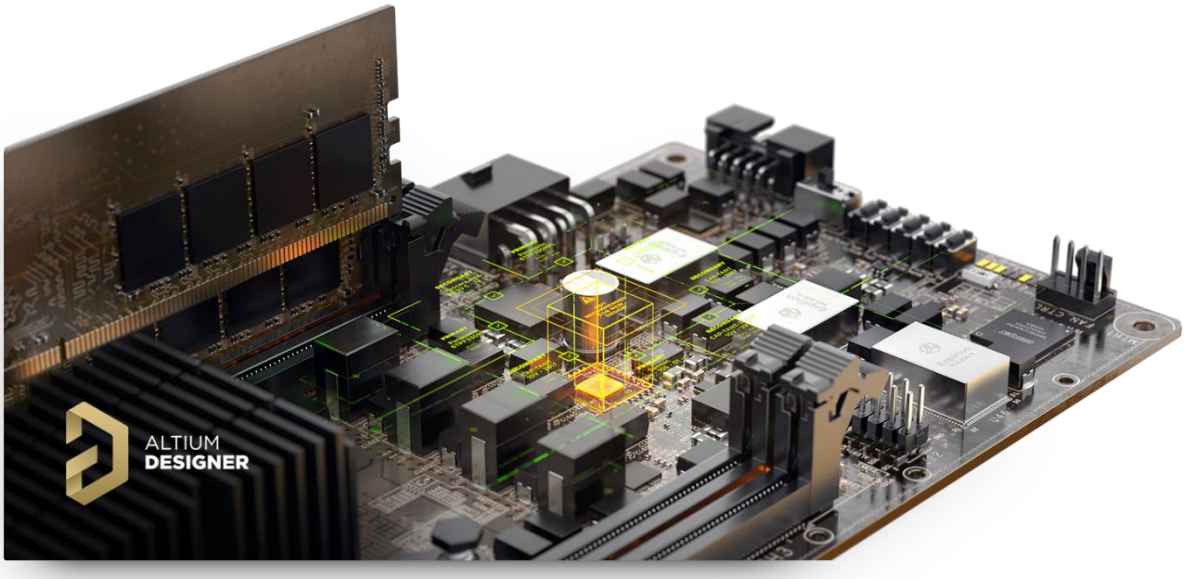


# Altium Designer

## Advanced Course

### Module: Embedded Board Array

Software, documentation and related materials:

# Table of Contents

# Embedded Board Array

## 1.1 Purpose

Even if you send out your design to fabrication as a single board, the majority of boards are panelized by the board fabrication house. They use standard sized panels in their processes so multiple quantities of the board are placed efficiently on that panel. Panels can also be created as a mixture of different boards as long as they all share the same stackup of copper layers. These panels are often used through the assembly process for installed components as well.

## 1.2 Shortcuts

Shortcuts when working with Embedded Board Array
**F1:** **Help**
**CTRL+S:** **Save Document**

## 1.3 Preparation

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

## 1.4 Placing The Board Array Into A Panel Template

3. Open the `1SL1 Xilinx Spartan-11E PQ208.PcbDoc` file. This is the design we will be panelizing.
4. Open the `14X18_PanelTemplate_Size_D.PcbDoc` file. This is the panel template that we will use to panelize our design.
5. From the `14X18_PanelTemplate_Size_D.PcbDoc`, go to the **Place** menu and select **Embedded Board Array/Panelize**.
   a) Once the command is active, the cursor will change to a cross hair with a green box.
   b) Press the **TAB** key to edit the Embedded Board Array properties in the *Properties* panel.
   c) In the *Properties* panel, click the ··· button to the right of the *PCB Document* field as shown in Figure 1 below.
   d) Select the `1SL1 Xilinx Spartan-11E PQ208` PCB document in the `Embedded Board Array` folder and click **Open**. The PCB design will now be on your cursor.

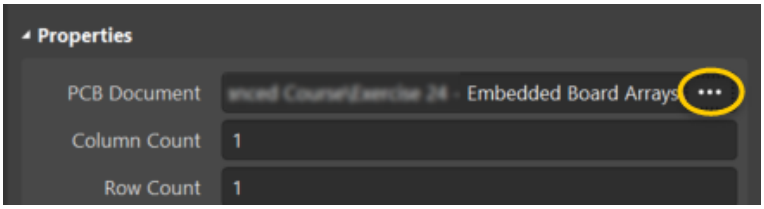


*Figure 1. Source PCB for panelizing*

6. With the PCB design still on your cursor, we'll change some of the values in the *Properties* panel. Refer to the information below, as well as Figure 2 on the following page.
   a) Set the *Column Count* to `2`
   b) Set the *Row Count* to `5`
   c) Set the *Column Margin* to `94mil`. This is the X axis gap between the boards in the panel.
   d) Set the *Row Margin* to `94mil`. This is Y axis gap between the boards in the panel.
   e) Leave the *Row Spacing* and *Column Spacing* at the default values of 2082.189mil and 7180.614mil, respectively.
   f) Deactivate the Drill Table.
   g) During placement Altium is checking the layer stacks. If the stacks from the Panel PCB and the Source PCB are not compatible Altium will give you a hint at the *Properties* Panel.

