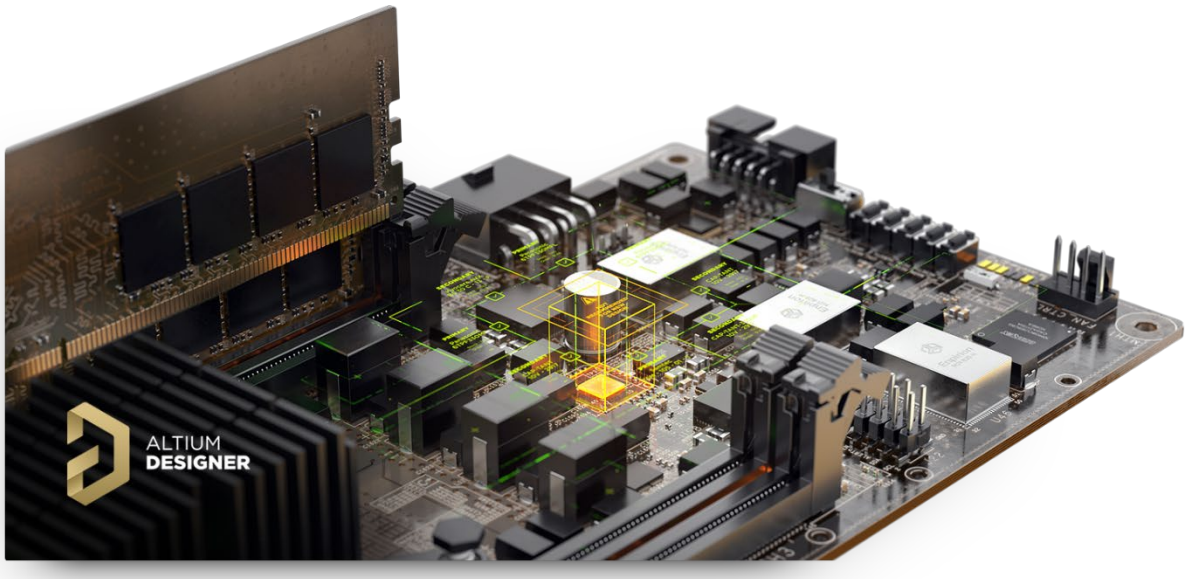

Altium Designer

Advanced Course

Module: Back Drills

Table of Contents

Back Drills

1.1 Purpose

Back Drilling (or Controlled Depth Drilling CDD) removes unwanted copper stubs from Vias, and if needed, Pads to improve the signal integrity through the copper traces and Vias. For this exercise, we will be using a pre-routed multi-layer PCB to illustrate how to set up for Back Drilling. Once the setup is completed the PCB will have its Vias modified automatically based on the Rules and Drill Pairs that were set up.

1.2 Shortcuts

Shortcuts when working with Back Drills

F1:	**Help**
D-K:	Open Layer Stack Manager
2:	2D Mode
3:	3D Mode
CTRL+S:	Save Document

1.3 Preparation

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

1.4 Back Drill - Layer Stack Manager

The basic steps for configuring the PCB for Back Drilling are as follows:

- Creating the needed/desired Drill Pairs that drive the Back Drilling of Vias/Pads
- Setting up the design rule for Back Drilling – Maximum Via Stub Length Rule

Once these two steps have been made, existing and added Vias/Pads will be converted into the proper types for Back Drilling.

3. In reviewing the schematic (open `Back Drill.schdoc`), note the use of Blanket directives with the added Parameters to create a Net Class of all the nets in the design. This will be used in subsequent steps.
4. Switching to the `BackDrill.PcbDoc` document, we see all the routing and the vias used. Open up the *Layer Stack Manager (LSM)* from the **Design** dropdown menu to see the PCB stack-up. This is a 10-layer board with six signal layers and 4 power plane layers.
5. From the *Layer Stack Manager* window click on the **Features Button,** then the **Back Drills** button, Figure 1.

