



Altium Designer

Advanced Training with Altium 365

Back Drills

Altium
TRAINING





Software, documentation and related materials:

Copyright © 2024 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in the whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties.

TRADEMARKS

ACTIVEBOM®, ActiveRoute®, A365™, Altium 365®, Altium Concord™, Altium Concord Pro™, Altium Designer®, AD™, Altium NEXUS®, Altium OnTrack™, Altium Vault®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Common Parts Library™, Concord™, Concord Pro®, Draftsman®, Dream, Design, Deliver®, DXP™, Easytrax®, EE Concierge®, Fearless HDI™, Geppetto®, Gumstix®, Learn, Connect, Get Inspired™, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, OnTrack™, Overo®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, Upverter®, X2®, XSignals® and their respective logos are trademarks or registered trademarks of Altium LLC or its affiliated companies. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.





Table of Contents

Back Drills	3
1 Purpose	3
2 Shortcuts	3
3 Preparation	4
4 Back Drill – Layer Stack Manager	5
5 Back Drill – Design Rules	8
6 Back Drill – PCB View	10
7 Drill Documentation	12





Back Drills

1 Purpose

Back Drilling, also called Controlled Depth Drilling (CDD) removes unwanted copper stubs from Vias, and if needed, Pads, to improve the signal integrity within PCB transmission lines. For this exercise, we will use 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.

2 Shortcuts


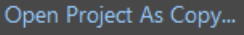

Shortcuts used when working with Back Drills

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





3 Preparation

1. Close all existing projects and documents.
2. Next, create a copy of the Training Project: `Back Drills`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button .
5. Navigate to the predefined Training Project `Back Drills` (`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: `Back Drills - [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 Back Drill – Layer Stack Manager

The basic steps for configuring the PCB for Back Drilling include:

- Creating the needed or desired Drill Pairs that drive the Back Drilling of Vias and 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 and Pads will be converted into the proper types for Back Drilling.

- 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.
- Switching to the `BackDrill.PcbDoc` document, you can see all the routing and the vias used.
- Open 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 four power plane layers.
- From the *Layer Stack Manager* window, select the **Features Button**, then the **Back Drills** button, Figure 1.

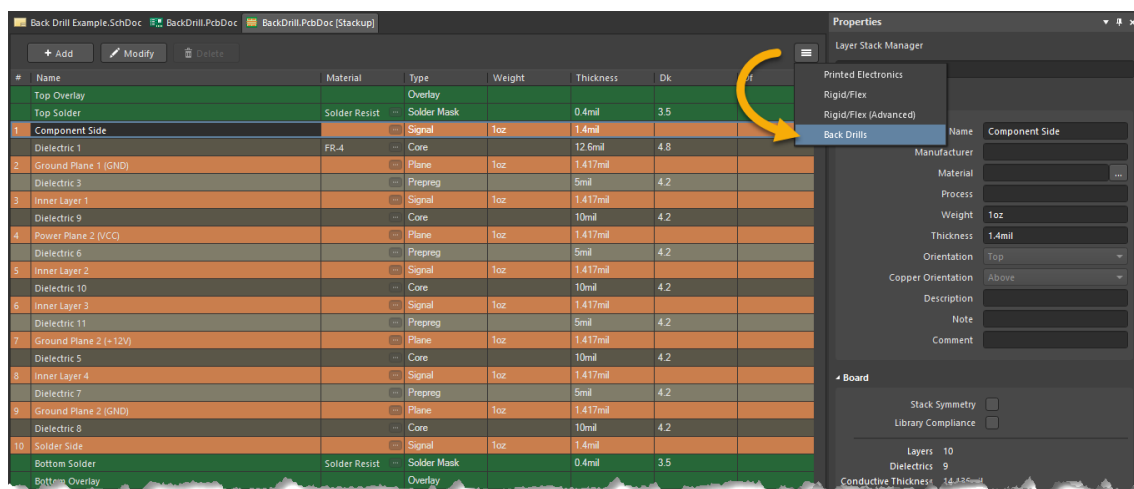


Figure 1. LSM showing 10 Layers and the Back Drills button

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

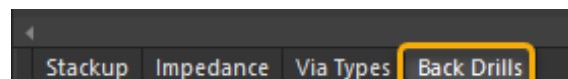


Figure 2. Layer Stack Manager and the Features

14. 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 select the **+ Add** button to create the first Back Drilled via. You will notice the default via is a Component Side back drill, meaning the Top Layer (Component Side) will be back drilled, Figure 3.

