

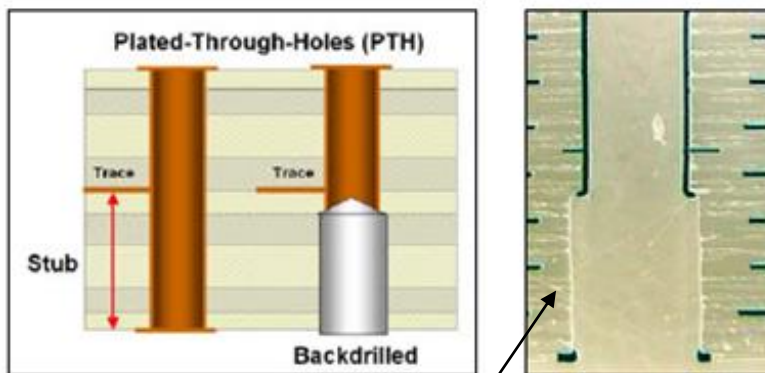
Backdrill

Adds needed functionality to the standard Cadence Backdrill process. The main purpose is to automatically add route keepouts and optionally modify padstack features in the secondary non plated backdrill areas.

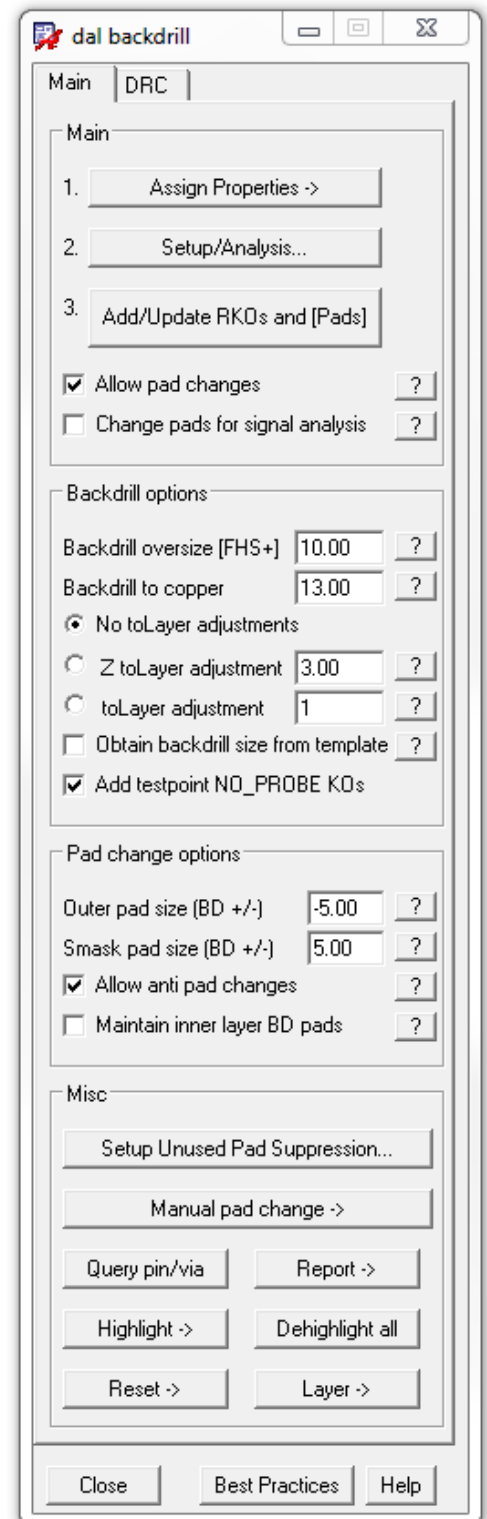
These keepout areas ensure that trace and plane copper does not encroach in the backdrill plunge areas potentially causing shorts and opens.

Features:

- ❑ Makes the process of generating backdrill databases much easier.
- ❑ Automatically creates and updates the route keepouts for the backdrill plunge areas. This prevents shorts and opens.
- ❑ Option to automatically change the padstacks for backdrill requirements.
- ❑ Many options for quickly adding, querying and hi-lighting backdrill properties.
- ❑ Option to remove the BD copper from the barrel along with the external BD pads. This provides a very accurate Allegro database that can be imported into Sigrity for accurate real world signal integrity analysis.



Route keepouts added here along with optional automatic padstack modifications.



Main Options

Add BACKDRILL_MAX_PTH_STUB by select (net)
 Add BACKDRILL_EXCLUDE by select (symbol, pin, via)

 Add BACKDRILL_MIN_PIN_PTH by select (symbol, pin)
 Add BACKDRILL_OVERRIDE by select (symbol, pin, via)

 Add BACKDRILL_PRESSFIT_CONNECTOR by select (symbol)

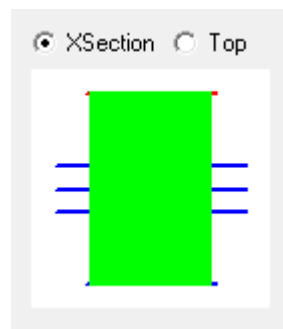
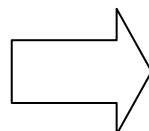
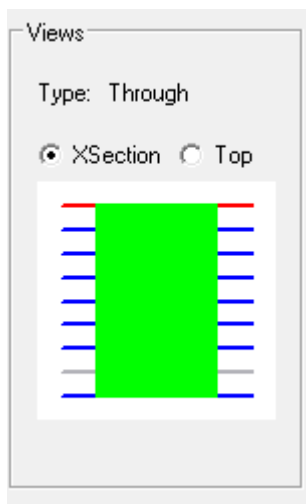
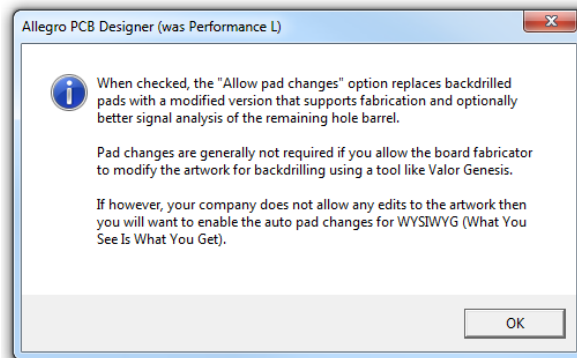
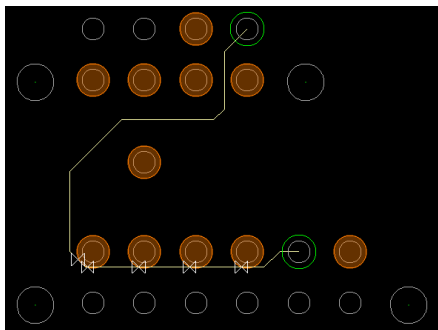
 Add DTS_BACKDRILL_OVERSIZE by select (symbol, pin, via)