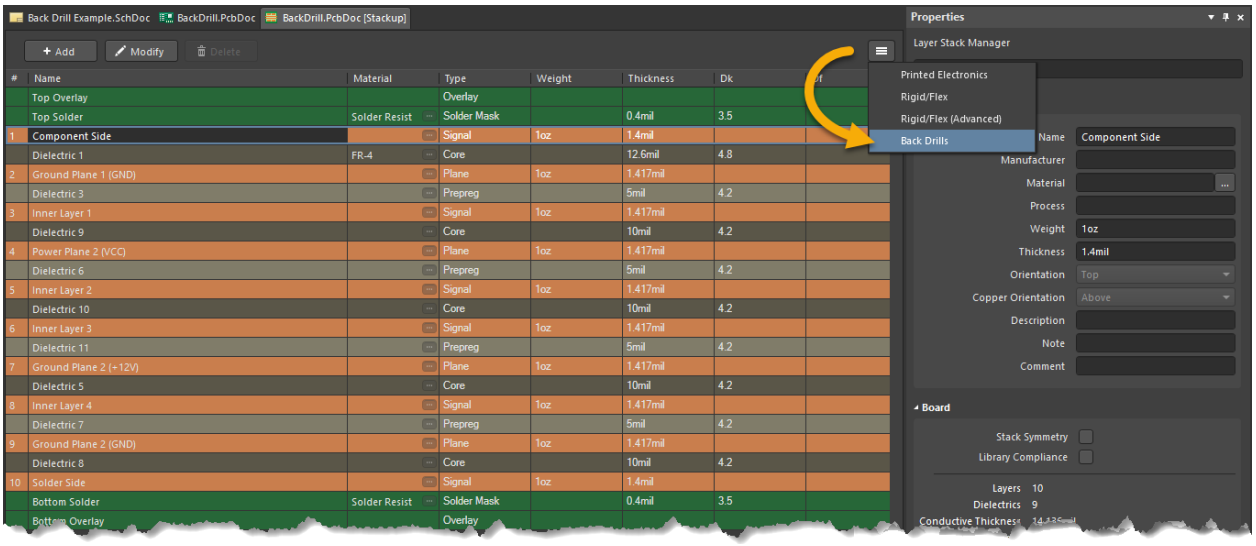

Figure 1. LSM showing 10 Layers and the Back Drills button

6. Once selected, you'll notice the Features become available as tabs along the bottom. Select *Back Drills*.

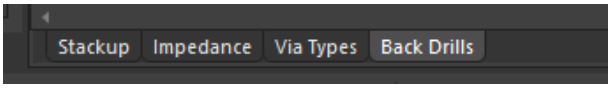

Figure 2. Layer Stack Manager and the Features

7. The LSM is now split with the layers on the left and Back Drills on the right. The back drill pane starts off empty. First, we will click the **+ Add** button to create the first Back Drilled via. You'll note the default via is a Component Side back drill, meaning the Top Layer (Component Side) will be back drilled, Figure 3.

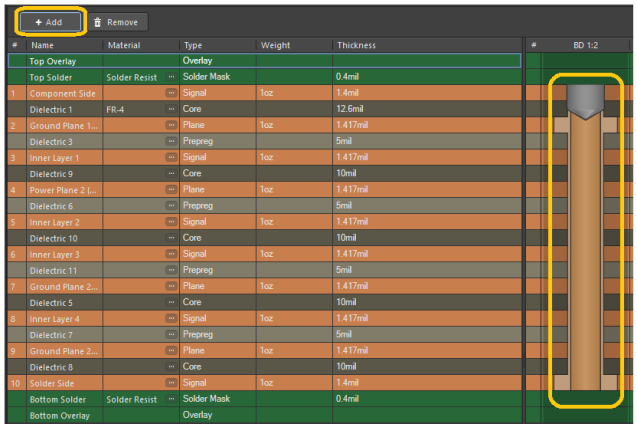

Figure 3. Default Back Drill added to the Layer Stack Manager

8. Once the Back Drill has been added, select it and open the *Properties* panel. We will leave the *First Layer* as **1 - Component Side**. Let's set the *Last Layer* (or Stop Layer) as **3 – Inner Layer 1**. Note the back drill display updates to reflect the new Back Drill setting, Figure 4.

