

Power Tips: Simplify Creation of Multiphase and Multimodular Board Designs with Altium



Sarmad Abedin

As the demand for automotive and industrial power increases, so does the need for multiphase and modular-based designs. Designing a multiphase or multimodule board can be tedious and take a long time. In this Altium tips blog, I will show you how to easily copy and paste an existing schematic and layout to create a multiphase or multimodule designs.

Step No. 1: Build a simple buck-converter schematic in Altium (Figure 1).

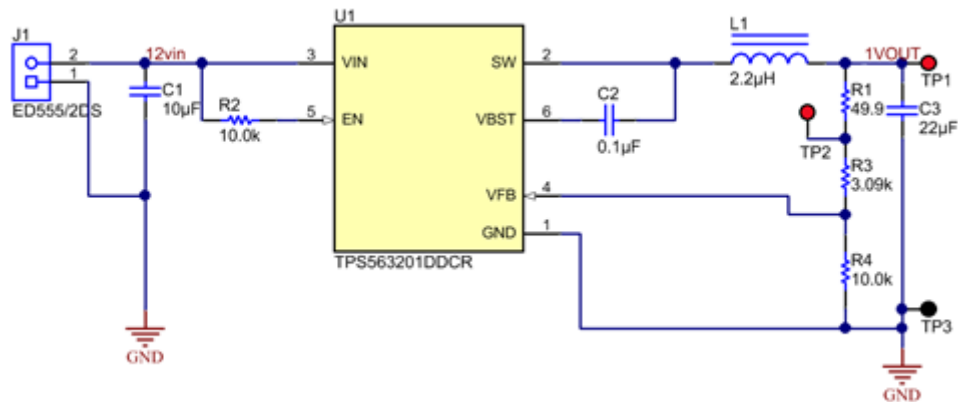


Figure 1. Altium Schematic

Step No. 2: Update the printed circuit board (PCB) and layout the components. Pour the polygon pours as normal and connect all of the nets (Figure 2).

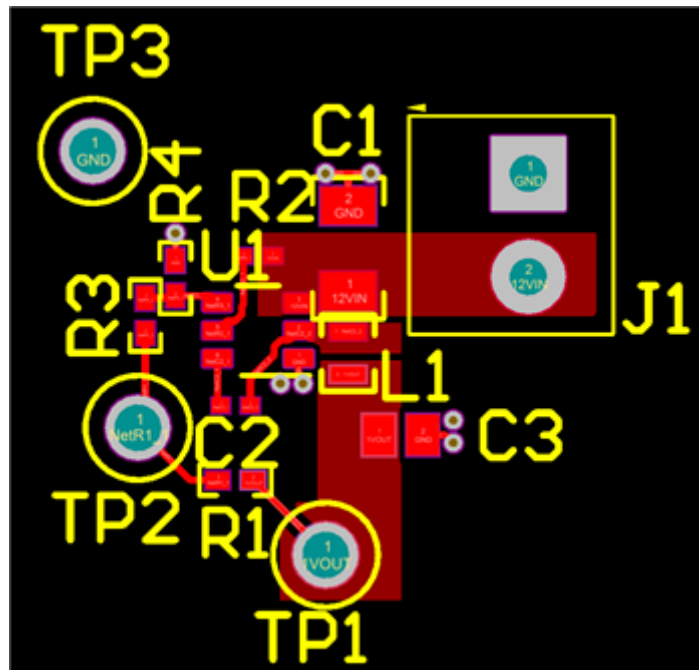


Figure 2. Layout in Altium

Step No. 3: Select the components you want to copy by clicking and dragging; then right-click and select Copy (Figure 3).