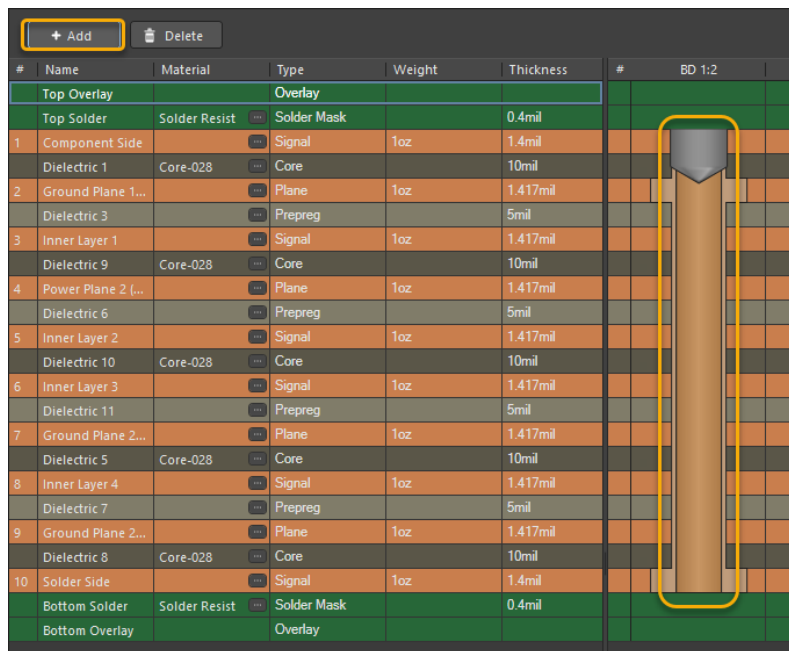


Figure 3. Default Back Drill added to the Layer Stack Manager

15. Once the Back Drill has been added, select it and open the *Properties* panel.
- Leave the *First Layer* as **1 - Component Side**.
 - Now, set the *Last Layer* (or Stop Layer) as **3 - Inner Layer 1**. Notice the back drill display updates to reflect the new Back Drill setting, Figure 4.

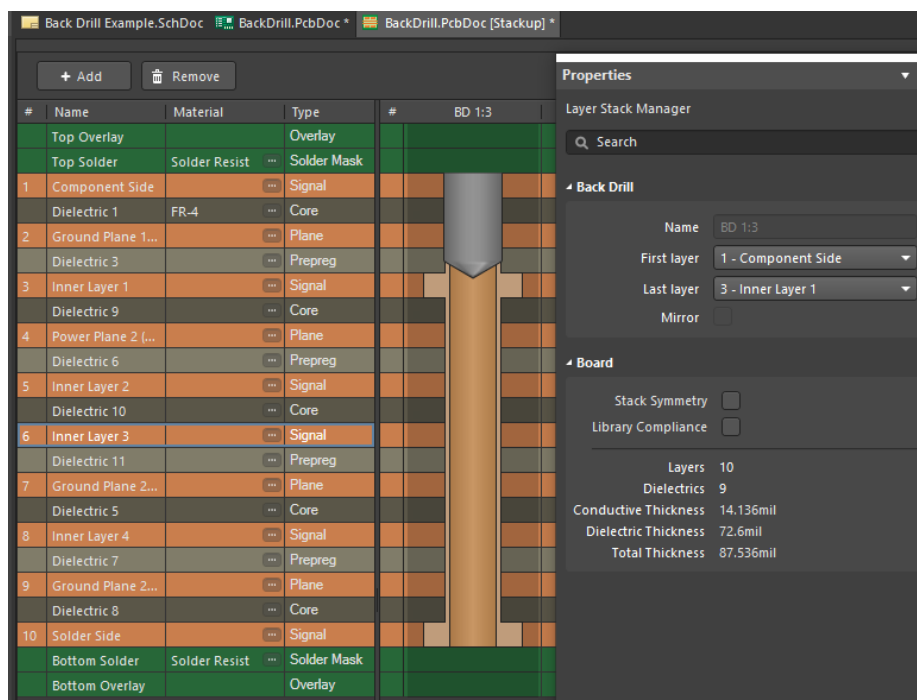


Figure 4. Manually created Back Drill enabled Drill Pair



16. Now, using Figure 5 as your guide, add the remaining Back Drills.

#	Name	Material	Type	#	BD 1:3	BD 1:5	BD 1:6	BD 1:8	BD 10:8	BD 10:6	BD 10:5	BD 10:3
	Top Overlay		Overlay									
	Top Solder	Solder Resist	Solder Mask									
1	Component Side		Signal									
	Dielectric 1	FR-4	Core									
2	Ground Plane 1...		Plane									
	Dielectric 3		Prepreg									
3	Inner Layer 1		Signal									
	Dielectric 9		Core									
4	Power Plane 2 L...		Plane									
	Dielectric 6		Prepreg									
5	Inner Layer 2		Signal									
	Dielectric 10		Core									
6	Inner Layer 3		Signal									
	Dielectric 11		Prepreg									
7	Ground Plane 2...		Plane									
	Dielectric 5		Core									
8	Inner Layer 4		Signal									
	Dielectric 7		Prepreg									
9	Ground Plane 2...		Plane									
	Dielectric 8		Core									
10	Solder Side		Signal									
	Bottom Solder	Solder Resist	Solder Mask									
	Bottom Overlay		Overlay									