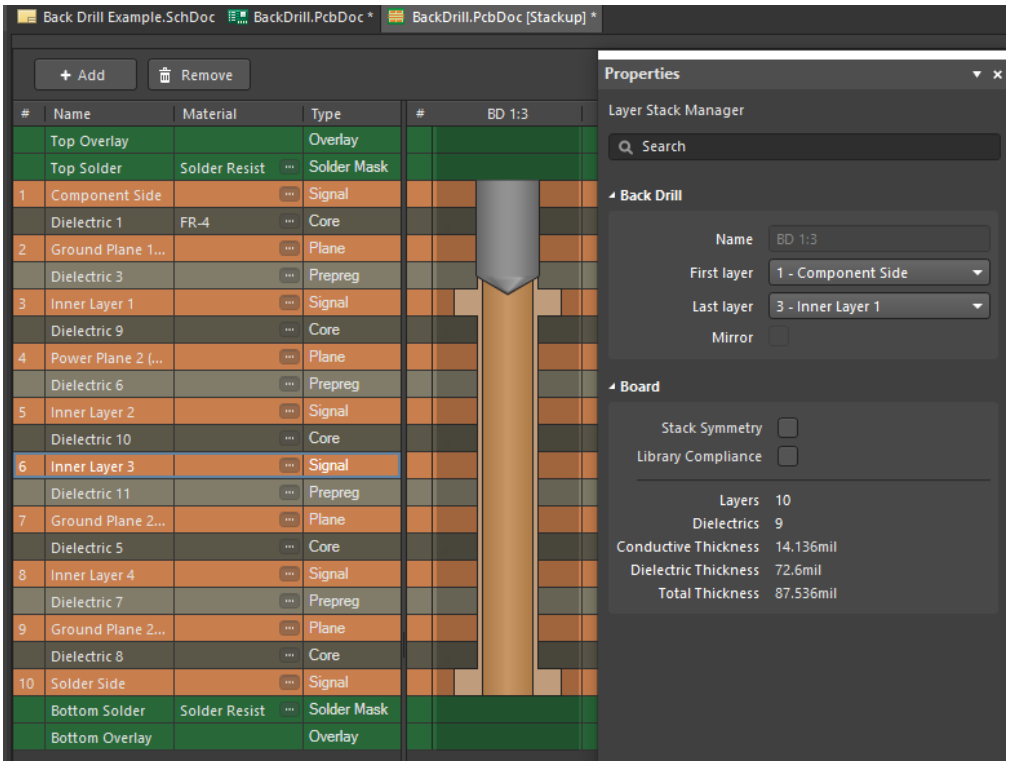

Figure 4. Manually created Back Drill enabled Drill Pair

9. Now we need to add the remaining Back Drills. Using Figure 5 as your guide, add the remaining Back Drills.

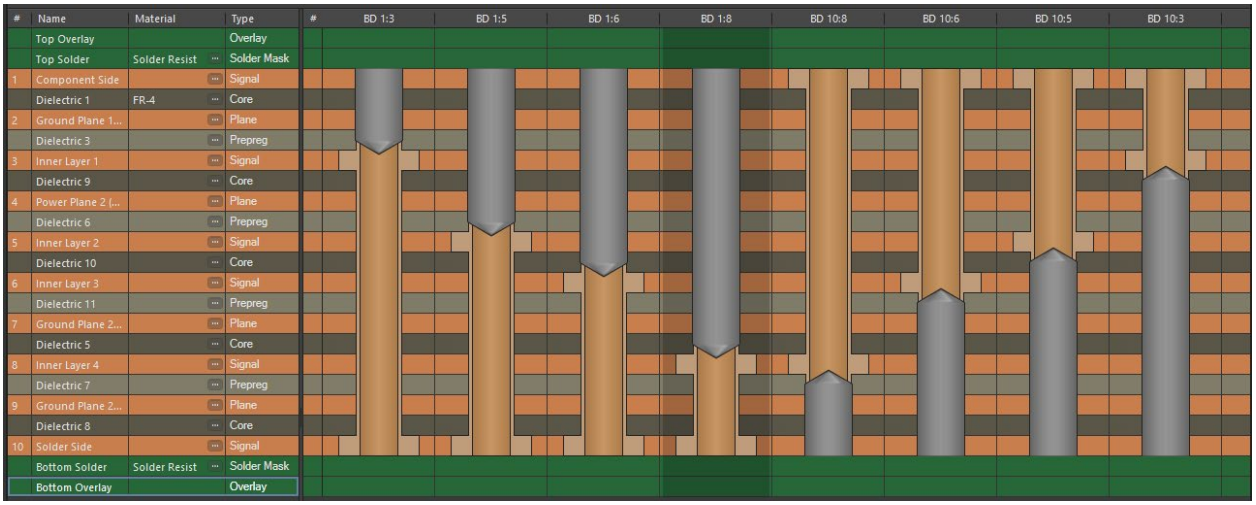

Figure 5. Finished set of Back Drills

10. Right-click on tab `BackDrill.PcbDoc [Stackup]` and select **Save BackDrill.PcbDoc [Stackup]**. Right-click on the tab again and select **Close BackDrill.PcbDoc [Stackup]**.

1.5 Back Drill – Design Rules

11. Open up the PCB Rules (**Design » Rules**) and expand the *High-Speed* category.
12. Under the *High-Speed* category click on the *Max Via Stub Length (Back Drilling)* Rule, right click on it and select **New Rule,** Figure 6.

