A standard element in the package of data provided to the PCB fabricator is a drill drawing. The actual drill instructions are provided in a drill file, which presents the data in a format suitable for loading into an NC drill machine. To provide that same information in a human-friendly, visual format, designers will typically include a drill drawing.

The drill drawing is an outline of the board with each drill site marked by a small symbol. Next to the board image is a table, referred to as the drill table, that lists the symbols, the hole size assigned to that symbol, and well as other important hole properties, such as plated / non-plated, or the drill layer-pair that hole passes between.

Altium Designer 15.1 sees the introduction of new and enhanced features that improve and accelerate the Drill Drawing creation process.

- Drill symbols are now assigned in the Drill Symbols dialog, which offers a high-level of control over the criteria used to group sizes.
- The drill drawing is live - add a new hole and it is automatically added to the drill drawing and the Drill Table.
- Drill pairs are fully supported, the Create Drill Drawing command
Displaying the Live Drill Drawing

No longer do you need to generate output to see the drill drawing, it can be examined at any stage, directly in the PCB editor. This does require that the drill symbols are enabled for display, which happens automatically if a drill table has been placed, or if you have enabled the **Always show Drill Symbols** option in the **Drill Symbols** dialog (described below).

The drill drawing layer then becomes live - showing a drill symbol at each drill site. To view the drill drawing, make the Drill Drawing layer the active, or current layer, then switch the workspace display to single layer mode (**Shift+S**).

With the Drill Drawing layer active, switch to single layer mode (**Shift+S**) to examine the drill drawing.

There are up to three different single layer display modes available, enable your preferred mode(s) on the **PCB Editor - Board Insight Display page** of the **Preferences dialog**. The **Shift+S** shortcut cycles through the enabled modes, and back to the standard display mode.

Designs that include Multiple Drill Pairs

If the design includes multiple drill pairs, for example it uses blind and/or buried vias, then you can display (and print) multiple drill drawings, one for each drill pair. In the PCB editor you can control the display of the drill drawing for each drill pair using the small triangle that appears automatically when the design includes multiple drill pairs.
Drill Drawing Layer Tab Menu

As well as the special drill pair menu just described, the Drill Drawing layer tab also has a right-click menu (as all layer tabs do). As well including the standard layer display control options, this menu is also used to access useful drill-specific commands, including:

- **Configure Drill Pairs** - it is the presence of drill pairs that determines if multiple drill drawings are required, select this menu command to configure these in the Drill Pair Manager dialog.
- **Configure Drill Symbols** - this command opens the Drill Symbols dialog, where you define the grouping criteria used to assign symbols to holes.
Configuring the Drill Symbols

Dialog page: Drill Symbols

Drill symbols are assigned to hole sizes in the Drill Symbols dialog. The assignment of a symbols to hole sizes is performed automatically. Each symbol is mapped to a hole size based on the currently enabled grouping options, click the Grouping button to enable the grouping options you want used as criteria for determining uniqueness of hole size. Hole objects must share the same values in all visible grouping columns to be considered the same, if they are they will be assigned to use the same symbol.

For example, pads and vias with the same size hole will share a symbol, but if the Via/Pad option is enabled in the Grouping list (so that the Via/Pad column is displayed), then they will be listed separately and be assigned different symbols.
Enable the attributes you want used to determine uniqueness of each hole size.

The **Symbol Size** setting defines the size of the symbol displayed on the Drill Drawing layer, the value entered applies to all of the symbols. To learn more about configuring drill symbols, refer to the [Drill Symbols dialog page](#).

---

**Improved Drill Table**

*Main article: Drill Table*

When you place a Drill Table into the workspace (**Place » Drill Table**), it displays with the default columns, font, units and alignment defined on the [PCB Editor - Defaults page](#) of the [Preferences dialog](#). If there are non-circular shapes present in the design (such as slots or punched (rectangular) holes), then the **Hole Length** column is automatically added during placement.

The display of columns is then configured via the right-click menu available in the grid region of the [Drill Table dialog](#).
Configure the columns, including their display, order, text alignment, and font properties in the Drill Table dialog.

The line width of the symbols is based on the Text Width setting in the Drill Table dialog. This setting is available when the Font Kind is set to Stroke, it will still be obeyed if you set the Font Kind back to TrueType.

### Defining a Drill Table when there are Multiple Drill Pairs

The Drill Table supports multi-layer boards that use layer-pair drilling. To support the designer in using the Drill Table during the design process the table includes the Layer pairs to preview option, set this to restrict the table to only show the drill holes for just the selected layer-pair.

The series of images below show the Drill Drawing layer as each layer-pair is selected, the drill table updates automatically to suit the chosen layer-pair.
Use the special menu to select the drill pair you want displayed on the drill drawing layer, as you switch drill pairs the drill table is automatically updated to suit.

Note that the **Layer pairs to preview** option is *not* used to control which layer-pair drilling is included in the Table during output generation, that is defined by the **Drill Layers** setting in the **Layer Properties dialog** in the output setup. That means that a single table can be placed on the design, the table will only include the appropriate layer-pair data during output generation for a drill drawing of that layer-pair.

![Drill Table Example](image)

Typically you only need to place one drill table - the layer pair to preview option is automatically switched to reflect which drill pair you select in the special Drill Drawing layer tab pop-up menu.

### Configuring the Display of Columns

In the **Drill Symbols dialog**, the columns are enabled/disabled by setting their state in the **Grouping** list. In the **Drill Table dialog**, the columns are initially defined by the default settings configured in the **Preferences** dialog. To add/remove columns from the Drill Table, right click in the grid region of the **Drill Table** dialog and select the required **Add Column** or **Remove Column** command. If a column is not used for grouping, but is displayed in the Drill Table, then it will display an * (asterix) whenever there are multiple possible values to display.
Printing the Drill Drawings

When you add a new Drill Drawing fabrication output, it is automatically configured to include a printout for each drill pair currently configured for the design.

<table>
<thead>
<tr>
<th>Name</th>
<th>Data Source</th>
<th>Output Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drill Drawings</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Add the Drill Drawing to the outputs in the usual way, it will include a printout for each drill pair.

Note the 3 printouts generated for this drill drawing output.

Source URL: https://www.altium.com/documentation/display/ADES/NFS_15_1((Live+Drill+Drawing))_AD