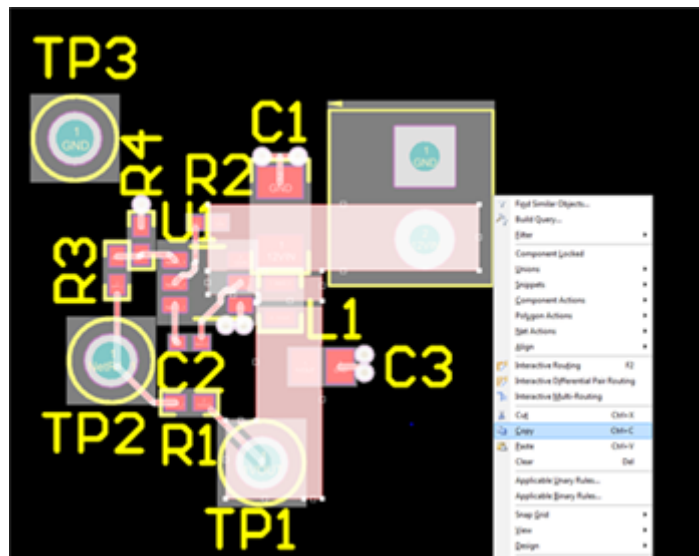


Figure 3. Screenshot of Step No. 3

After copying, Altium will ask for a reference point; click a specific component.

Step No. 4: Paste the new components next to the existing components, as shown in Figure 4. Altium will add “_1” next to each reference designator to signify that it’s a copied component.

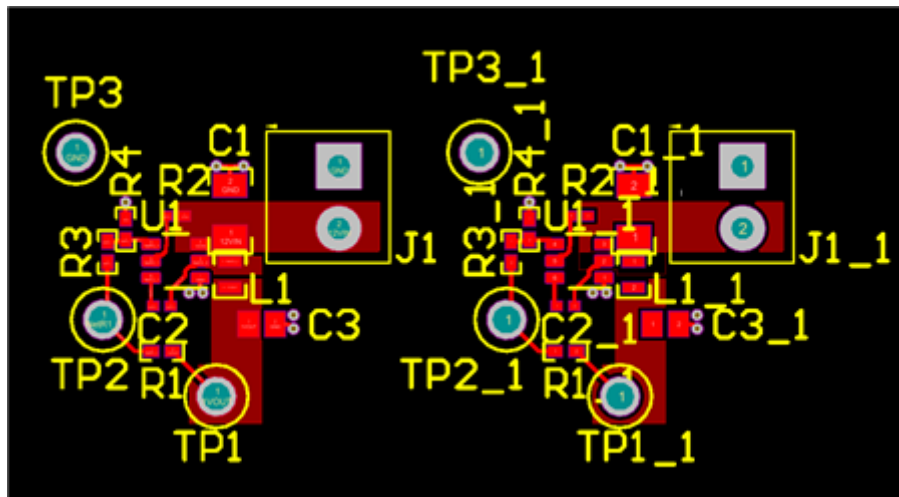


Figure 4. Layout in Altium Showing Copied Components

Now that you've successfully copied the components into the layout, you'll need to do the same in the schematic window and then link the components.

Step No. 5: In the Projects window, right-click the project name and click Add New to Project; then click Schematic (Figure 5). This will give you a new schematic sheet to copy components into.