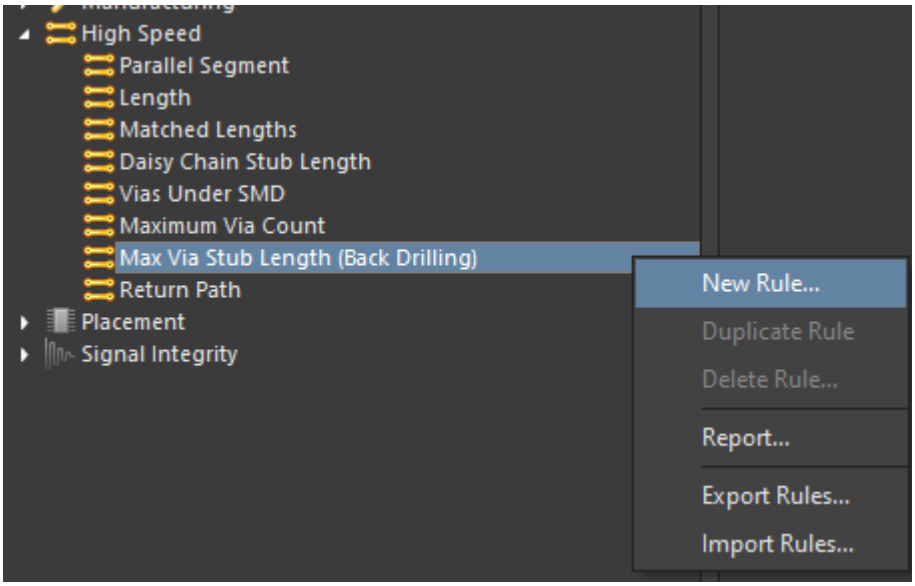

Figure 6. Adding New Max Via Stub Length rule

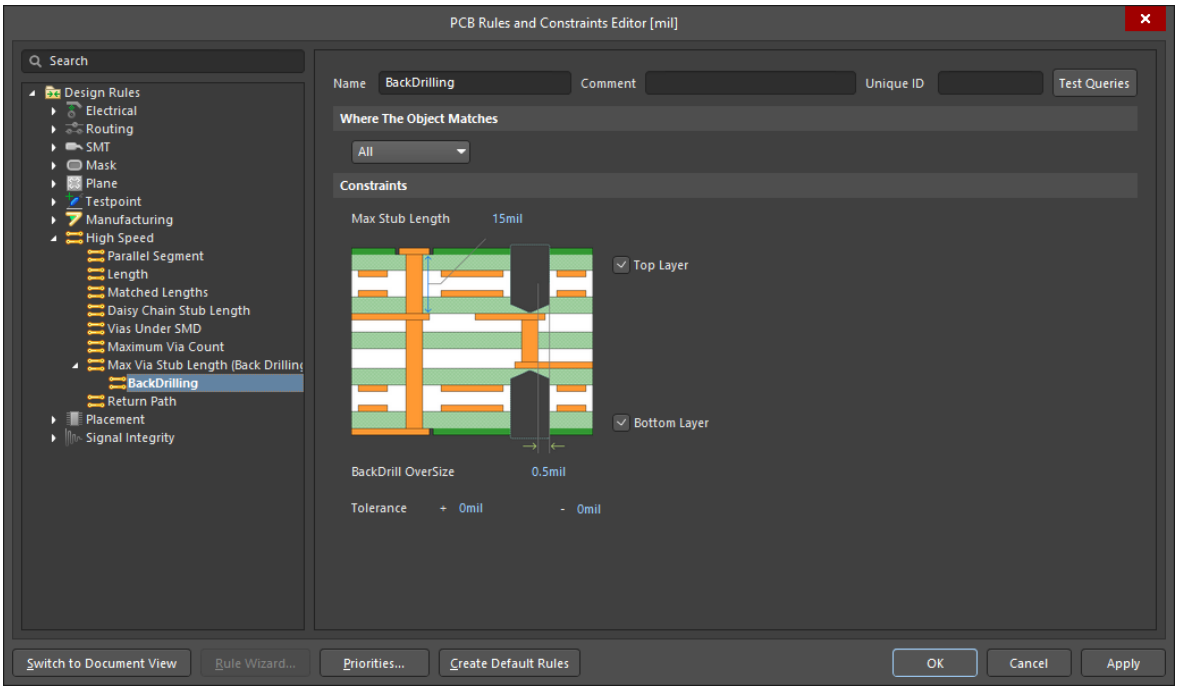

Figure 7. Newly added default Max Via Stub Length Rule

13. Now we will edit the new default rule to reduce the Max Stub Length to 9mil to ensure that the back drilling will remove unneeded vias longer than 9mil.

14. Now we would normally want to limit the Back Drilling to only the high-speed nets requiring it. In order to scope this new rule to our targeted nets we could use the Object Matching drop down menu and pick **Net Class** – then select the **HSdata Net Class** from the 2nd pulldown menu. This would limit Drill Pairs to only pads and vias associated with the HSdata nets, Figure 8.

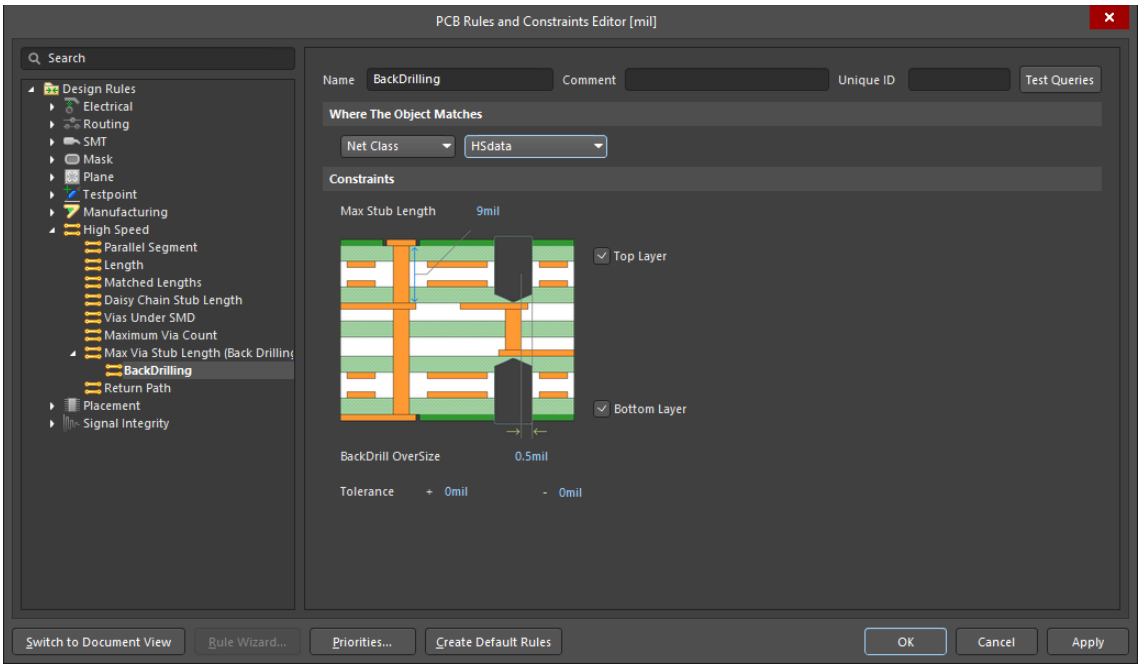

Figure 8. HSdata Net Class Scope for rule

15. This would work well enough on the current design, but to illustrate the capability of the Rules we can limit the rule to only Vias associated with the HSdata Net Class using the Custom Query option: `InNetClass('HSdata') and IsVia`, Figure 9.