Figure 5. Finished set of Back Drills

17. Right-click on tab `BackDrill1.PcbDoc [Stackup]` and select **Save BackDrill.PcbDoc [Stackup]**.

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





5 Back Drill – Design Rules

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

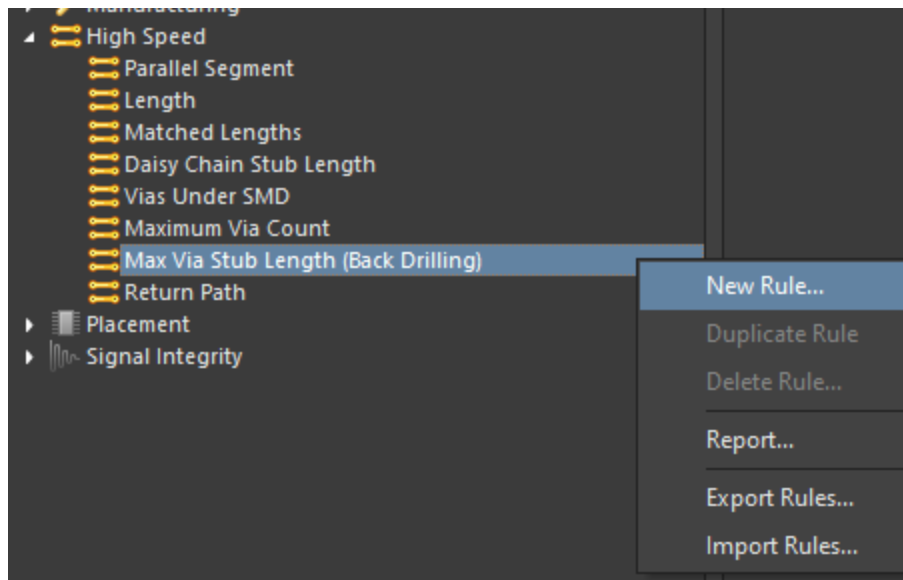


Figure 6. Adding New Max Via Stub Length rule

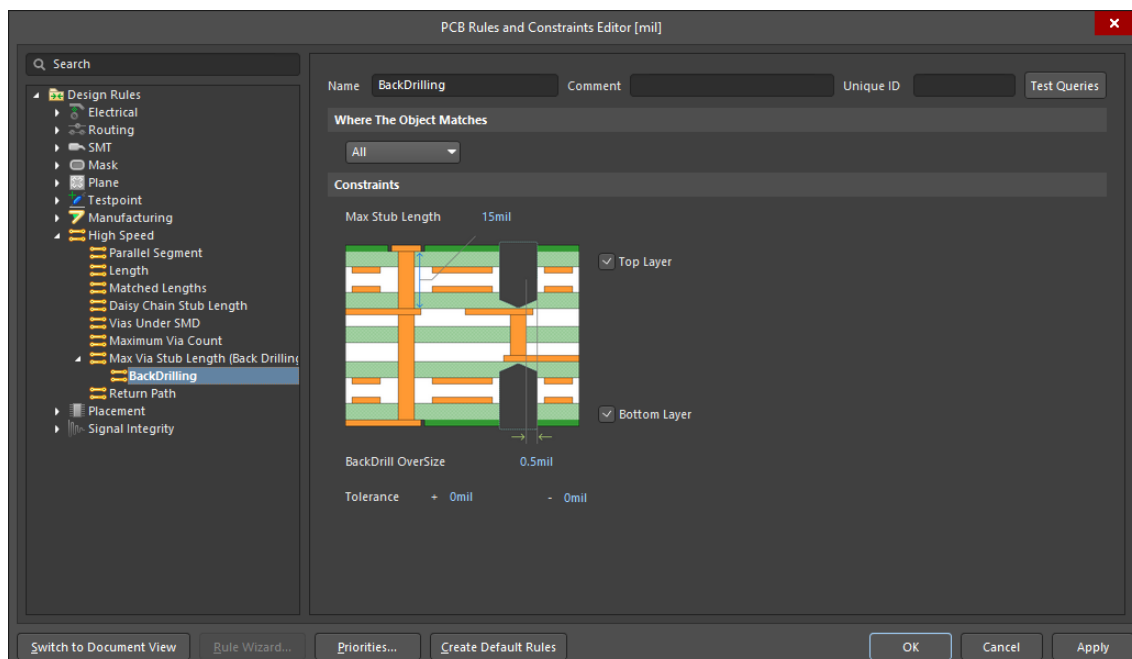


Figure 7. Newly added default Max Via Stub Length Rule



21. Now, you will edit the new default rule to reduce the Max Stub Length to 9mil to ensure that the back drilling will remove redundant vias longer than 9mil.
22. Next, you will limit the Back Drilling to only include the high-speed nets. In order to scope this new rule to your targeted nets, you can use the Object Matching drop down menu and pick **Net Class** – then select the **HSdata net class** from the second pulldown menu. This will limit Drill Pairs to only pads and vias associated with the HSdata nets, Figure 8.

