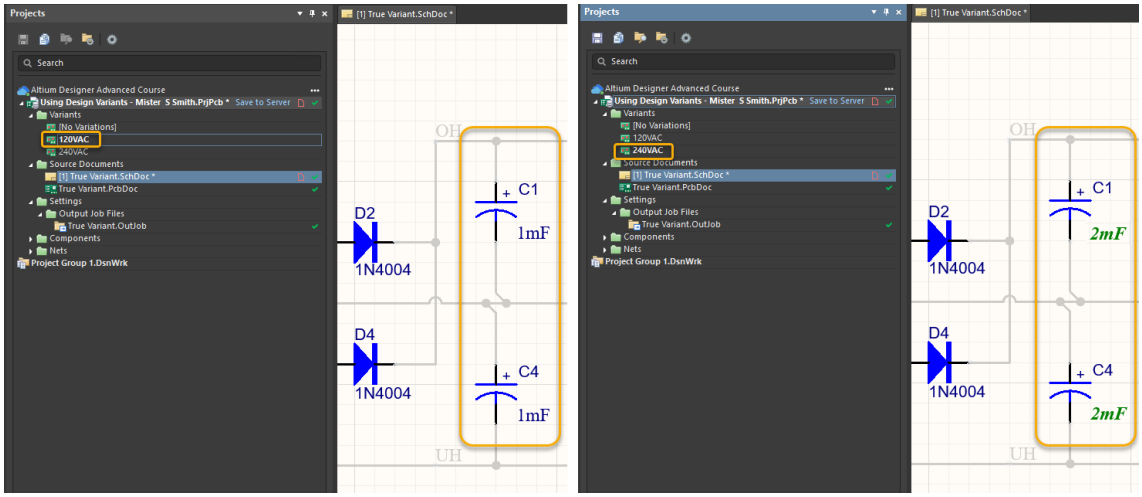


Figure 16. 120VAC and 240VAC Alternate Part Variant

7 Impact of Design Variants on PCB

61. Open the PCB file `True Variant.PcbDoc`.
62. From the **Design** menu, select **Import Changes from Using Design Variants.PRJPCB** to add the alternate components to the PCB. In the ECO dialog, you will see the original and alternate components added to the PCB, as shown in Figure 17. This might seem confusing initially, but it's the expected behavior.

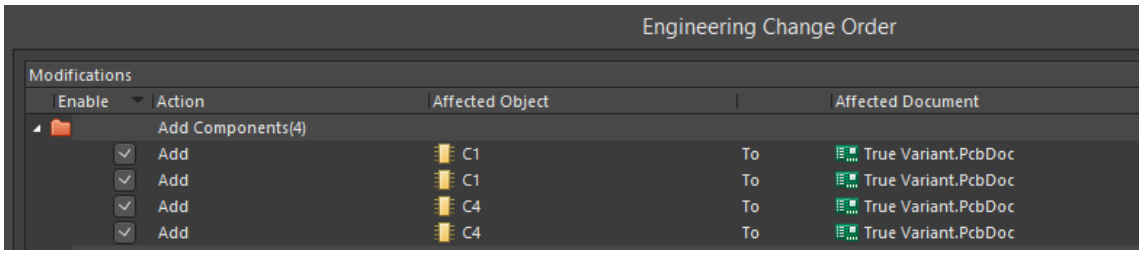


Figure 17. The base part and alternate parts are imported into the PCB

63. Select the **Execute Changes** button to import the changes to the PCB.
64. Select **Close** to exit the *Engineering Change Order* window.

Caution: You will have duplicate footprints for C1 and C4 in the PCB project. These components will be enabled or disabled based on the selected variant mode.

65. Go to the 3D View mode by pressing the **3** key.
    a) Switch between the `120VAC` and `240VAC` modes in the *Projects* panel.
    b) Confirm that `120VAC` and `240VAC` variant modes display the 3D body of its respective variant, as shown in Figure 18.
    c) Check the Strings with the Project Number and Variant Name at the lower left corner of the PCB.