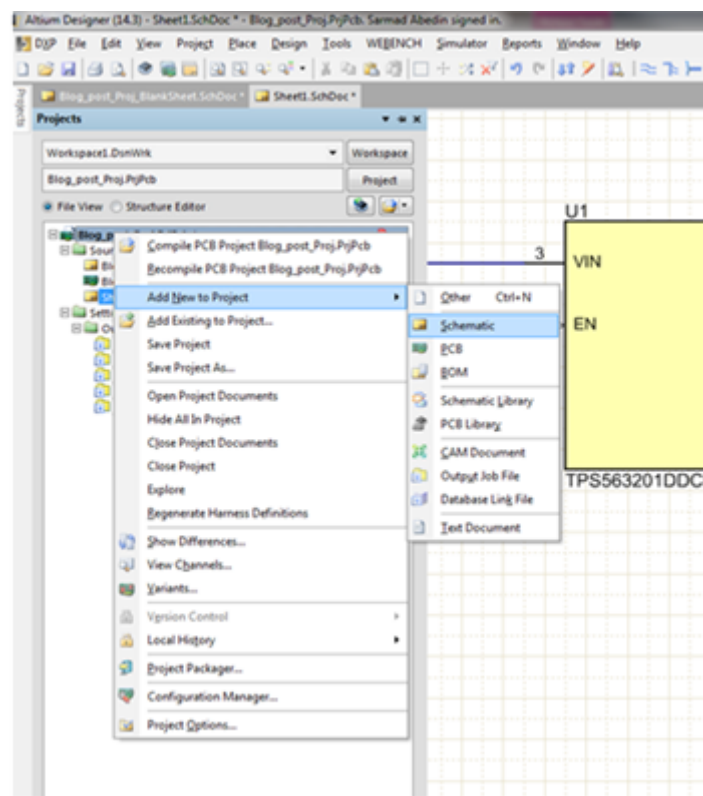


Figure 5. Screenshot of Step No. 5

Step No. 6: Copy and paste the schematic in Figure 1 to the new sheet. You will have to add “_1” to all of your new components before you can link the schematic with the current layout. To do that, you can use the Smart Edit option in Altium.

Step No. 7: Right-click U1 (new sheet) and select Find Similar Objects (Figure 6). Click OK in the pop-up menu.

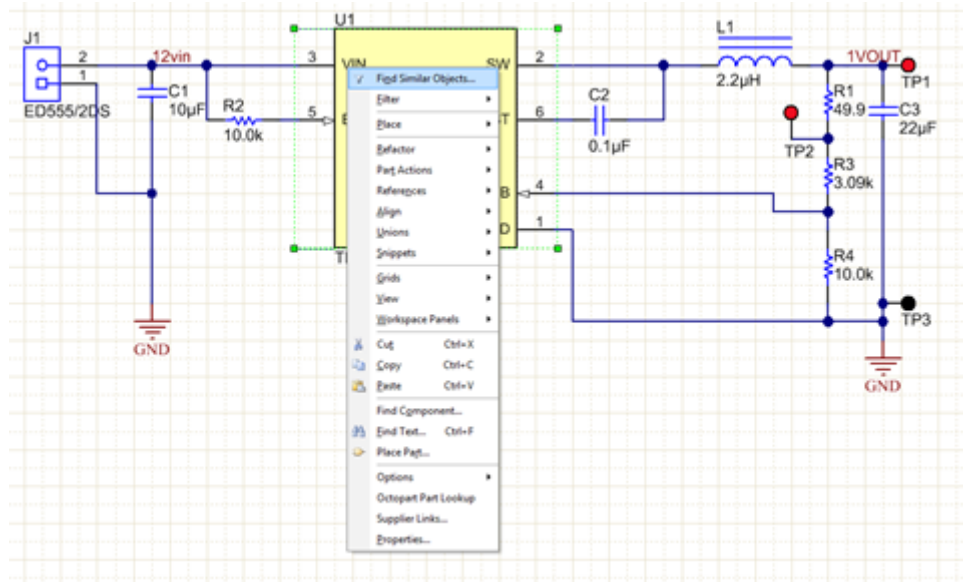


Figure 6. Screenshot of Step No. 7

Altium will highlight all of the components on the schematic and open the SCH Inspector window.
Step No. 8: Click the “...” button to the right of Component Designator (Figure 7).