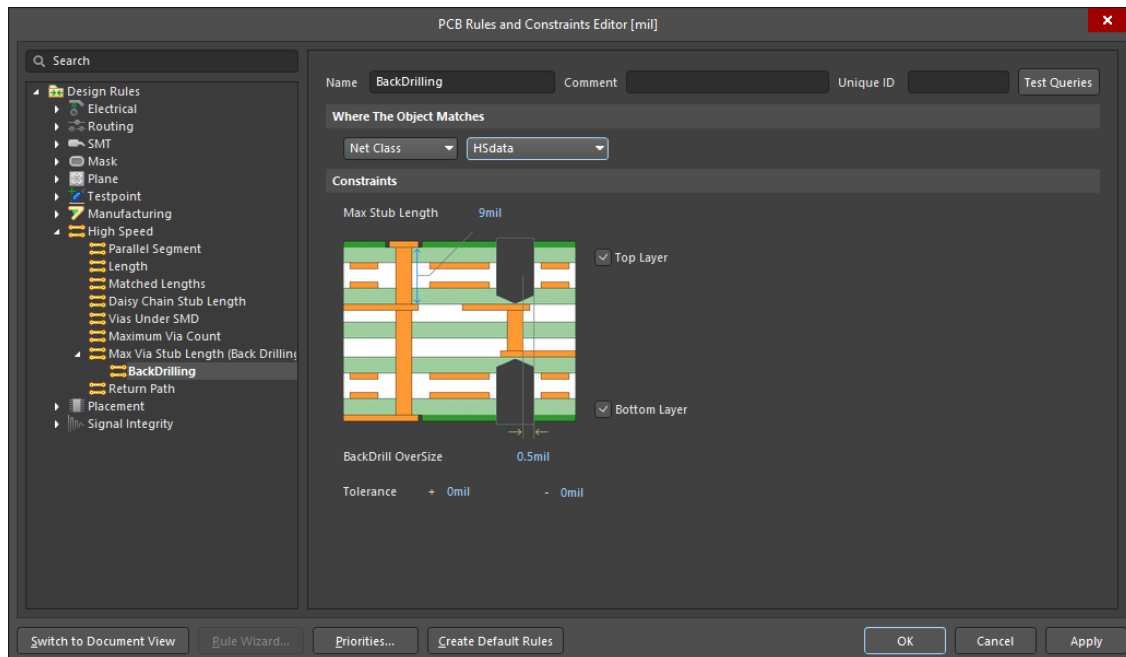


Figure 8. HSdata Net Class Scope for rule

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

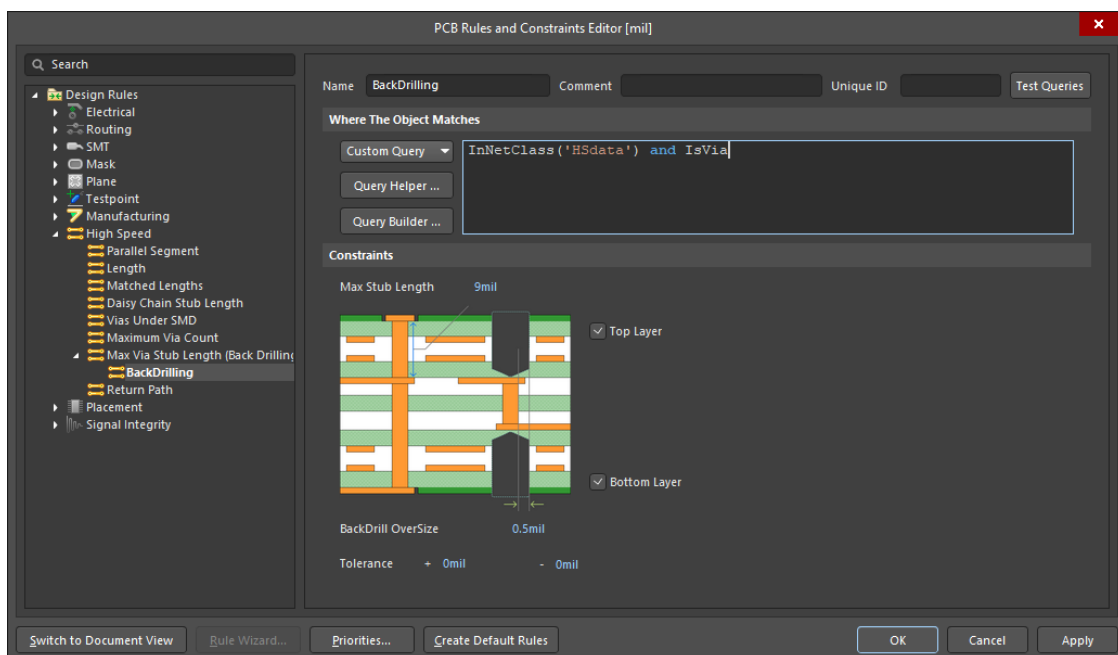


Figure 9. Using Custom Query to limit Rule to Vias with the nets of the Net Class HSdata



6 Back Drill – PCB View

24. Hit **OK** and close the *PCB Rules and Constraint Editor* to see the effect on the PCB vias. In 3D, there are holes for the internal vias but no metal showing on the top as this would be removed by the Back Drilling Operation, Figure 10.