 Edit/Query property

Backdrill Setup and Analysis

Backdrill Passes

#	Enable	From Layer	Objects	Passes	To Layer	Depth
1	<input checked="" type="checkbox"/>	Top	Pins&Vias	13	All layers	
2	<input checked="" type="checkbox"/>	Bottom	Pins&Vias	13	All layers	



dal backdrill

Main | DRC

Main

- Assign Properties ->
- Setup/Analysis...
- Add/Update RKO's and [Pads]

☒ Allow pad changes ?

☐ Change pads for signal analysis ?

Backdrill options

Backdrill oversize (FHS+) 10.00 ?

Backdrill to copper 13.00 ?

☒ No toLayer adjustments

☐ Z toLayer adjustment 3.00 ?

☐ toLayer adjustment 1 ?

☐ Obtain backdrill size from template ?

☒ Add testpoint NO_PROBE KO's

Pad change options

Outer pad size (BD +/-) -5.00 ?

Smask pad size (BD +/-) 5.00 ?

☒ Allow anti pad changes ?

☐ Maintain inner layer BD pads ?

Misc

Setup Unused Pad Suppression...

Manual pad change ->

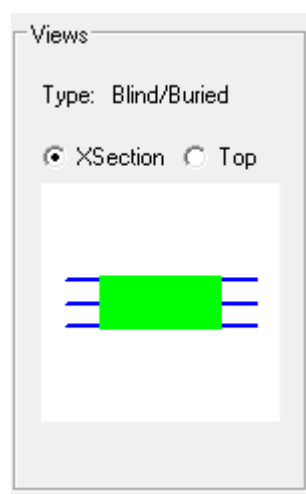
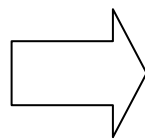
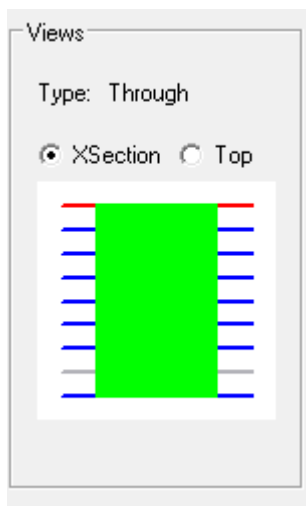
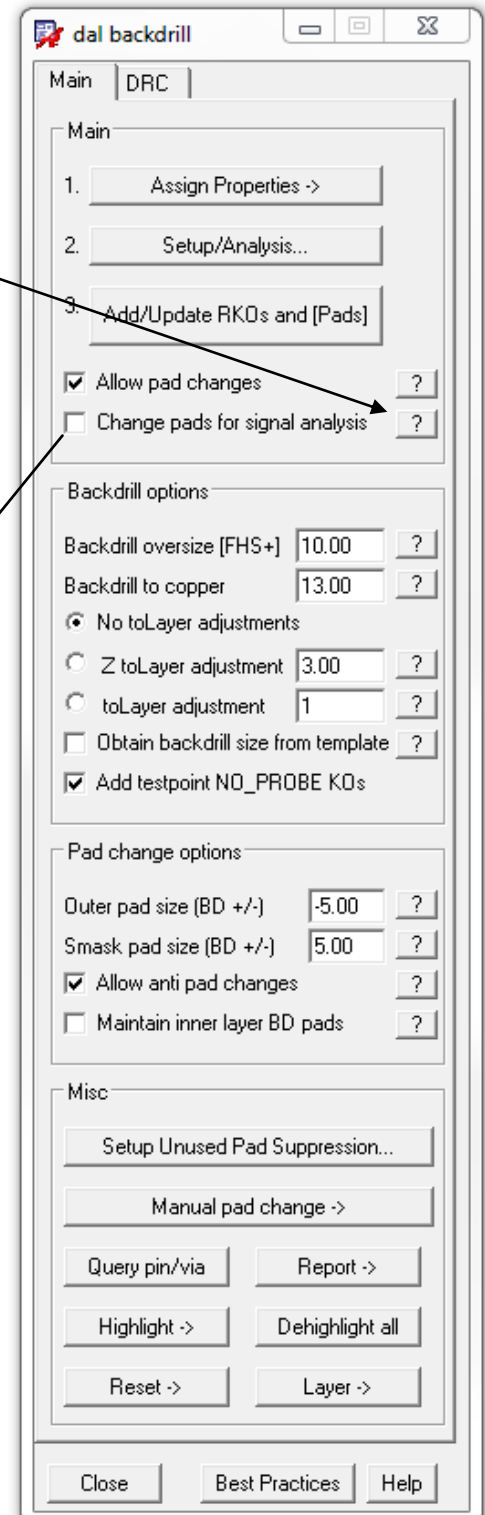
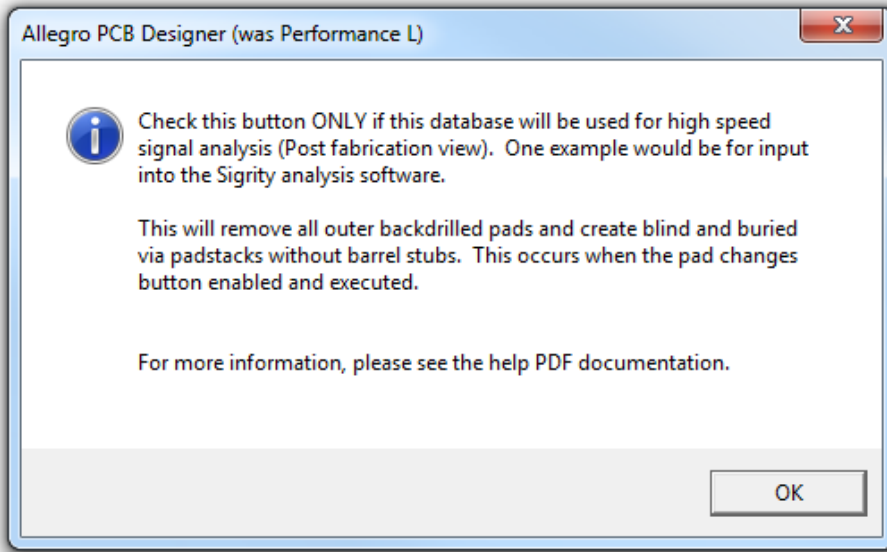
Query pin/via Report ->