If the *Properties* panel is too narrow, the *Margin* and *Spacing* values may not be shown. If you do not see these values, enlarge the width of the *Properties* panel.


*Figure 2. Embedded Board Array dialog*

7. The mechanical layers of the reference design are still displayed and these layers will transfer to the panel unless we hide them. We will configure the display of the Mechanical Layers to hide the layers that we're not going to need in the panel.
   a) With the place command still paused, open the *View Configuration* panel from the **Panels** button. If you've exited the paused placement prematurely, simply press the **TAB** key.
   b) Under the *Mechanical Layers (M)* section, click the eye icon to hide layers `19_FABRICATION_NOTES` and `21_TITLE_BLOCK` as shown in Figure 3.

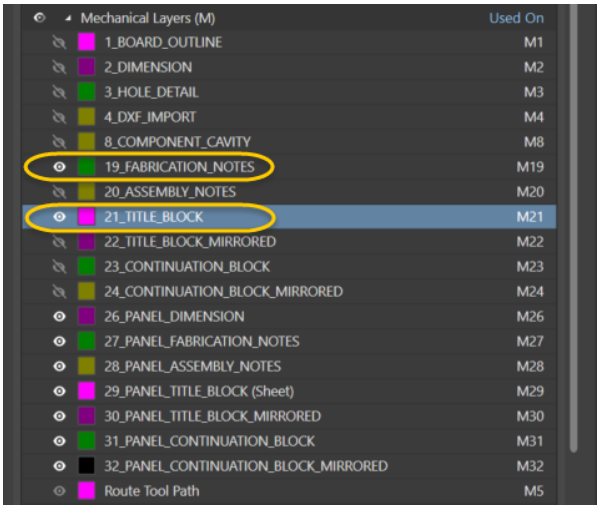


*Figure 3. Mechanical layer display for referenced PCB*

8. Click the **Pause** icon to resume placement and place the Embedded Board Array at the Location x:2000 y:2000 mil, as shown in Figure 4

To place the Embedded Board Array at the Location x:2000 y:2000 mil (in the middle of the panel template), use the cross as reference or use the Jump Location J » L command.

9. Once the Embedded Board Array has been placed, press **Escape** or **right-click** to exit the placement mode. Your display should look like Figure 4.

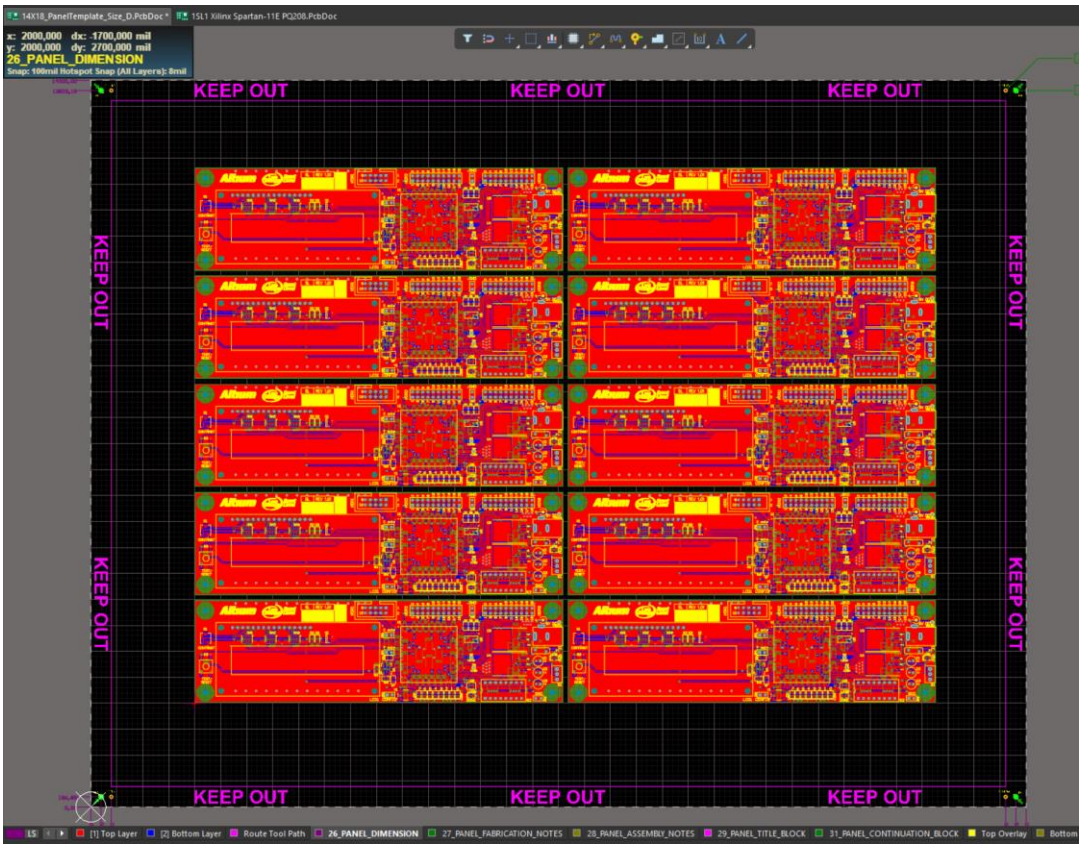


*Figure 4. Placed Embedded Board Array*

10. Switch to the 3D view of the PCB by hitting the **3** key. Note the one solid PCB with ten instances of the `1SL1 Xilinx Spartan-11E PQ208.PcbDoc` is shown in Figure 5.