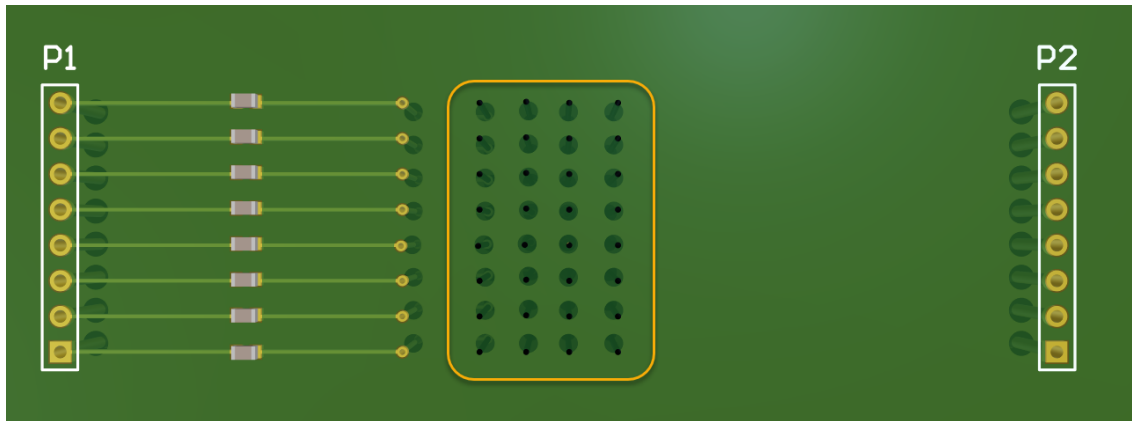


Figure 10. Top of PCB in 3D view showing holes without metal for the Back Drilled vias

25. Switch to the 2D viewing mode and zoom into the back drilled vias. You can see that they have a bicolored inner circle indicating that they are back drilled. The colors indicate which layers are involved, Figure 11.

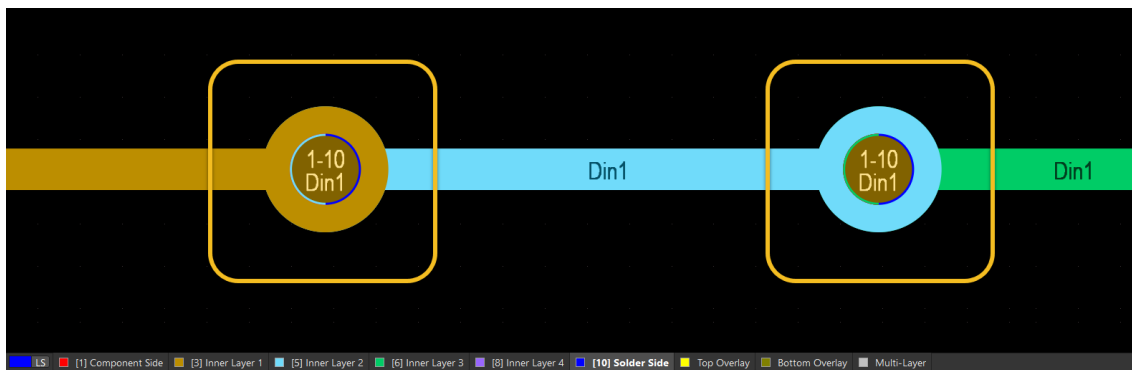


Figure 11. Via color coding for Back Drill





26. Switch back to 3D mode and use the **SHIFT+S** single layer viewing option to cycle through the various layers of the PCB, using the **+** button, as you can see in the series of images below, Figure 12.