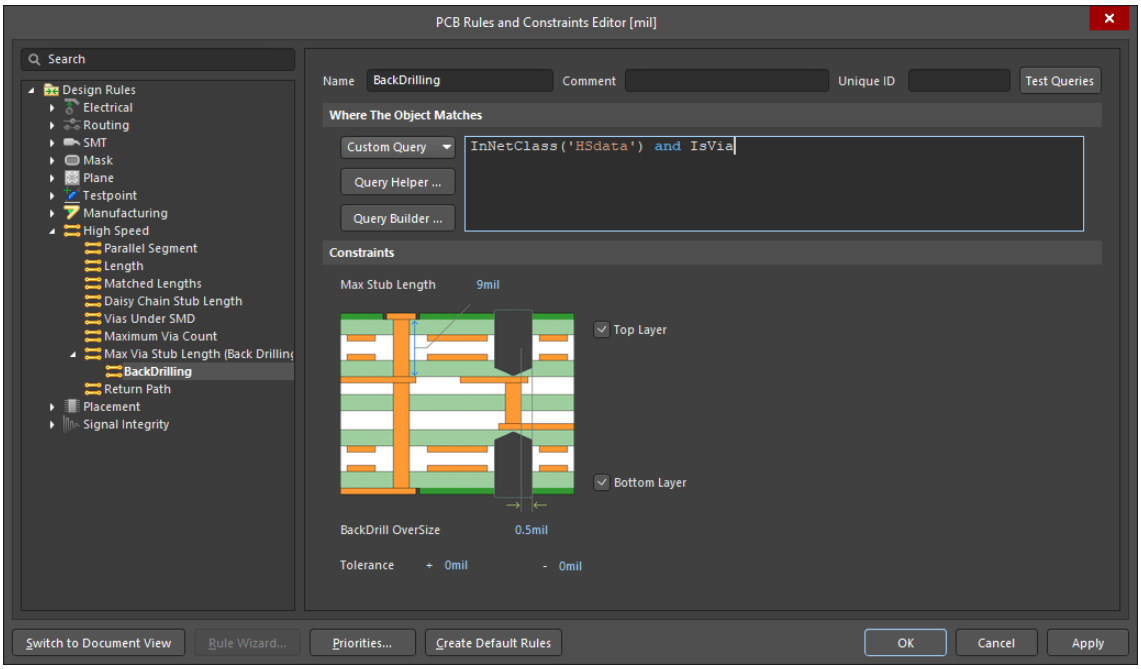

Figure 9. Using Custom Query to limit Rule to Vias with the nets of the Net Class HSdata

1.6 Back Drill – PCB View

16. Hitting **OK** and closing the PCB Rules we now look at the effect on the PCB's vias. In 3D we see holes for the internal vias but no metal showing on the top as this would be removed by the Back Drilling Operation, Figure 10.

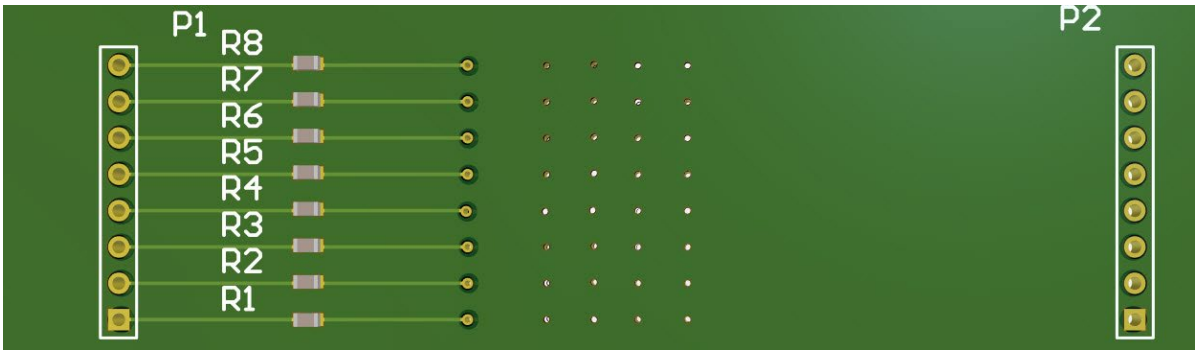

Figure 10. Top of PCB in 3D view showing holes without metal for the Back Drilled vias

17. Switching to the 2D viewing mode, zoom into those back drilled vias. Here we see they have a bicolored inner circle indicating that they are back drilled, the colors indicate which layers are involved, Figure 11.

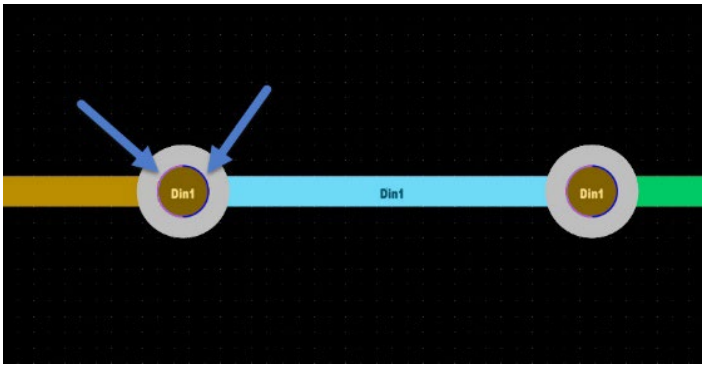

Figure 11. Via color coding for Back Drill

18. Switching back to 3D mode and using the **SHIFT+S** single layer viewing option we can cycle through the various layers of the PCB using the **+** button as you can see in the series of figures below, Figure 12:

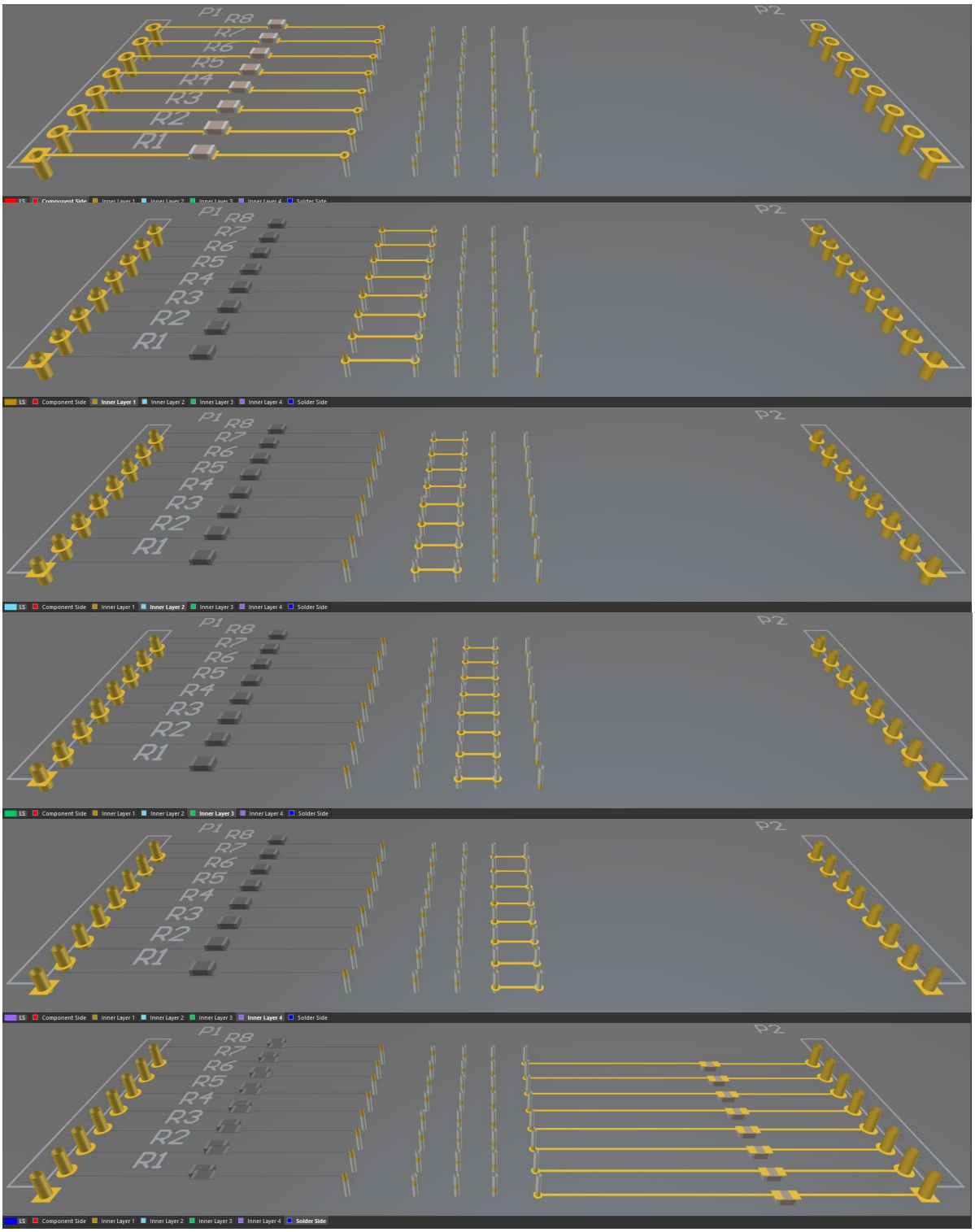

Figure 12. 3D Single Layer Viewing stepping from Top to Bottom layers

1.7 Drill Documentation

19. When the NC Drill file is generated there will be multiple files for the Back Drilling, ensure that all are sent to the PCB fabricator.
20. To see all back drills from the design, open the *PCB* panel and select the *Hole Size Editor***.** As for the other views from the *PCB* panel you can jump / mask individual objects by selecting/ deselecting the different objects, Figure 13.