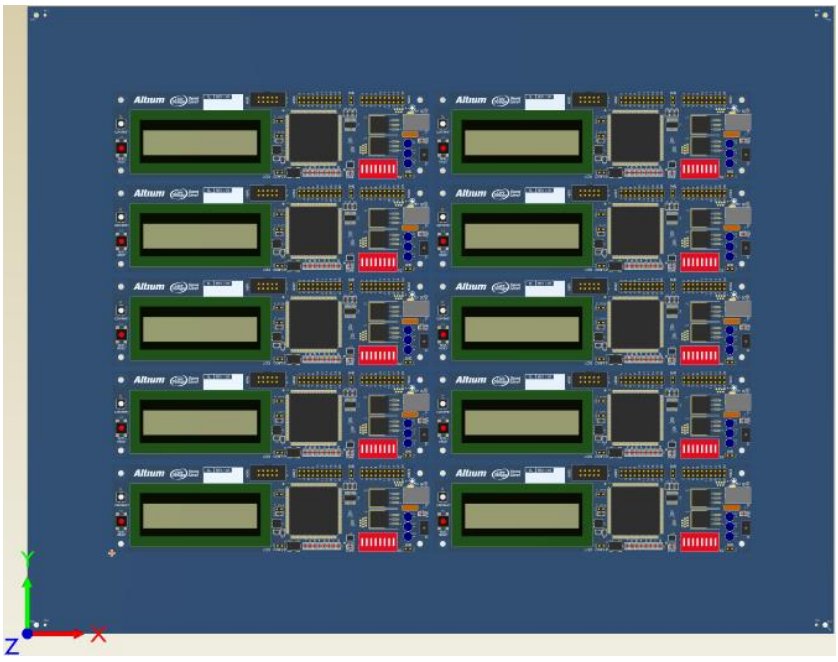

*Figure 5. 3D view of the panel*

11. Switch back to the 2D viewing mode by hitting the **2** key.

# 1.5 Placing the Route Tool Path

The route tool path is used by the fabricator to remove the individual PCBs from the panel. A mechanical layer is allocated for this information, and the route file is generated with the drill files for the final fabrication data.

## 1.5.1 Route Tool Path

12. Switch the focus to the `1SL1 Xilinx Spartan-11E PQ208.PcbDoc` file.
13. In this PCB, there's an existing Mechanical Layer already defined as the Route Tool Path.
14. Open the *View Configuration* panel.
15. In the `Mechanical Layers (M)` section, right-click on the `Route Tool Path` layer name and select **Edit Layer** as shown in Figure 6.

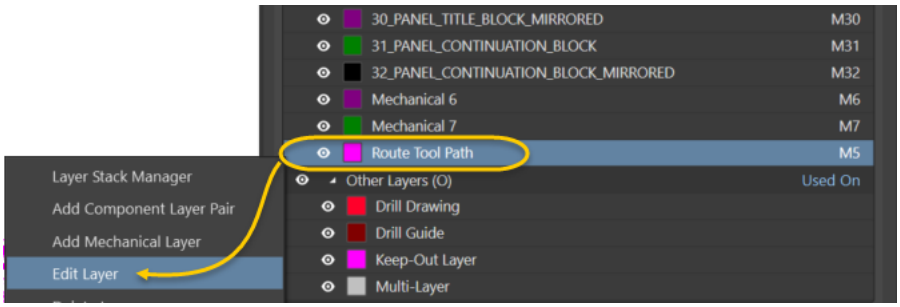


*Figure 6. Edit Route Tool Path layer*

16. The *Layer Type* for this layer is set to **Route Tool Path** as shown in Figure 7.
17. Close the *Edit Layer Route Tool Path* dialog without making any modifications.