Highlight -> Dehighlight all

Reset -> Layer ->

Close Best Practices Help

Main Options (continued)



Backdrill Options

The diagram illustrates the relationship between five informational dialog boxes in Allegro PCB Designer and the 'dal backdrill' tool window. Each dialog box explains a specific backdrill option, and an arrow points from the dialog to the corresponding control in the tool window.

Dialog Box 1 (Top): Explains the "Backdrill oversize [FHS+]" value. It states that this value is added to the existing via or pin finished drill hole size to create a secondary oversized non-plated hole drill operation that removes the PTH barrel stub. An example shows a 40 mil PTH being drilled out with a 50 mil secondary non-plated hole drill. A hint suggests adding the property DTS_BACKDRILL_OVERSIZE to individual symbols, pins, or vias.

Dialog Box 2: Explains the "Backdrill to copper" value. It determines the route keepout clearance added to each side of the backdrill. An example shows a value of 13.0 clearing the copper from the backdrill edge by 13.0 mils on each side.

Dialog Box 3: Explains the "Z to Layer adjustment" value. It determines extra toLayer adjustments for the route keepout and optional replaced padstack layers. An example shows a top backdrill with a toLayer equal to 5 and a Z layer adjustment of 3.0, which would not have route keepouts on layers located less than 3.0 units away from layer 5 in the stackup. The default value is 3.0 mils.

Dialog Box 4: Explains the "toLayer adjustment" value. It determines the extra toLayer adjustment for the route keepout and optional replaced padstack layers. A note states that the toLayer Cadence selects is already one layer below any layer that is connected on the backdrilled pin or via. An example shows a top backdrill with a toLayer equal to 5 and an adjustment of 1, which would have added route keepout shapes down to layer 4 instead of layer 5. Another example shows a bottom backdrill with a toLayer equal to 5 and an adjustment of 1, which would have added route keepouts down to layer 6 instead of layer 5. The default value is 0.

Dialog Box 5 (Bottom): Explains the "Obtain backdrill size from template" feature. It states that when checked, the backdrill oversize values may be unique per FHS and are obtained from the table file: Ex: backdrill-mil.dtl or default-mm.dtl. A hint provides the file directory location: SPB_XX\X\share\pcb\text\nclegend.