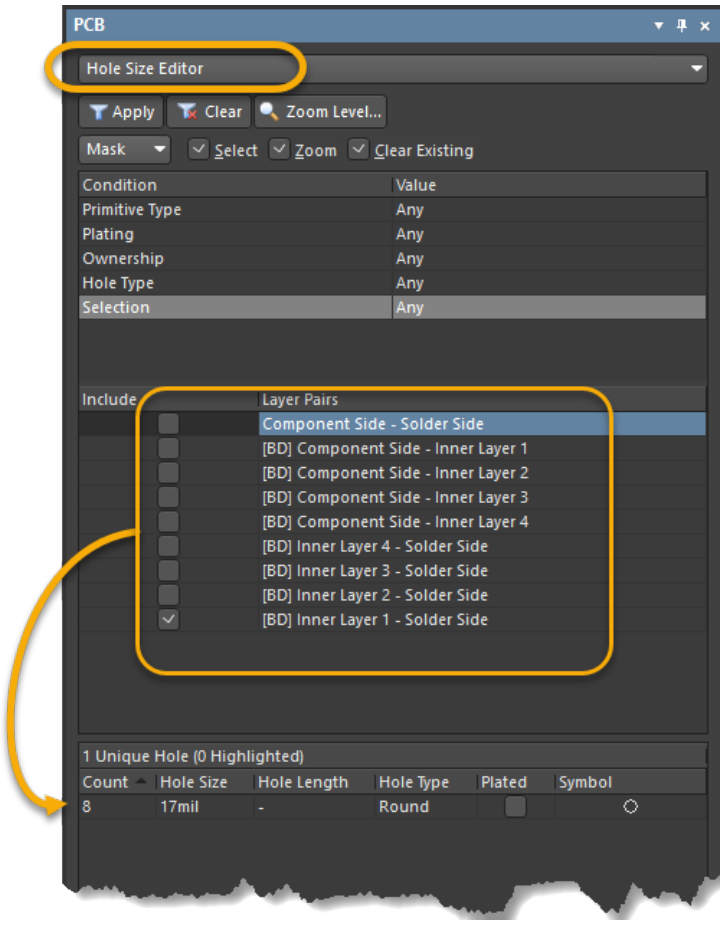

Figure 13. Hole Size Editor for BD Vias

21. Right click on the *Hole Information* pane to generate a **Backdrill Report,** Figure 14.

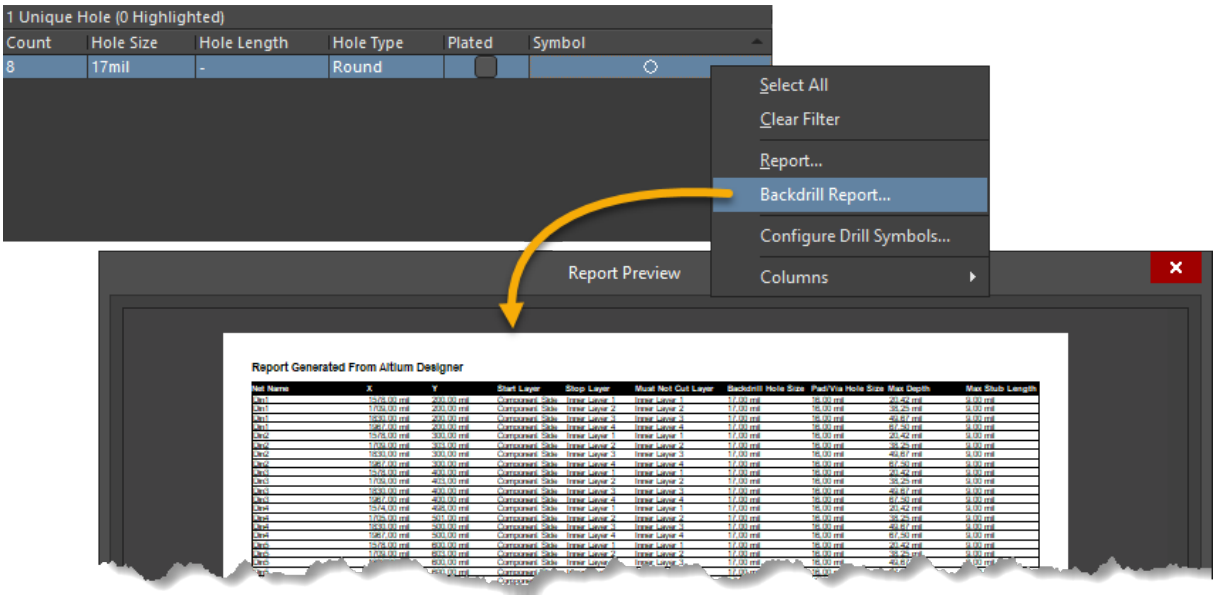

Figure 14. Back Drill Report

22. Right click on the *Hole Information* pane to see and configure the **Drill Symbols,** Figure 15.
23. If the table inside the *Drill Symbol* dialog does not show the information you expect, open the configuration **Grouping** to activate/ deactivate grouping parameters, use Figure 16 as reference.