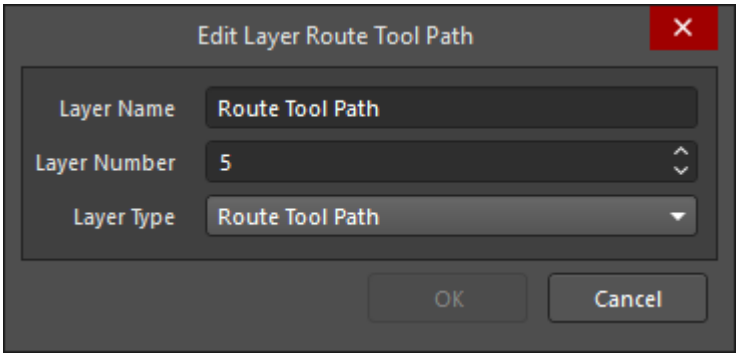


*Figure 7. Mechanical Layer with a Route Tool Path type information*

## 1.5.2 Define the Route Tool Path

18. Switch the focus **back** to the `14X18_PanelTemplate_Size_D.PcbDoc` file.
19. From the **Design** menu, select **Board Shape » Create Primitives From Board Shape**.
20. Set the options in the *Line/Arc Primitives From Board Shape* dialog as follows below. Also use Figure 8 below as a reference:
    a) Set the *Width* to `93mil`.
    b) Ensure the *Layer* field is set to the **Route Tool Path** layer.
    c) Enable only the option for the **Route Tool Outline**.

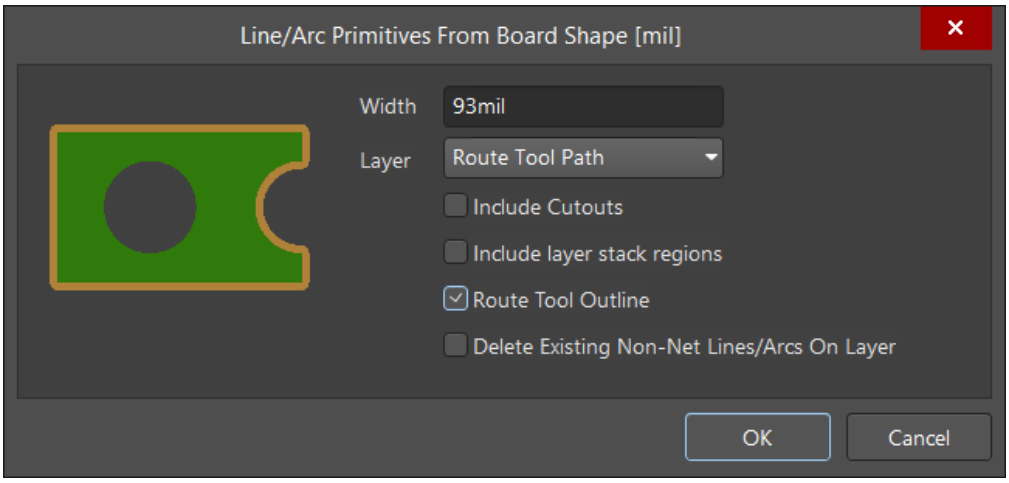


*Figure 8. Line/Arc Primitives From Board Shape dialog*

21. Click **OK** to close the dialog.
22. Switch to the 3D view of the PCB by hitting the 3 key and note the route tool path cuts through the panel, isolating the ten individual instances of the panel.