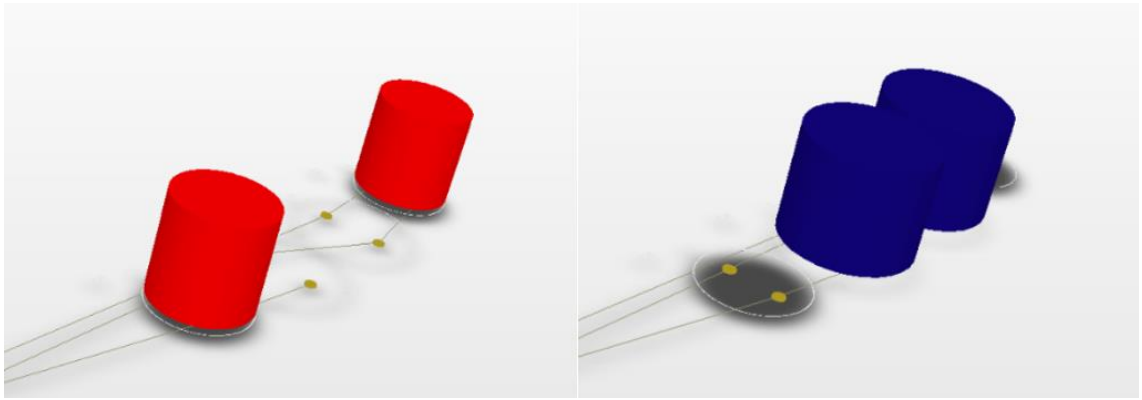


Figure 18. 3D View of Alternate Parts 120VAC and 240VAC

66. Switch back to 2D mode by hitting the **2** key.
67. Position the new components into the PCB, overlapping the duplicate footprints. If needed, activate the visibility of the Designators for the new capacitors.

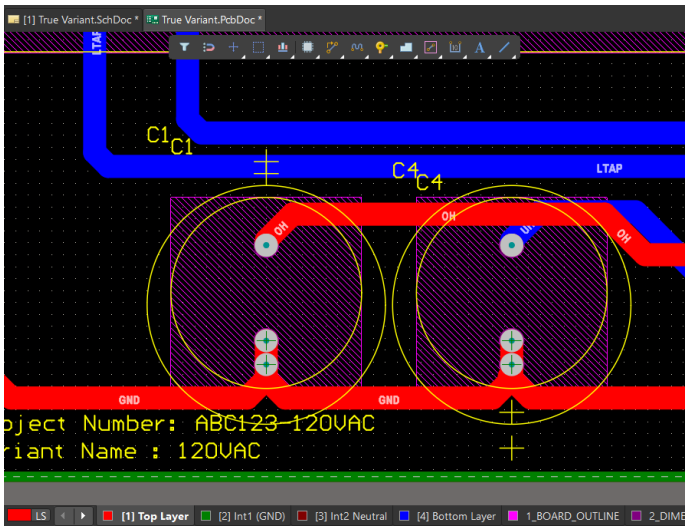


Figure 19. Overlapped Alternate Part Footprints in 2D View

68. Go back to the 3D View by pressing the **3** key.
69. Switch between the `120VAC` and `240VAC` modes in the *Projects* panel. Confirm that `120VAC` and `240VAC` variant modes display the 3D, as shown in Figure 20.

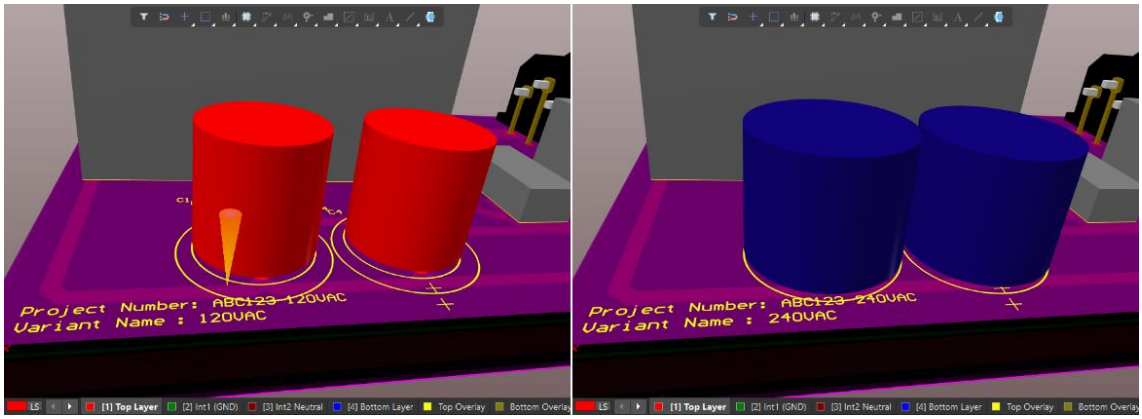
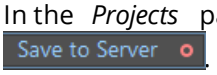


Figure 20. 3D View of the two Variants

70. Save all documents using **File » Save All**.
71. Save the modifications to the server:
    a) In the *Projects* panel, next to the Project name you find the command **Save to Server** 
    b) Select **Save to Server**.
    c) In the dialog *Save [Project Name]:*
        i) Add the comment `Using Design Variants - [Add Your Name] - Finished.`
        ii) Select **OK**.
72. When ready, close the project and any open documents, **Window » Close All**.

Congratulations on completing the Module!

Using Design Variants

from

Altium Designer Advanced Training with Altium 365

Thank you for choosing **Altium Designer**