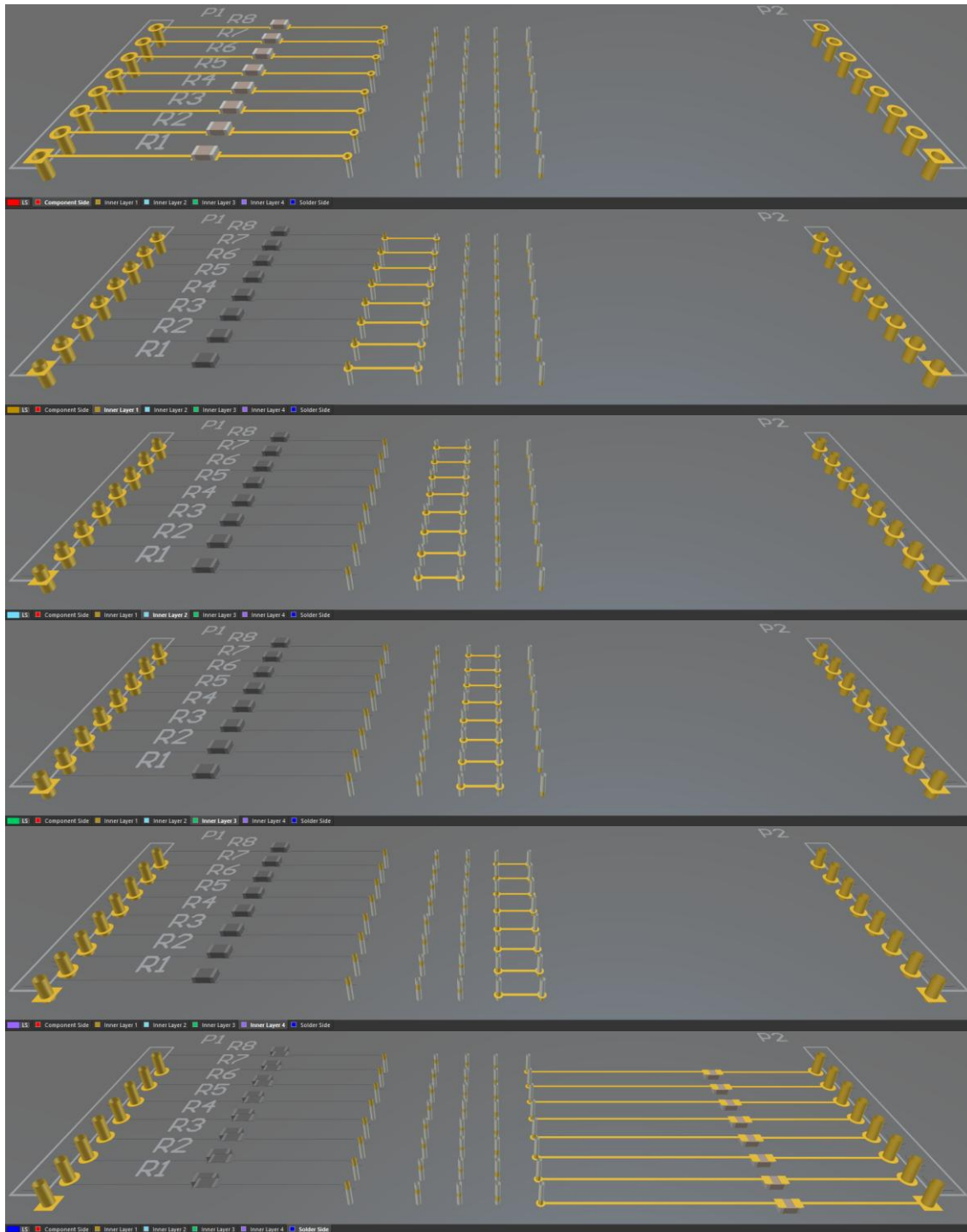


Figure 12. 3D Single Layer Viewing stepping from Top to Bottom layers





7 Drill Documentation

27. When the NC Drill file is generated, there will be multiple files for the Back Drilling. Please ensure that all are sent to the PCB fabricator.
28. 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 and mask individual objects by selecting and deselecting the different objects, Figure 13.