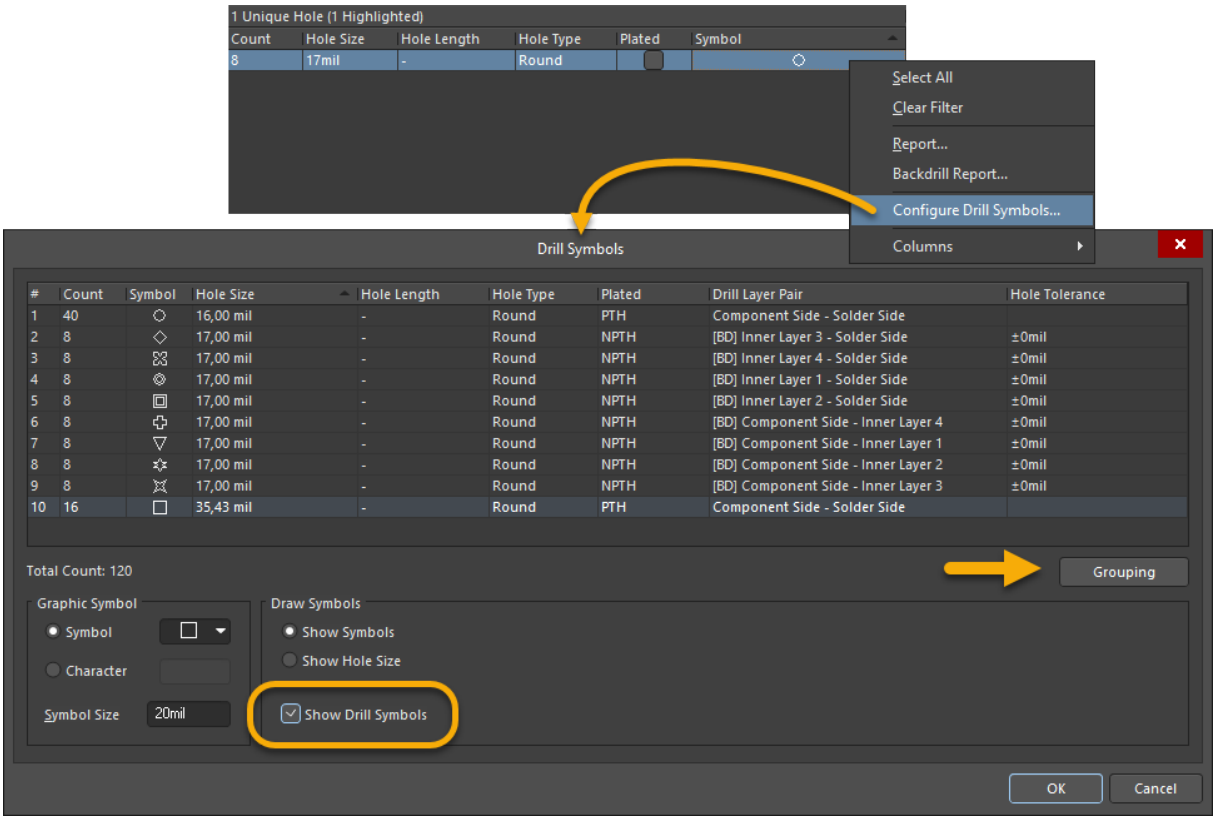

Figure 15. Open Drill Symbol configuration with option for visibility

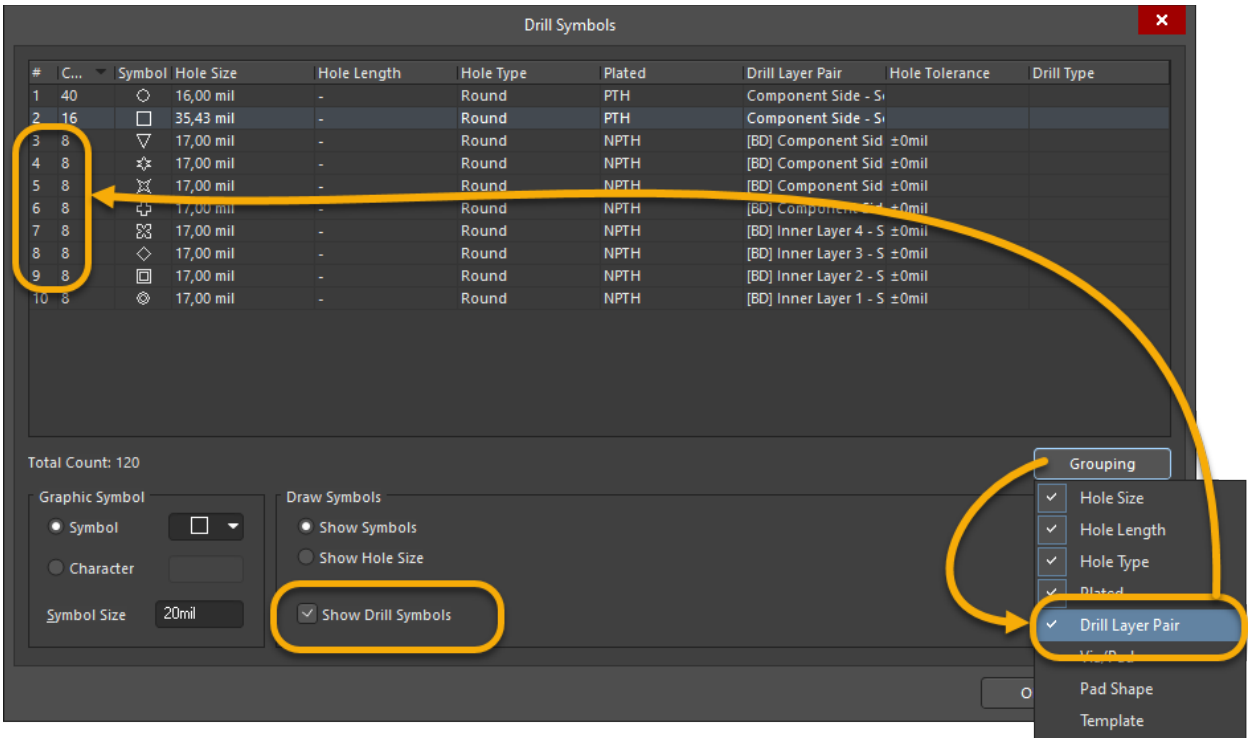

Figure 16 Open Drill Symbol configuration with option for visibility

24. Change to 2D mode, activate the Drill Drawing layer (**View Configuration** - **L**) and now you can control the visibility for the Drill Symbols by clicking on the **up-arrow** icon beside the layer tab, Figure 17.

Figure 17. Configuration for Drill Drawing

25. Close the project and any open documents.

Congratulations on completing module

Back Drills

from the

Altium Designer Advanced Course

Thank you for choosing Altium Designer