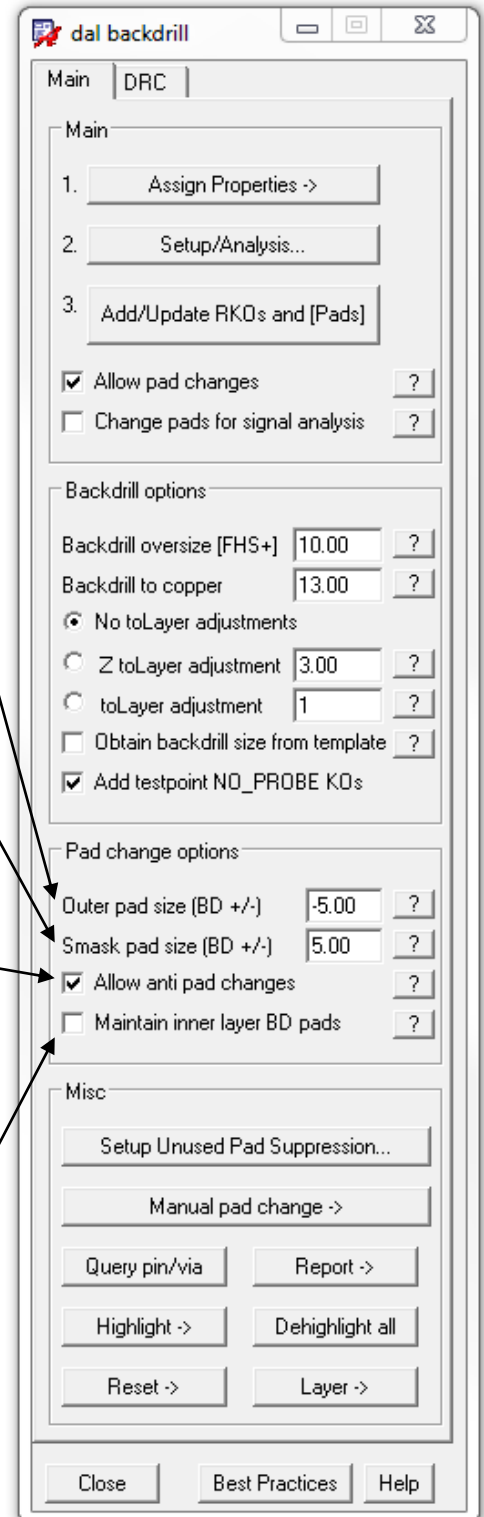
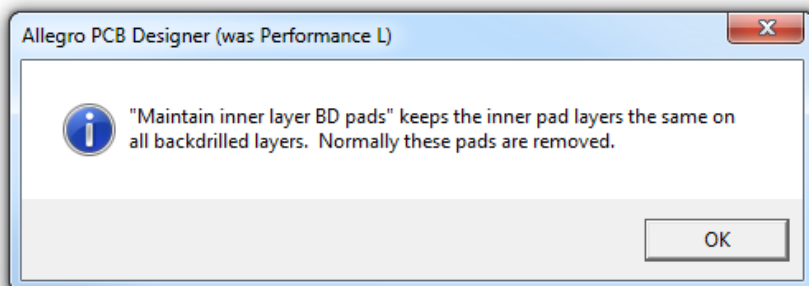
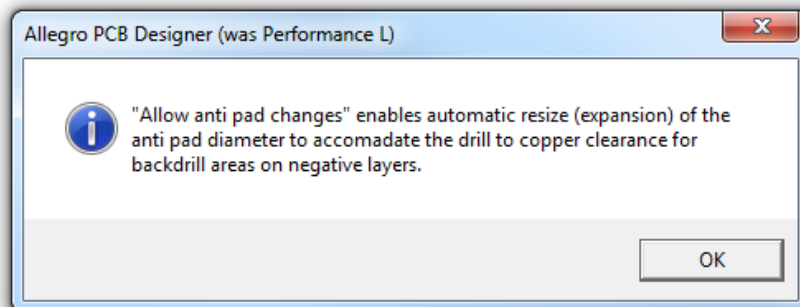
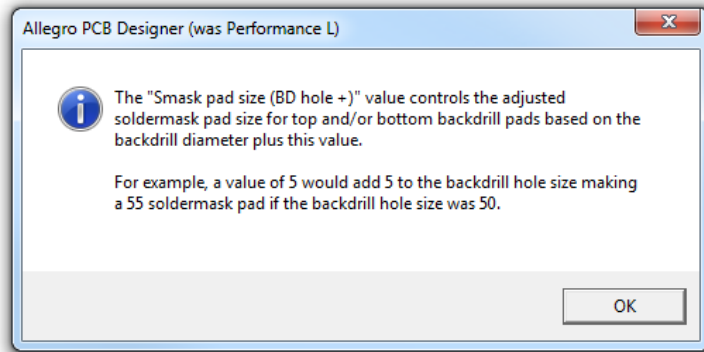
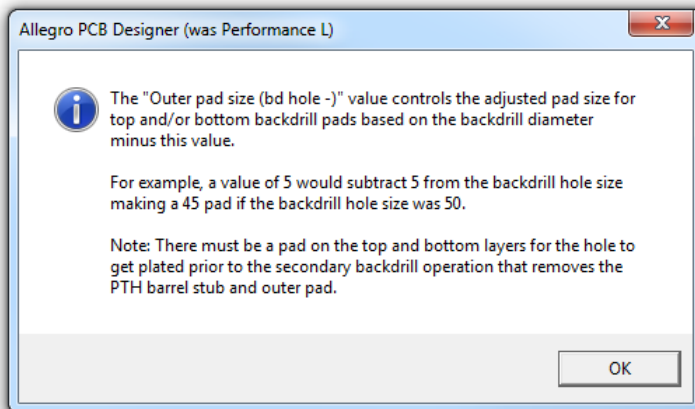
'dal backdrill' Tool Window: This window contains the following sections and controls:

- Main:**
 - Buttons: Assign Properties ->, Setup/Analysis..., Add/Update RKDs and [Pads]
 - Checkboxes: ☒ Allow pad changes, ☐ Change pads for signal analysis
- Backdrill options:**
 - Backdrill oversize [FHS+]: 10.00
 - Backdrill to copper: 13.00
 - ☒ No toLayer adjustments
 - ☐ Z toLayer adjustment: 3.00
 - ☐ toLayer adjustment: 1
 - ☐ Obtain backdrill size from template
 - ☒ Add testpoint NO_PROBE KDs
- Pad change options:**
 - Outer pad size (BD +/-): -5.00
 - Smask pad size (BD +/-): 5.00
 - ☒ Allow anti pad changes
 - ☐ Maintain inner layer BD pads
- Misc:**
 - Buttons: Setup Unused Pad Suppression..., Manual pad change ->
 - Buttons: Query pin/via, Report ->
 - Buttons: Highlight ->, Dehighlight all
 - Buttons: Reset ->, Layer ->
- Bottom Buttons:** Close, Best Practices, Help

Arrows from the dialog boxes point to the following controls in the 'dal backdrill' window:

- Dialog 1 points to the "Backdrill oversize [FHS+]" field.
- Dialog 2 points to the "Backdrill to copper" field.
- Dialog 3 points to the "Z toLayer adjustment" radio button and its associated field.
- Dialog 4 points to the "toLayer adjustment" radio button and its associated field.
- Dialog 5 points to the "Obtain backdrill size from template" checkbox.

Pad Change Options



Misc Options

Unused Pads Suppression

Layer	Type	Negative Artwork	Pins	Vias
TOP	CONDUCTOR			
LAYER2	CONDUCTOR			
LAYER3	PLANE			
LAYER4	CONDUCTOR			
LAYER5	CONDUCTOR			
LAYER6	PLANE			

Report

```

File Close Help
Status = excludePinSide
Tolayer = 0
Remainstubs = 0.0
Lengthstubs = 0.0
Depthdrill = 0.0
Maxstubs = 10.0
Minpth = 0.0
Bdside = "top"
Objtype = "pin"
Drillsizes = 71.0
Backdrilldiameter = 81.0
Backdrillrkoradius = 54.0
Dtsbackdrilloversize = 0.0
Rkolays =
Netprops = ((BACKDRILL_MAX_PTH_STUB "10 MIL"))
Symprops =
Viaiprops =

Status = t
Tolayer = 5
Remainstubs = 0.0
Lengthstubs = 110.1
Depthdrill = 110.1
Maxstubs = 10.0
Minpth = 0.0
Bdside = "bottom"
Objtype = "pin"
Drillsizes = 71.0
Backdrilldiameter = 81.0
Backdrillrkoradius = 54.0
Dtsbackdrilloversize = 0.0
Rkolays = (14 13 12 11 10 9 8 7 6 5)
Netprops = ((BACKDRILL_MAX_PTH_STUB "10 MIL"))
Symprops =
Viaiprops =

EOF
  
```

Options

dal pinChange

Pin names

From: *

Name sel ...