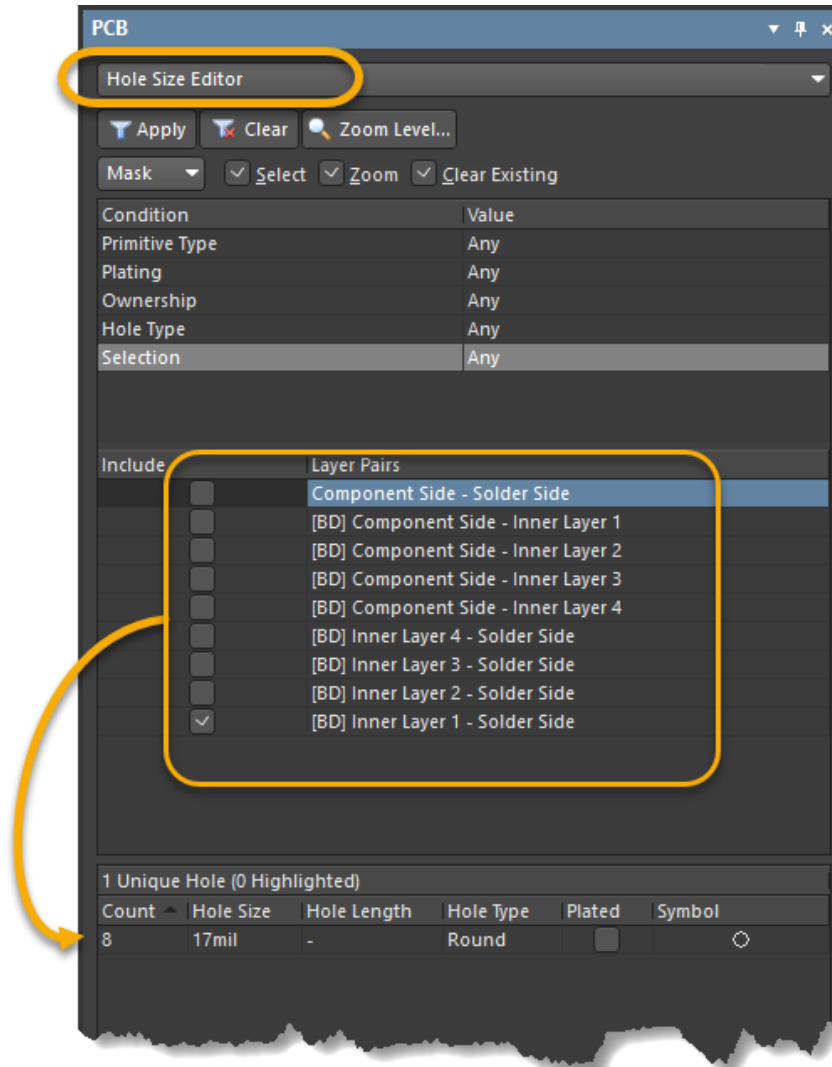


Figure 13. Hole Size Editor for BD Vias



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

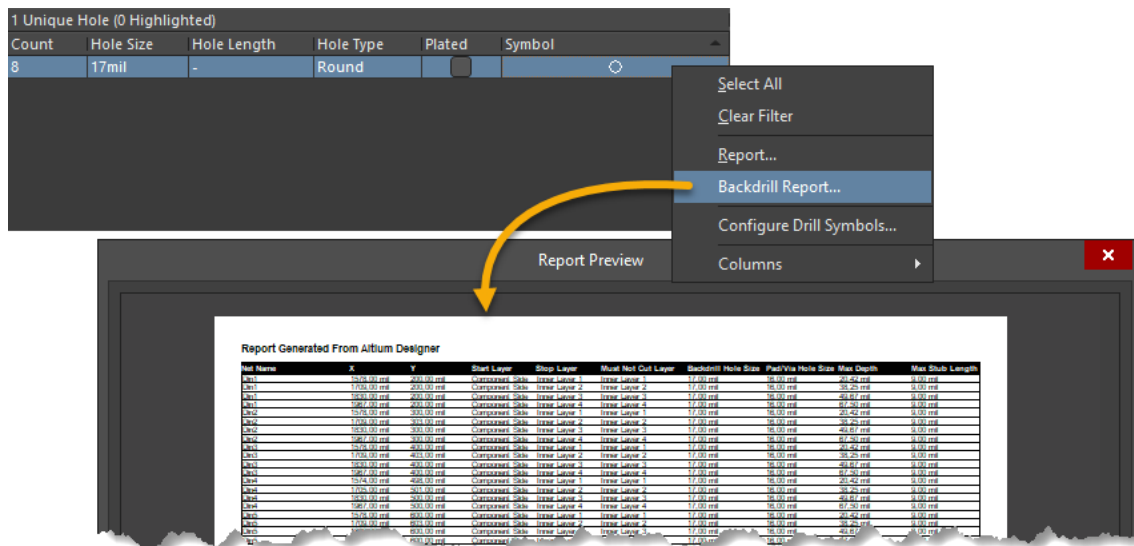


Figure 14. Back Drill Report

30. Right-click on the *Hole Information* pane to see and configure the **Drill Symbols**, Figure 15.

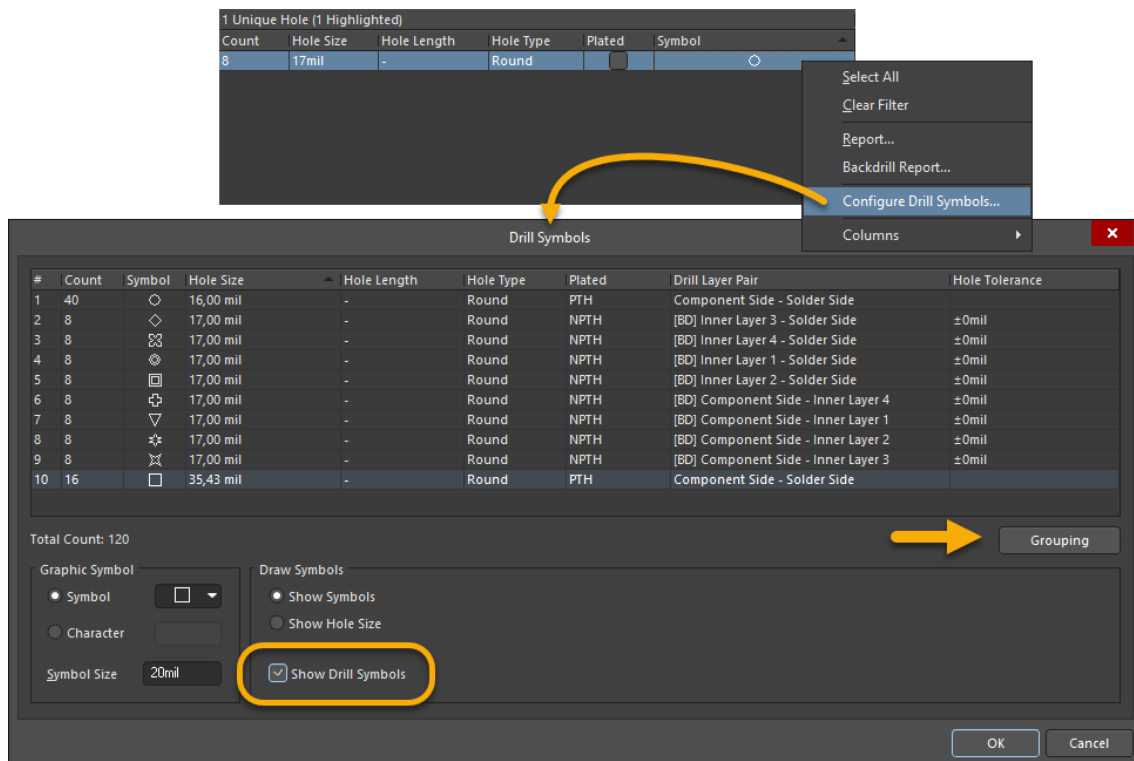


Figure 15. Open Drill Symbol configuration with option for visibility

31. If the table inside the *Drill Symbol* dialog doesn't show the information you expect, open the configuration **Grouping** to activate and deactivate grouping parameters. Use Figure 16 as reference.