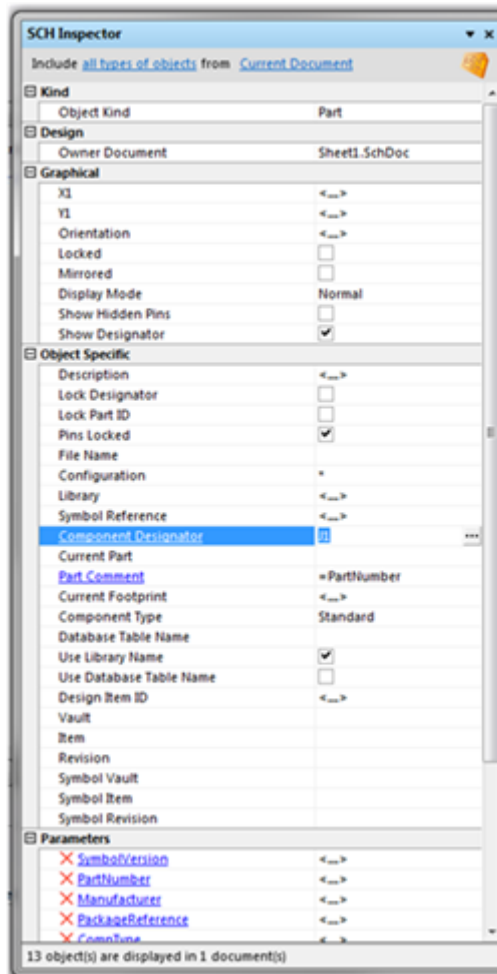


Figure 7. Screenshot of Step No. 8

Step No. 9: In the Smart Edit window, click the Formula tab (Figure 8).

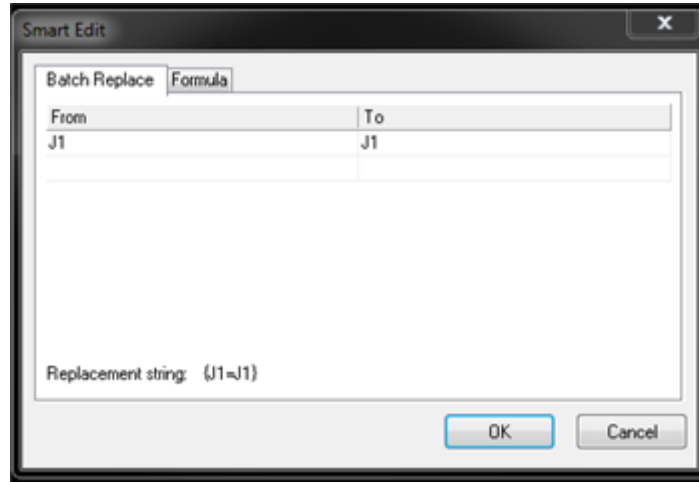


Figure 8. Screenshot of Step No. 9

Step No. 10: In the Apply formula field, type "! + '_1'" (no quotation marks) and click OK (Figure 9).

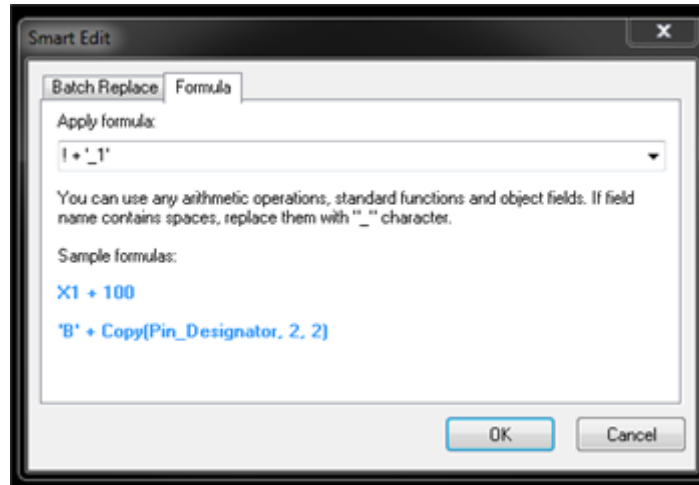


Figure 9. Screenshot of Step No. 10

Altium will add "_1" to the existing reference designators. This will now match your layout (Figure 10).