To: P111SQD071STD

Name sel ...

Change mode

All pins Highlighted pins

☐ Ignore FIXED property

☐ Only TOP Backdrills

☐ Only BOTTOM Backdrills

Highlight pins

HL like selected pin DHL all pins

HL all changed pins (exploded)

Report -> Help

dal backdrill

Main DRC

Main

1. Assign Properties ->

2. Setup/Analysis...

3. Add/Update RKO's and [Pads]

☒ Allow pad changes ?

☐ Change pads for signal analysis ?

Backdrill options

Backdrill oversize [FHS+] 10.00 ?

Backdrill to copper 13.00 ?

☒ No toLayer adjustments

☐ Z toLayer adjustment 3.00 ?

☐ toLayer adjustment 1 ?

☐ Obtain backdrill size from template ?

☒ Add testpoint NO_PROBE KO's

Pad change options

Outer pad size (BD +/-) -5.00 ?

Smask pad size (BD +/-) 5.00 ?

☒ Allow anti pad changes ?

☐ Maintain inner layer BD pads ?

Misc

Setup Unused Pad Suppression...

Manual pad change ->

Query pin/via Report ->

Highlight -> Dehighlight all

Reset -> Layer ->

Close Best Practices Help

All Backdrill nets [BACKDRILL_MAX_PTH_STUB]

BACKDRILL_EXCLUDE (symbol, pin, via)

BACKDRILL_MIN_PIN_PTH (symbol, pin)

BACKDRILL_OVERRIDE (symbol, pin, via)

BACKDRILL_PRESSFIT_CONNECTOR (symbol)

DTS_BACKDRILL_OVERSIZE (symbol, pin, via)

All Backdrill vias and pins (Top)

All Backdrill vias and pins (Bottom)

All failed backdrills

Backdrill pins/vias toLayer - Bottom 3 Qty [4]

Backdrill pins/vias toLayer - Bottom 5 Qty [2]

Backdrill pins/vias toLayer - Bottom 6 Qty [2]

Backdrill pins/vias toLayer - Bottom 9 Qty [2]

Backdrill pins/vias toLayer - Bottom 11 Qty [1]

Backdrill pins/vias toLayer - Bottom 14 Qty [2]

Backdrill pins/vias toLayer - Top 0 Qty [12]

Backdrill pins/vias toLayer - Top 4 Qty [1]

Delete all backdrill RKO shapes

Reset all pads to library prefix

Delete RKO and reset all pads

Reset help

Misc Options

Report .csv

Layer stackup data (Text)

Layer stackup data (Excel)

MANUFACTURING/BACKDRILL-FLAG-TOP (violations-top)

MANUFACTURING/BACKDRILL-FLAG-BOT (violations-bot)

MANUFACTURING/NCBACKDRILL-14-3

MANUFACTURING/NCBACKDRILL-14-6

MANUFACTURING/NCBACKDRILL-14-5

MANUFACTURING/NCBACKDRILL-14-9

MANUFACTURING/NCBACKDRILL-1-4

MANUFACTURING/NCBACKDRILL-14-11

TOP Layers

BOTTOM Layers

cadence™

Best Practices: Working with Backdrilling

Series XL and GXL

Product Version 16.0
June 2007

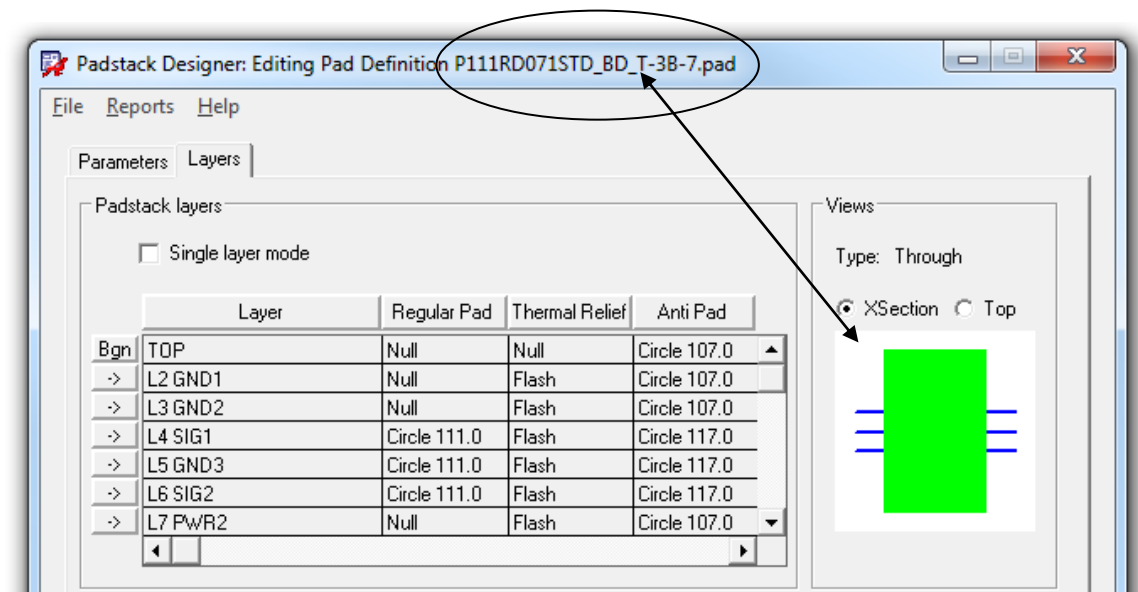
The screenshot shows the 'dal backdrill' application window with the 'DRC' tab selected. The 'Main' section contains three numbered steps: 1. Assign Properties ->, 2. Setup/Analysis..., and 3. Add/Update RKDs and [Pads]. Below these are checkboxes for 'Allow pad changes' (checked) and 'Change pads for signal analysis' (unchecked). The 'Backdrill options' section includes 'Backdrill oversize [FHS+]' (10.00), 'Backdrill to copper' (13.00), and radio buttons for 'No toLayer adjustments' (selected), 'Z toLayer adjustment' (3.00), and 'toLayer adjustment' (1). There are also checkboxes for 'Obtain backdrill size from template' (unchecked) and 'Add testpoint NO_PROBE KDs' (checked). The 'Pad change options' section includes 'Outer pad size (BD +/-)' (-5.00), 'Smask pad size (BD +/-)' (5.00), and checkboxes for 'Allow anti pad changes' (checked) and 'Maintain inner layer BD pads' (unchecked). The 'Misc' section contains buttons for 'Setup Unused Pad Suppression...', 'Manual pad change ->', 'Query pin/via', 'Report ->', 'Highlight ->', 'Dehighlight all', 'Reset ->', and 'Layer ->'. At the bottom are 'Close', 'Best Practices', and 'Help' buttons. Arrows indicate that the 'Report ->' button corresponds to 'Report .csv', the 'Layer ->' button corresponds to 'Layer stackup data (Text)', and the 'Layer ->' button corresponds to 'Layer stackup data (Excel)'. The 'Best Practices' button corresponds to the 'Best Practices: Working with Backdrilling' document.