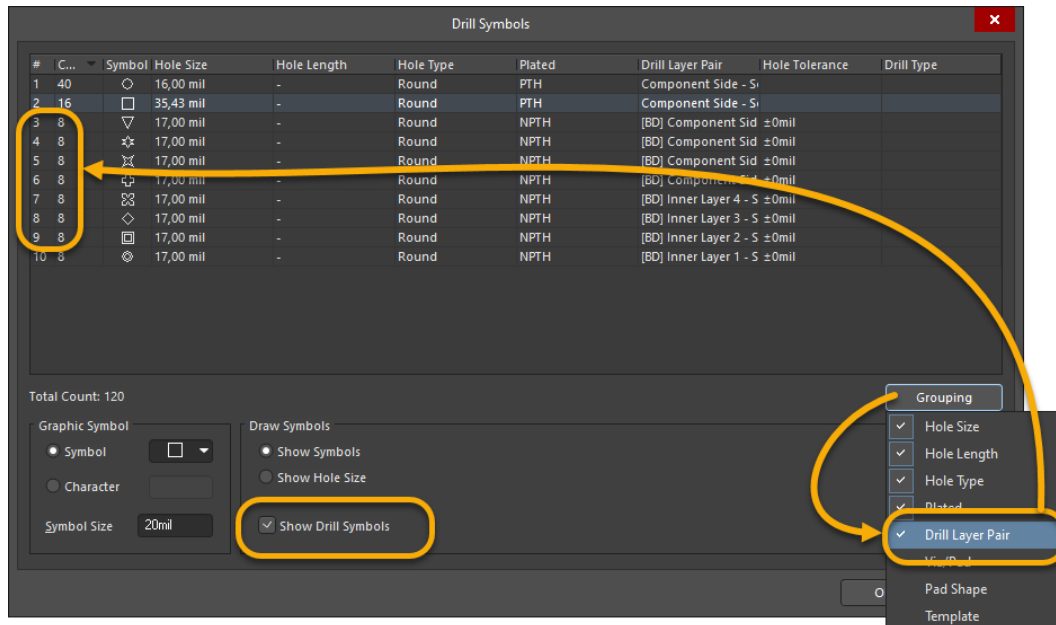


Figure 16. Open Drill Symbol configuration with option for visibility

32. If not already, change to 2D mode, activate the Drill Drawing layer in the **View Configuration panel (L - shortcut key to open panel)** and now you can control the visibility for the Drill Symbols by selecting the **up-arrow** icon beside the layer tab, Figure 17.

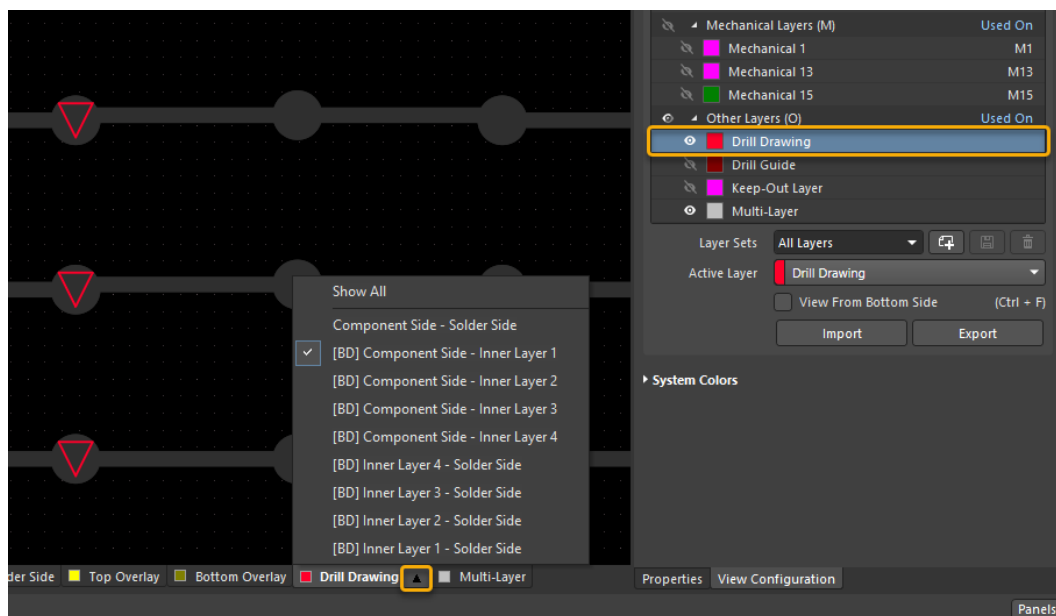



Figure 17. Configuration for Drill Drawing

Note: If the Drill Drawing layer isn't visible, click the layer display arrows  to reveal additional layers that are currently hidden or collapsed.



33. Save all documents using **File » Save All**.
34. Save the modifications to the server:
 - a) In the *Projects* panel, next to the Project name you find the command **Save to Server**
. The button is a small rectangular icon with the text "Save to Server" in blue and a red circular icon to its right.
 - b) Select **Save to Server**.
 - c) In the dialog *Save [Project Name]*:
 - i) Add the comment `Back Drills - [Add Your Name] - Finished`.
 - ii) Select **OK**.
35. When ready, close the project and any open documents, **Window » Close All**.





Congratulations on completing the Module!

Back Drills

from

**Altium Designer Advanced Training
with Altium 365**

Thank you for choosing **Altium Designer**