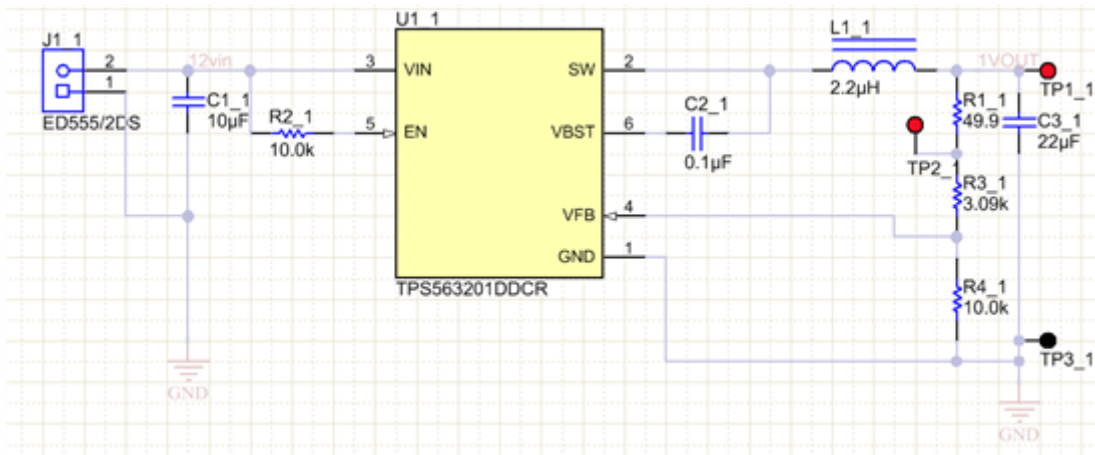


Figure 10. Revised Schematic

Step No. 11: While in the schematic window, update your PCB (Design > Update) to see the net(s) appear, as in Figure 11.

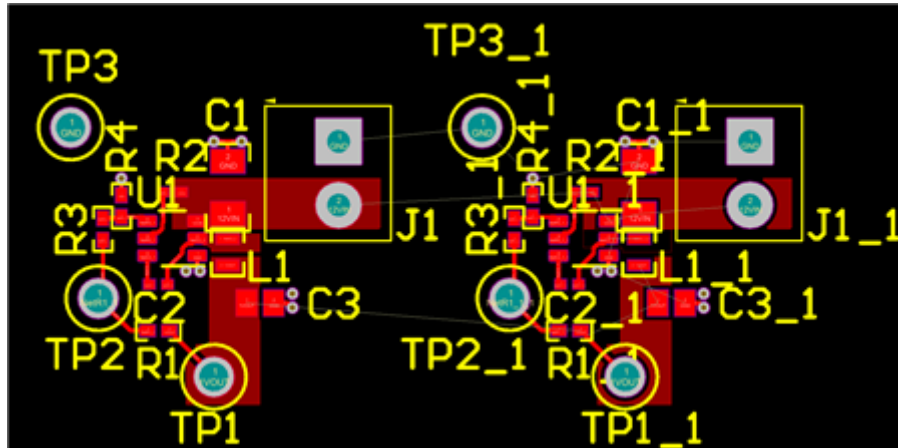


Figure 11. Revised Layout in Altium

You must also add “_1” to any nets where you don’t want the modules to share. Follow the same Smart Edit procedure.

Step No. 12: Right-click any net labels. Select Find Similar Objects and click OK (Figure 12). After Altium highlights all of the nets, click off of any nets you want the modules to share, like V_{IN} , etc.