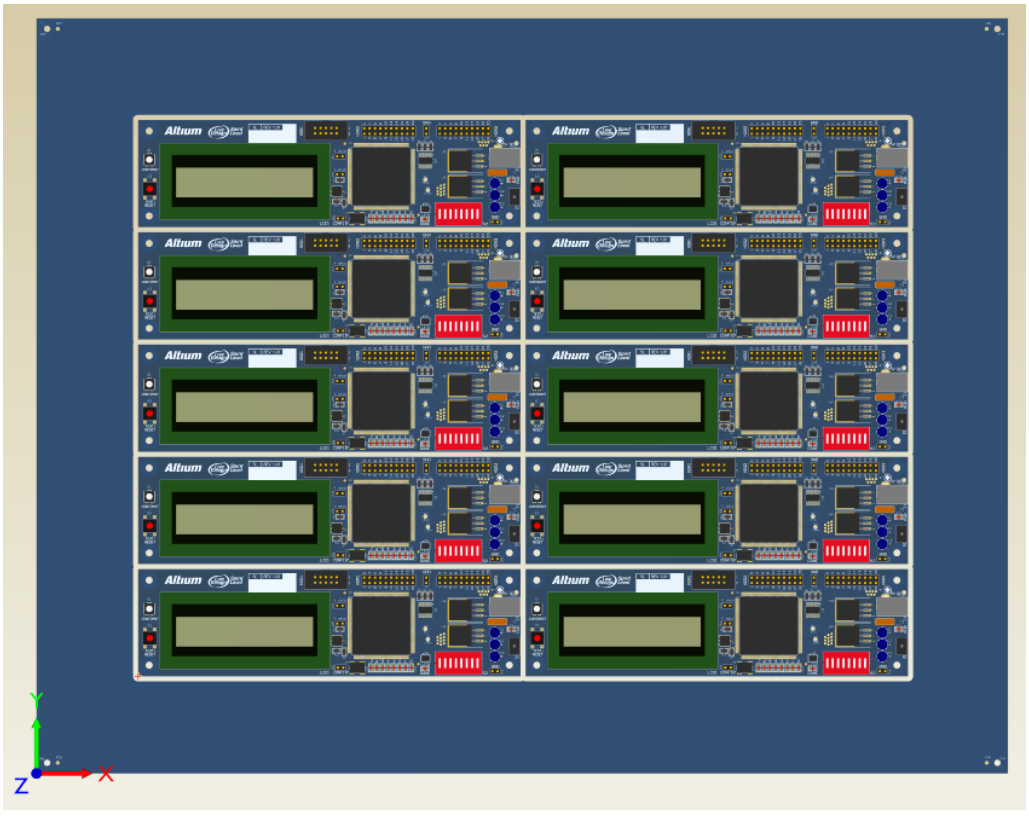


*Figure 9. 3D view showing Route Path cutouts*

23. Switch back to the 2D view by hitting the **2** key.

# 1.6 Creating Breakout Tabs (Mouse Bites)

Breakout tabs are used to hold the individual instances of the PCB in the panel while facilitating easy removal of the finished PCBs. In the next steps, we will create gaps in our Route Path so that we can add our breakout footprints.

In the `Breakouts.PcbLib`, the breakout tab footprints were specifically designed to be placed on a route tool path of 93mils in width, for a standard 94mil tool. With 31mil hole centers placed 0mil from routing edge, the origin is positioned so it can be placed at the center line of the route tool path. This is something you should discuss with your manufacturer.

24. From the **Edit** menu, select **Slice Tracks**. You'll notice a crosshair on your cursor.
25. We will start with some horizontal cuts:
    a) Confirm that your active layer is the `Route Tool Path` layer.
    b) Navigate to the first instance of the PCB in the top left corner, just below the pads of `R1` as shown in Figure 10.
    c) Left-click outside the board edge to start the cut.
    d) Start moving your cursor to the right as shown in Figure 10. You'll notice the solid and dotted lines appear, ready to start the slicing of the tracks.

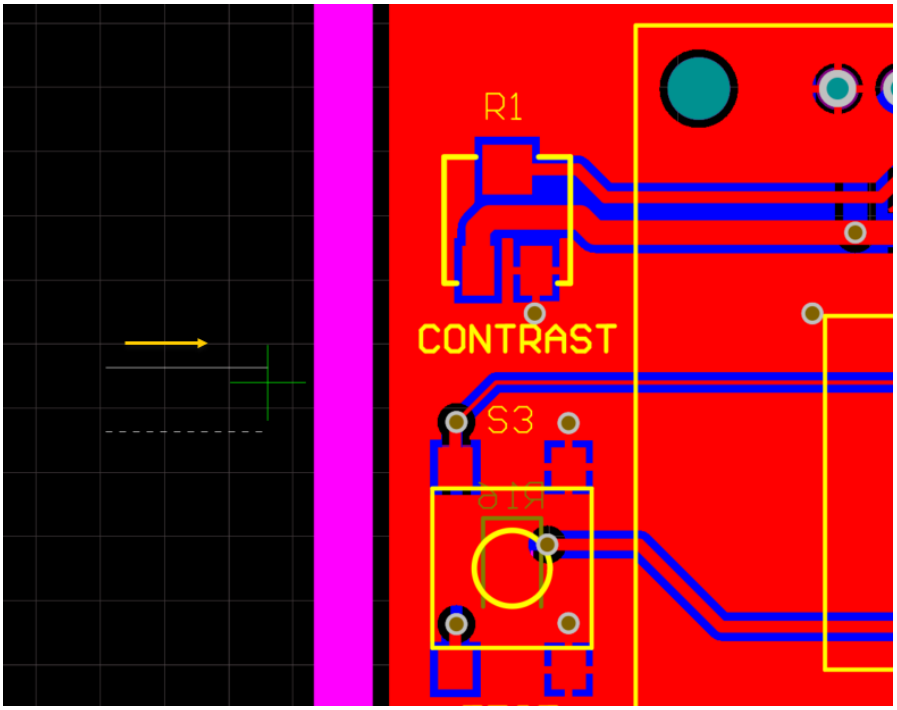


*Figure 10. Slick Tracks command near the board edge*

    e) Press the **TAB** key to bring up the *Slicer Properties* dialog as shown in Figure 11.
    f) Set the *Blade Width* to `240mil`.
    g) Enable the **Cut Current Layer Only** option.
    h) Disable the **Snap Blade Width To Grid** option.
    i) Click **OK** to return to the PCB.

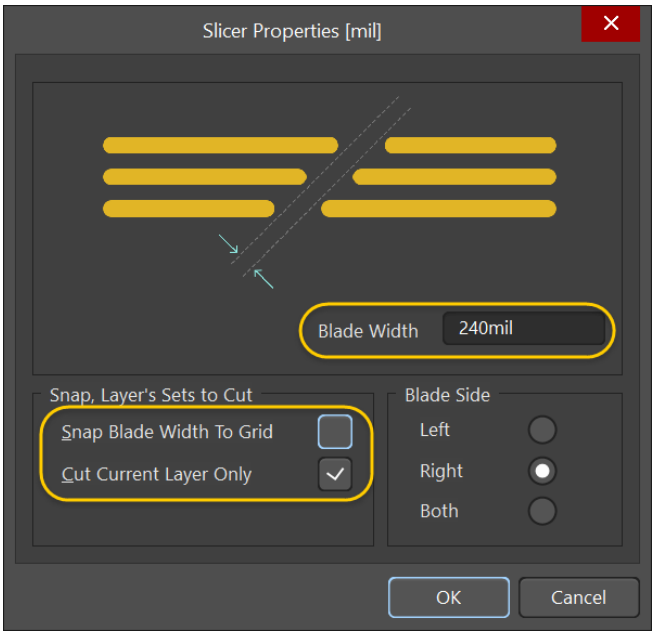


*Figure 11. Slicer Properties*

26. Drag the slice all the way horizontally, across all the rows of the PCB instances of the panel, similar to what is shown in Figure 12.