Examples

The following drill control files may be used to select the backdrill size table:

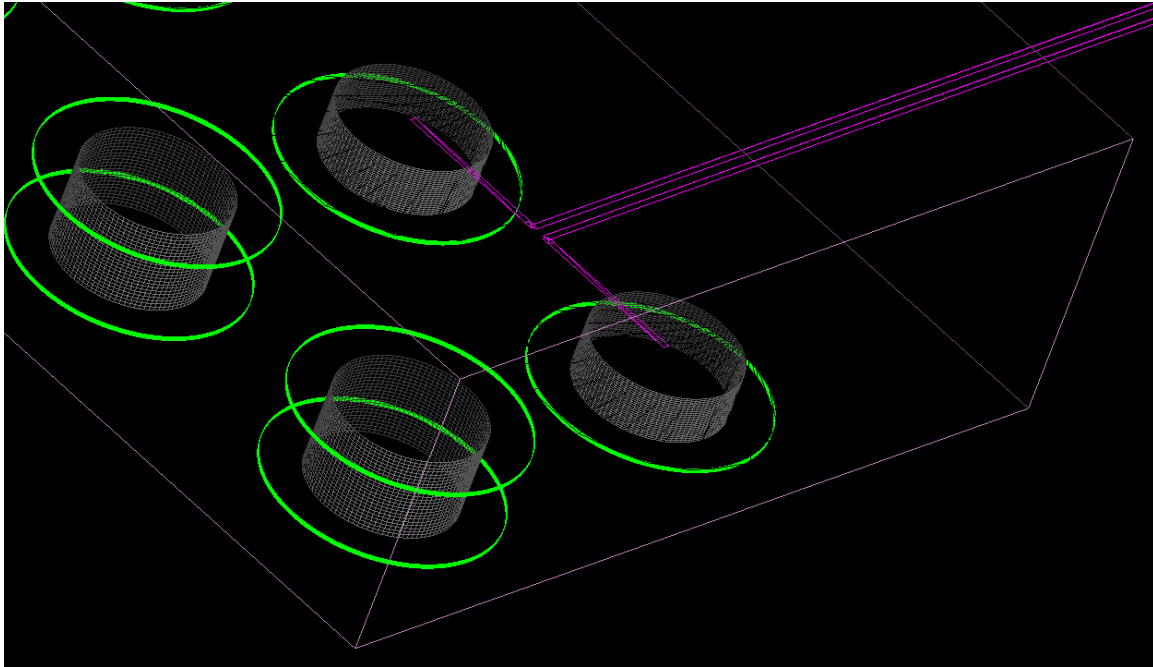
backdrill-mil.dlt or default-mm.dlt . They are located in the SPB_XX.X\share\pcb\text\nclegend directory.

```
?ColumnDefinitions '(
    ("Figure"      "FIGURE"      7)
    ("Holesize"    "FH SIZE"      10)
    ("Tolerance"   "TOLERANCE"    15)
    ("PlateStatus" "PLATED"       15)
    ("NonStandard" "NONSTANDARD"  15)
    ("Quantity"    "QTY"          15)
    ("User"        "DRILL SIZE"    10)
    ("User"        "BACKDRILL DRILL SIZE" 20)
)
;
; ?CustomData
; -----
;
; A column definition in ?ColumnDefinitions above can have "User" appear
; in the first field to indicate a column of user-defined data. The
; data is specified by ?CustomData definitions that are matched to holes
; appearing in the legend table.
;
; ?CustomData '(
;     ( 3.4      "Plated"      "5" 17.7)
;     ( 5.7      "Plated"      "9.8" 19.7)
;     ( 5.8      "Plated"      "9.8" 19.7)
;     ( 6        "Plated"      "9.8" 19.7)
;     ( 7.8      "Plated"      "11.8" 21.7)
```