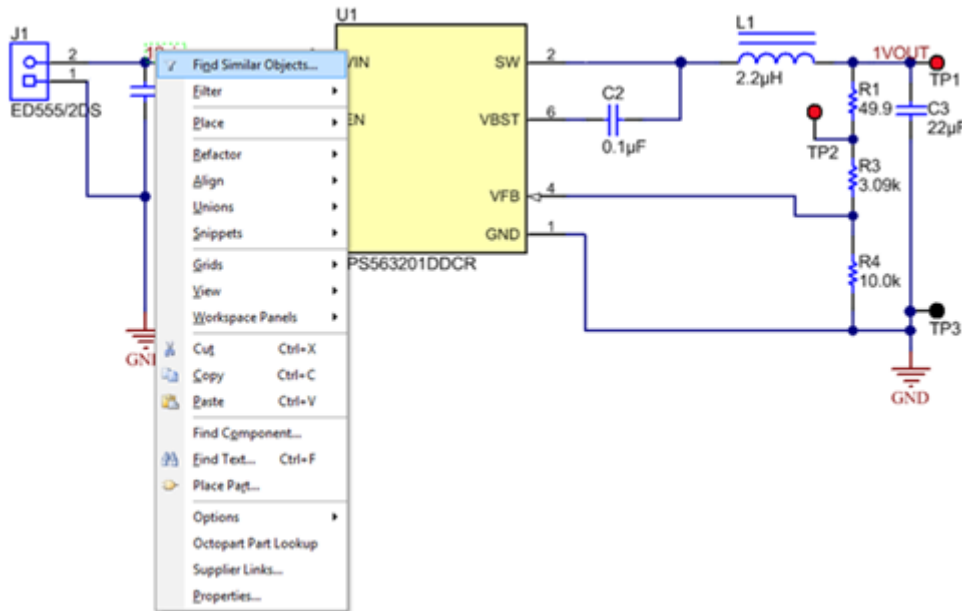


Figure 12. Screenshot of Step No. 12

Step No. 13: In the SCH Inspector window, click “...” to the right of Text (Figure 13).

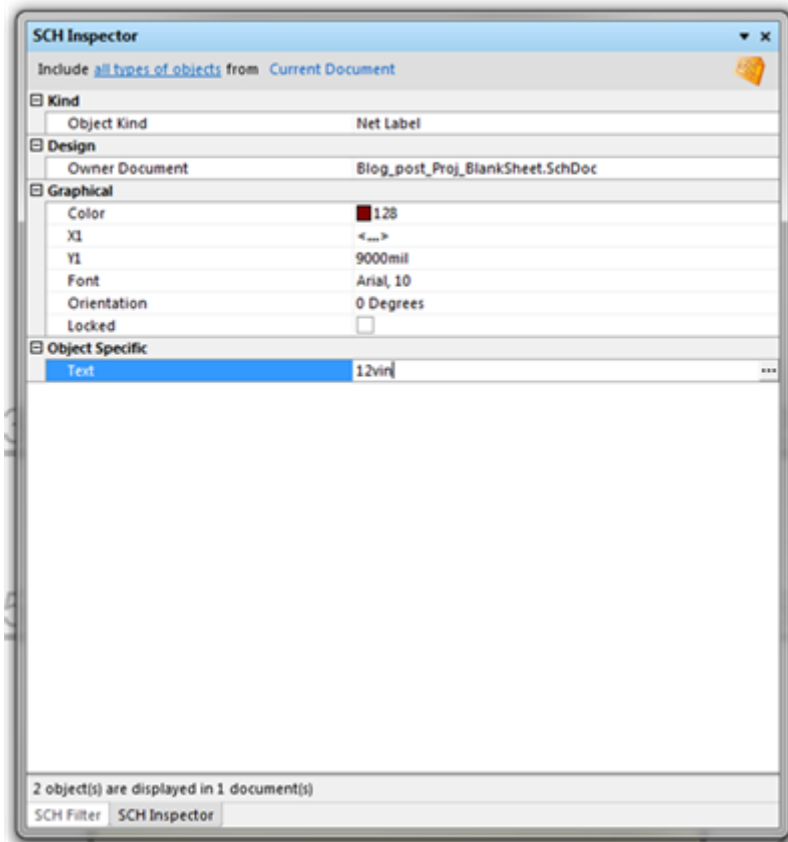


Figure 13. Screenshot of Step No. 13

Step No. 14: In the Smart Edit window, click in the Apply formula field and type ! + '_1'. Click OK (Figure 14).