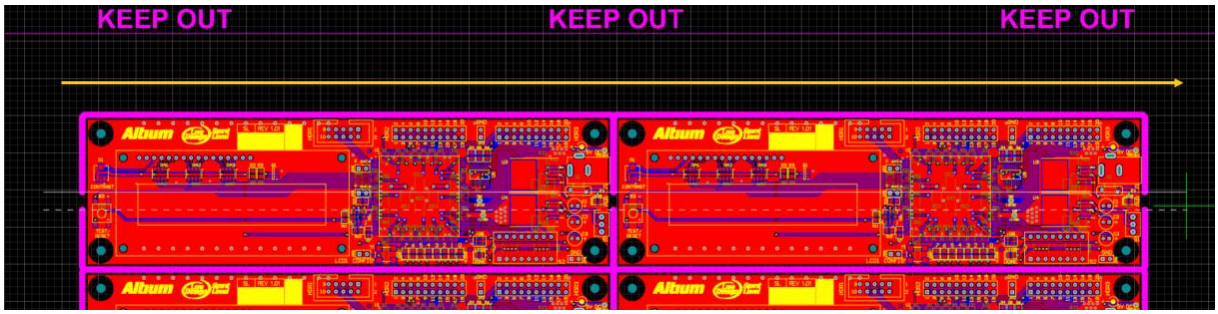


*Figure 12. Horizontal slice across entire panel*

27. Left-click to terminate the first cut. You'll notice the `Route Tool Path` layer will have a small opening on each the left, middle and right of the panel. This is where we will be inserting our breakout tabs.
28. With the **Slice** command still active on your cursor, repeat this process 4 more times for the remaining rows of panelized boards. The `Route Tool` Path layer should now look similar to Figure 13 on the following page.

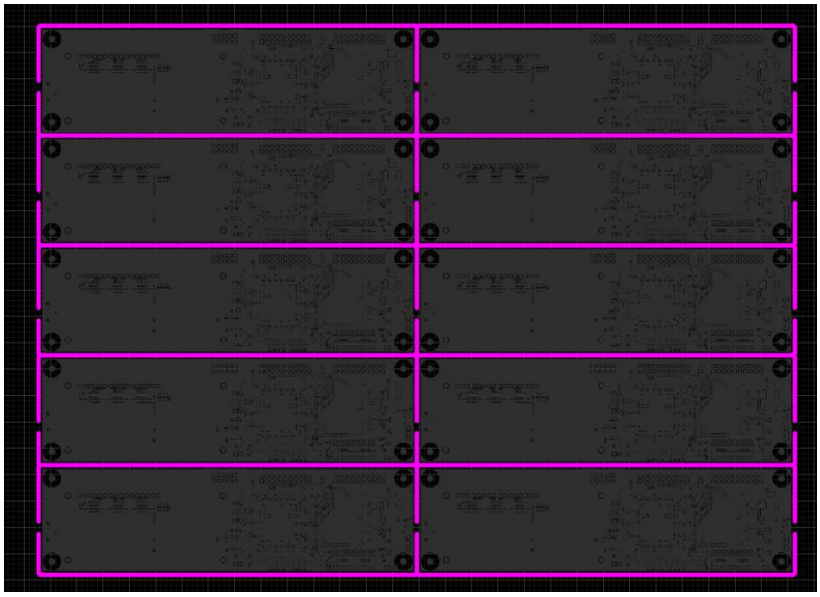

*Figure 13. Horizontal cuts for breakout tabs*

29. Now we will create the vertical cuts for our breakout tabs.
    a) While still in the **Slice Tracks** command, position the crosshair cursor near the top-left corner of the PCB as shown in Figure 14. If you've exited the command, it can be found from the **Edit** menu.