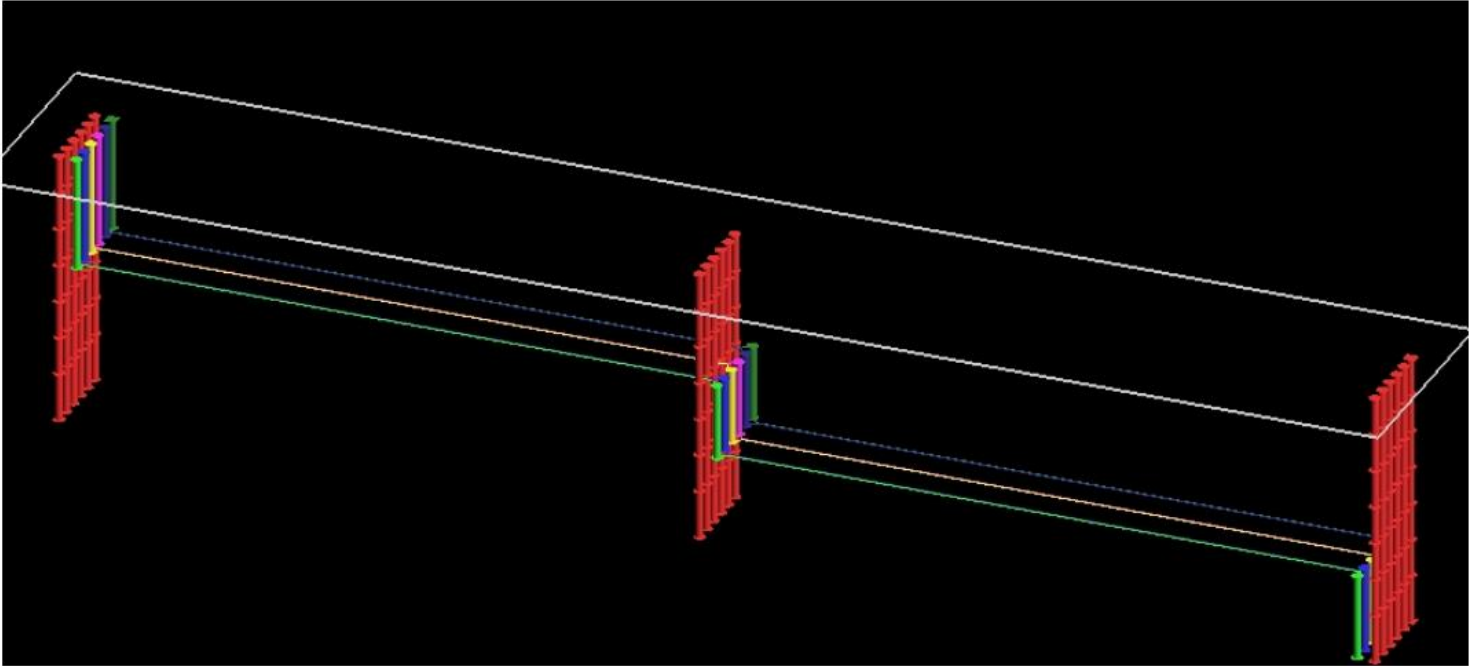
Examples (continued)

View from the Allegro 3D viewer



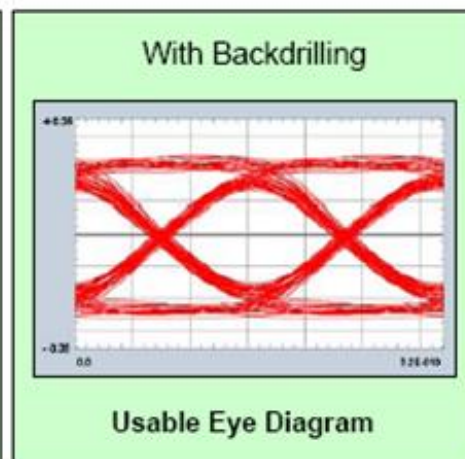
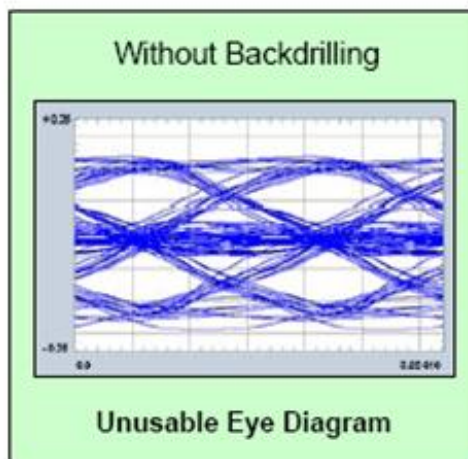
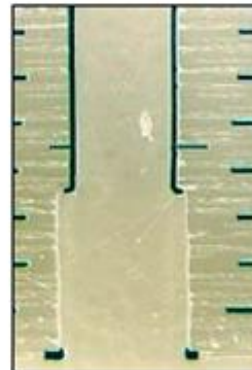
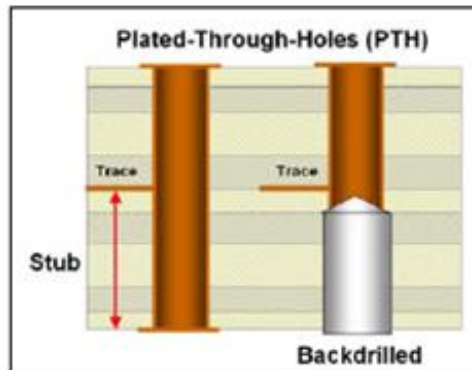
Signal Analysis (Sigrity)

When the modified annular ring is set to zero, the database is configured for real world signal analysis post backdrill operation. Notice the PTH barrels are removed for the backdrill layers.



Signal Integrity

Backdrilling eliminates detrimental plated-through-hole (PTH) via stub effects that distort signals passing through them



6.25 Gb/s Data Rate

Reference



Back-Drilling Technology can be used to create via paths

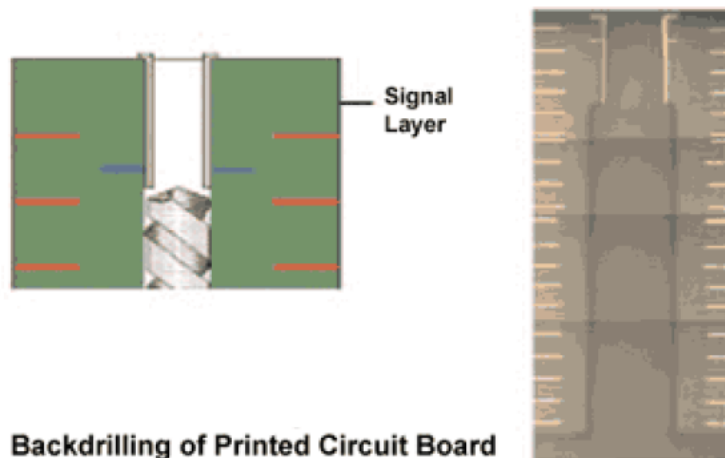
Back drilling, or controlled depth counter boring, is a process where plating is removed from the unused portion of the via. Multi-layer Printed Circuit Boards (PCBs) are processed in a standard manner, adding a secondary drilling operation after plating using PCB CNC drilling equipment with controlled depth capabilities. CNC drill files created from customer data allow this process to be automated and repeatable.