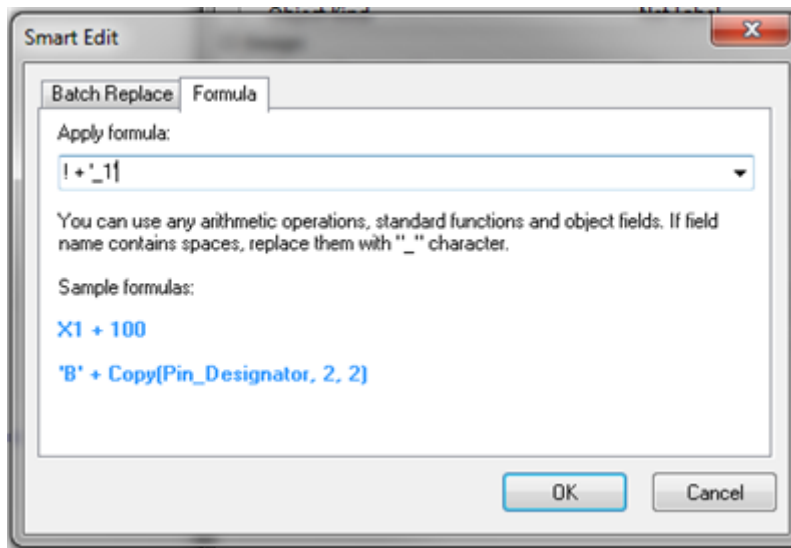


Figure 14. Screenshot of Step No. 14

All of the nets will now have “_1” next to them, signifying that they are from your original module. Your new schematic sheet will look like Figure 15.

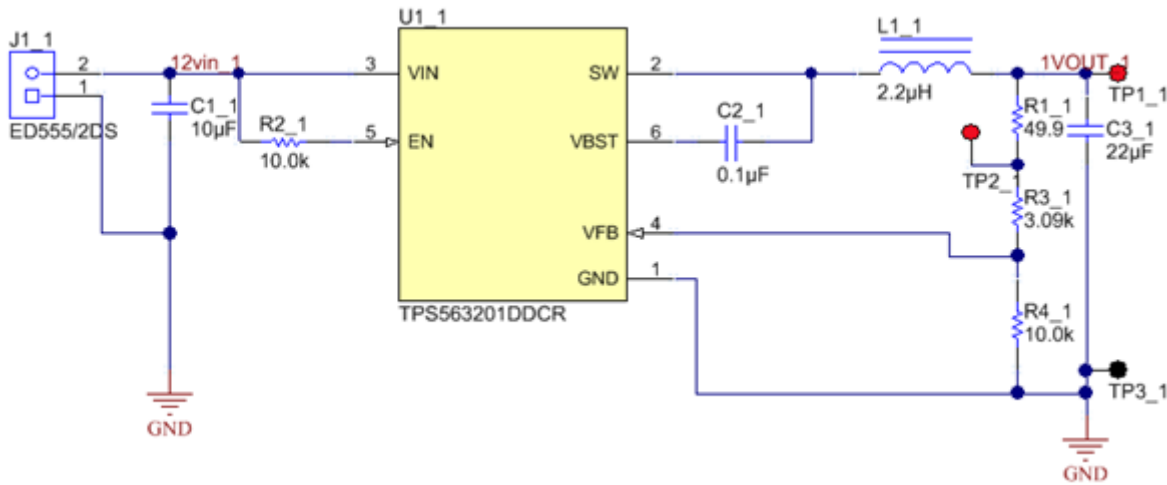


Figure 15. Revised Altium Schematic

Step No. 15: Update the PCB; you'll see new net names (Figure 16). The only net shared between the two modules/phases is now the GND net.