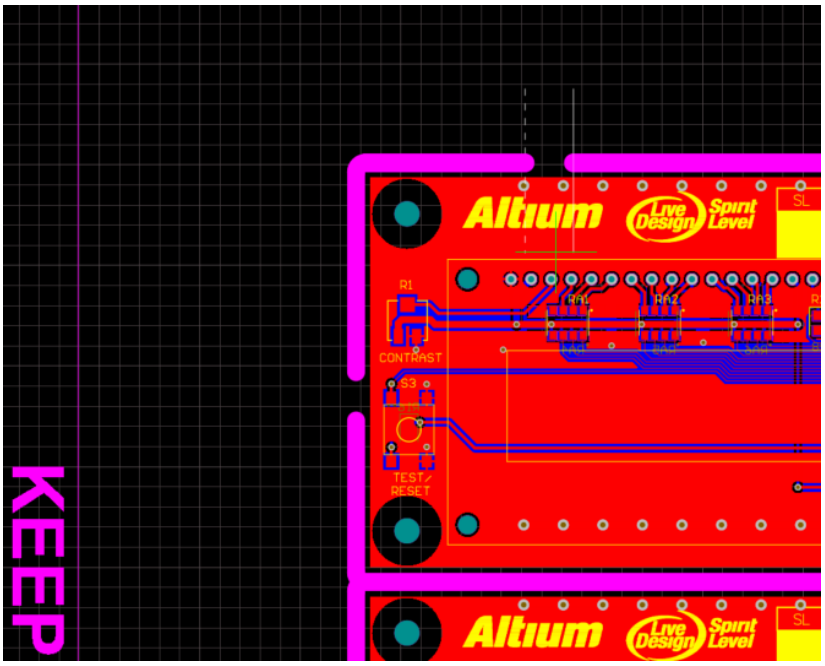


*Figure 14. Position of the Slice Tracks curser for first vertical cut*

    b) Left-click and slice the tracks vertically across the entire panel as shown in Figure 15.

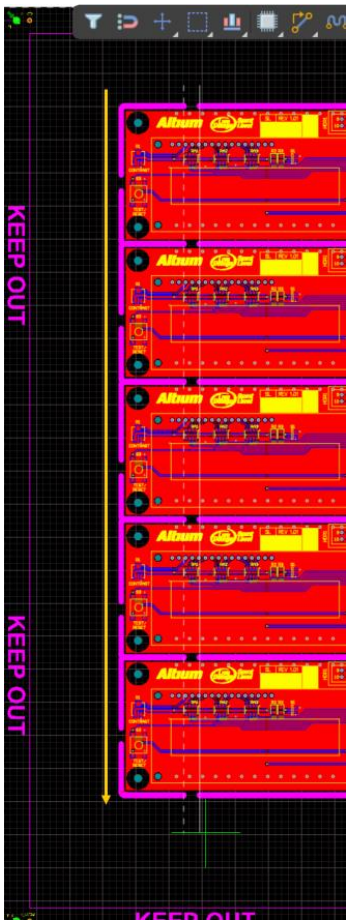


*Figure 15. Vertical cut across entire panel*

c) Left-click to complete the slice. You'll notice that the crosshair will still be on your cursor.

30. For the second vertical cut, move the cursor towards the top-right side of the first PCB instance as shown in Figure 16. We will start this cut near `Pin 18` of component `HDR2`.

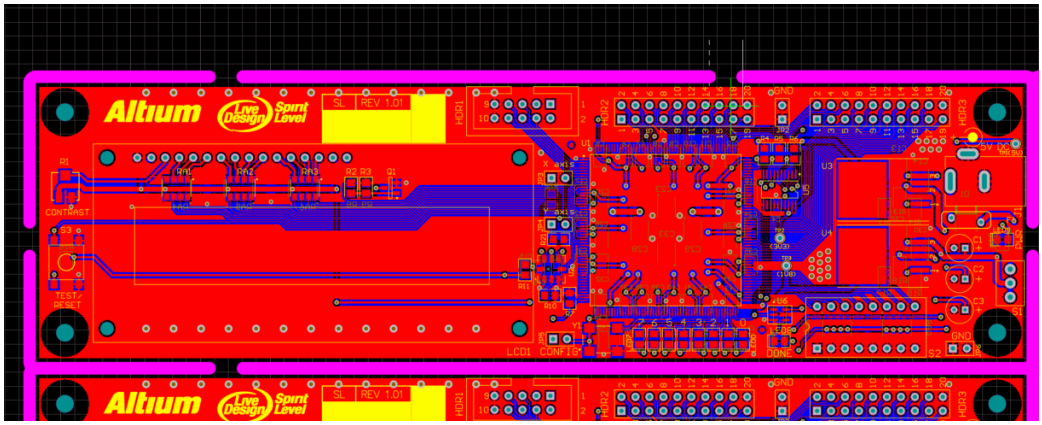

*Figure 16. Position of the Slice Tracks curser for second vertical cut*

a) Left-click and slice the tracks vertically down across the entire panel, left-clicking again to end the current cut.

b) Repeat the two vertical cuts for the second column of boards in the panel. The result should resemble Figure 17.

c) Right-click to exit the command once completed.