Via delay = PCB thickness .062" = 10 ps & >.200" 30 - 40 ps

Alternative Technique

Back drilling, or counter boring, the via is an alternative technique to minimize the stub. Back drilling is simply drilling out the unused portion of a via to a controlled depth on the same type of mechanical drill equipment used to initially produce the PCB.

Removing the Unused Portion of PTH



Backdrilling of Printed Circuit Board

Via delay = PCB thickness .062" = 10 ps & >.200" 30 - 40 ps

Reference



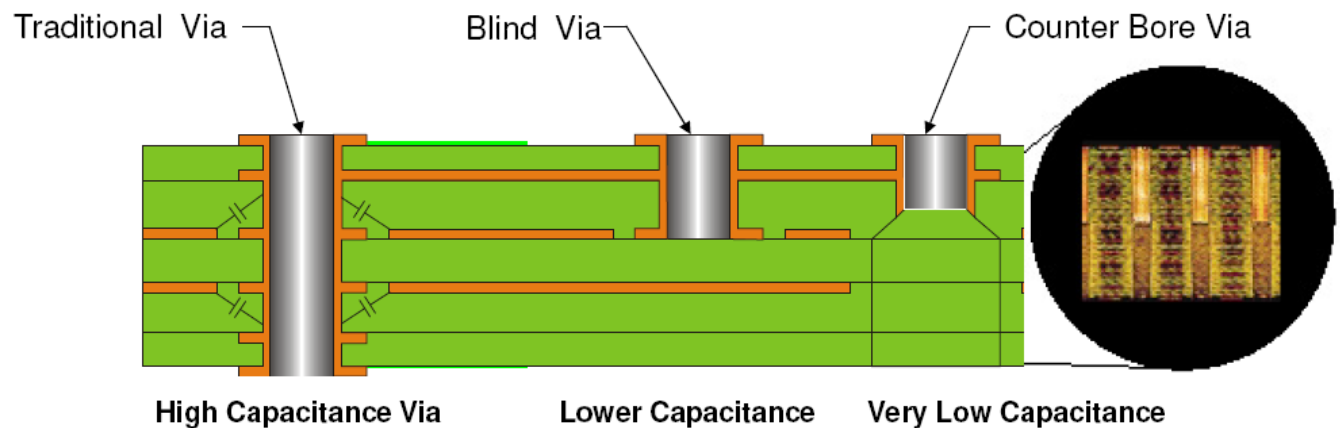
Over Drill Diameter:

One important parameter is the secondary drill diameter. This drill diameter must be greater in diameter than the primary drill to allow removal of all the electrodeposited plated metal, typically copper with an additional surface finish. Minimization of this diameter is important to avoid reduction of routing channels which compromise hole to trace spacing in the pin fields. **The current recommendation is 0.007" - 0.010" over original drill diameter depending upon depth. Drill-to-copper ranges from 0.012" – 0.015".**

Back-drill Depth:

Back drilling is a trade off between manufacturing cost and electrical performance. Contributors to back drill depth variation is mechanical depth and layer position. An optimized process achieves a 3 sigma overall variation of +/- 0.005" to nominal target depth. At least a 0.010" target nominal depth before the last layer connected is recommended.

Reference



Minimizing PTH Capacitance

0.012" (300 μm) drill & 0.022 (550 μm) Pad
.019" (475 μm) Drill diameter

Techniques required at:
2.5 Gb/s > 5 Gb/s

Drill 0.007" (175 μm) over original drill diameter
Z tolerance = + /- 0.005" (150 μm)
Drill location tolerance = + /- 0.002" (50 μm)
PCB thickness = + /- 10%

Sanmina Guidelines



Back-Drilling

- Each depth should be called out on the fabrication drawing. Drill files should be included for each desired depth.
- Back-drill diameters should be 0.010" larger than the initial drilled hole size.
- The non-plated through hole (NPTH) back-drill should be 0.020" minimum from conductor features on inner layers if following IPC recommendations. Sanmina-SCI recommends 0.010" absolute minimum.
- NPTH back-drill to NPTH back-drill spacing should be 0.010" minimum.
- NPTH back-drill to outer layer pad spacing should be 0.015". (0.020" per IPC-6012A).
- The pad size which will be back-drilled should be 0.004" larger than first drill diameter. Breakout is not a concern. Pad size on opposite side should be normal.
- Minimum dielectric between penetrated layer and non-penetrated layer should be 0.006".
- Each layer depth should be given a different drill code and file. It should also specify the initial penetrating layer and the final penetrated layer. E.g., back-drill from layer 14 through layer 9.

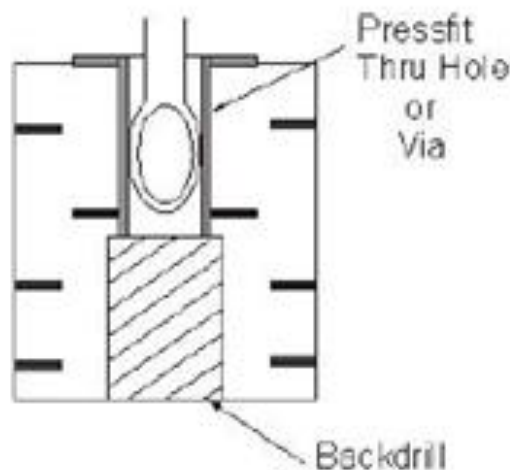


Figure 36 Back-drilling example