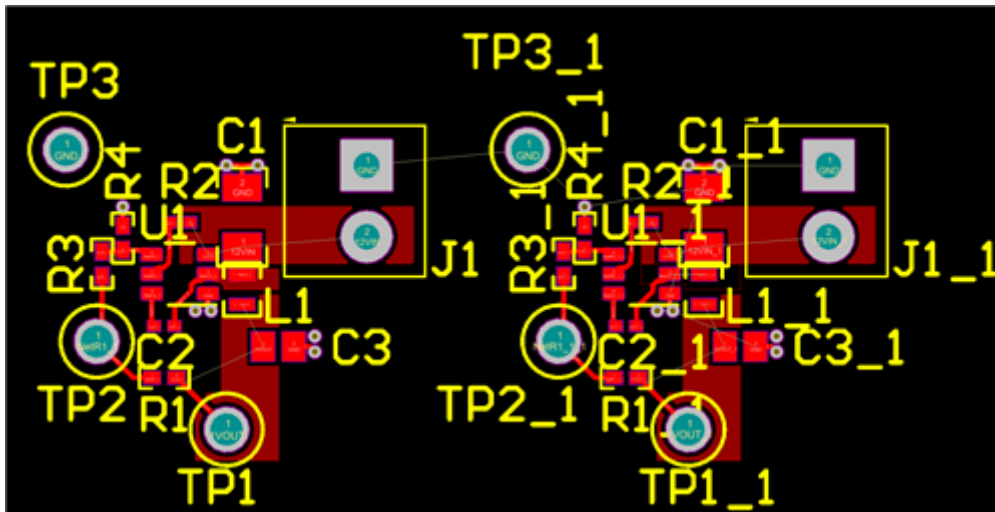


Figure 16. Revised Layout in Altium

Polygon pours will copy over, but you will need to rename them to the new nets (Figure 17).

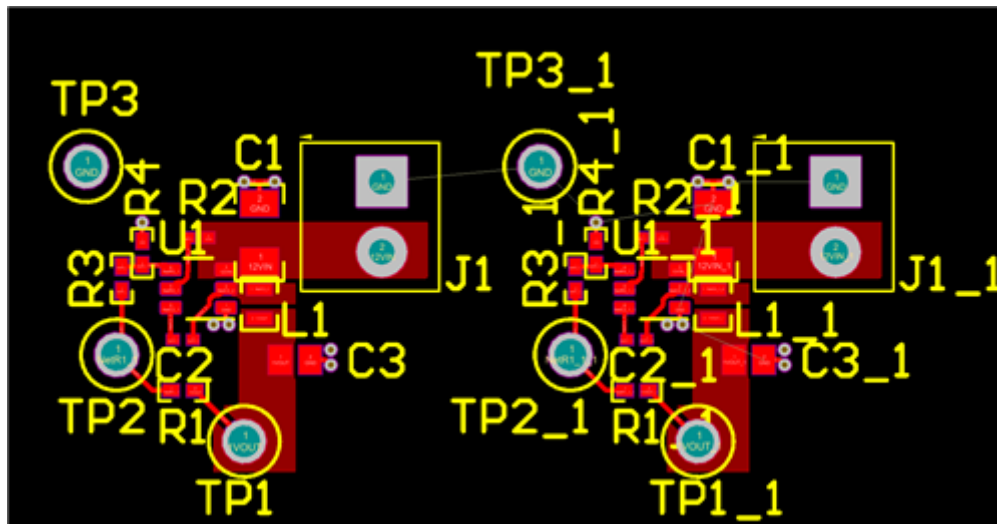


Figure 17. Revised Layout in Altium

These steps should save you substantial time when making multiphase or multimodular PCBs in Altium and help designers get their products to market faster.

With current-sharing designs in particular, the smallest of differences in layout can lead to nonequal sharing. So another great reason to use this copy-and-paste trick is that it helps reduce the small differences that can come from individual component placement.

Additional Resources

- Get more information on [Altium Designer](#).
- Explore more [power-supply topics](#).
- Watch Power Tips [videos](#) to help with your design challenges

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATA SHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES "AS IS" AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, regulatory or other requirements.

These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you will fully indemnify TI and its representatives against, any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI's products are provided subject to [TI's Terms of Sale](#) or other applicable terms available either on [ti.com](https://www.ti.com) or provided in conjunction with such TI products. TI's provision of these resources does not expand or otherwise alter TI's applicable warranties or warranty disclaimers for TI products.

TI objects to and rejects any additional or different terms you may have proposed.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265
Copyright © 2023, Texas Instruments Incorporated