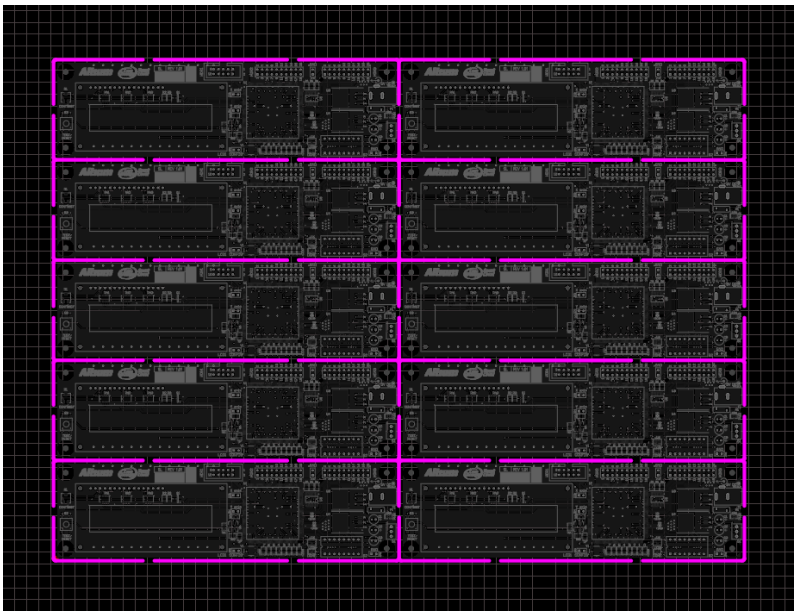

*Figure 17. The Embedded Board Array with properly positioned slices in the route tool path*

31. View the panel in 3D and observe the new route tool path.
32. Switch back to the 2D view.
33. Open the *Preferences* **.**
    a) From the *PCB Editor* section, go to the *General* page.
    b) Under the *Object Snap Options* section**,** enable **Snap To Center** as shown in Figure 18.
    c) Disable **Smart Component Snap.** This will allow ease of placement of the breakout tabs.
    d) Click **OK** to save and close the *Preferences*.

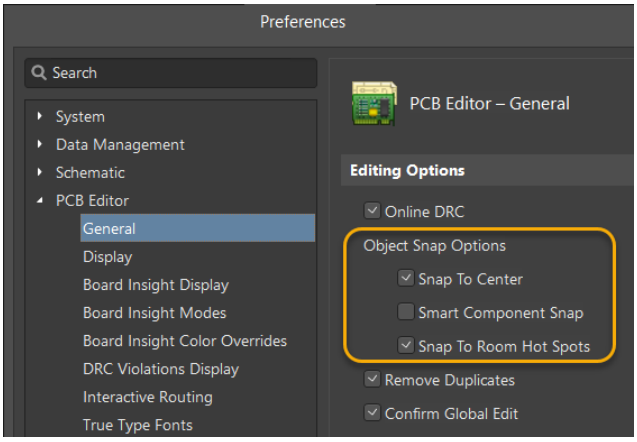


*Figure 18. PCB Preferences*

34. From the layer tab bar, ensure that the `Route Tool Path` layer is still the active layer.
35. With nothing selected in the PCB, hit the **G** key, and change the grid to `50 mil`**.**
36. Open the *Properties* panel, tab general.
37. Use Figure 19 below to set the following settings in the *Properties* panel:
    a) Disable the **Grid** button in the *Snap Options***.** If the button is grey, it means its disabled.
    b) In the Snapping section, ensure its set to **Current Layer.**

c) In the *Objects for Snapping*, enable **Track/Arcs Vertices**, as well as **Track/Arcs Lines**. Please note that you will likely have other options enabled in this section and that is okay.

d) Set the *Snap Distance* to `25mil`.

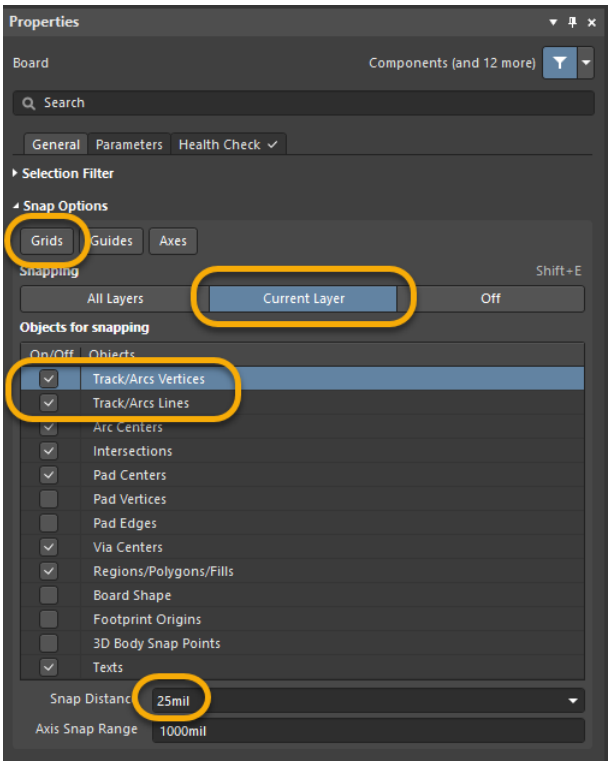


*Figure 19. Snap Settings for placement of Mouse Bites*

38. Open the *Components* panel from the **Panels** button.
39. Using the *triple line* icon to ensure that **Models** is enabled as shown in Figure 20. This will allow us to select the `Breakouts.PcbLib`.

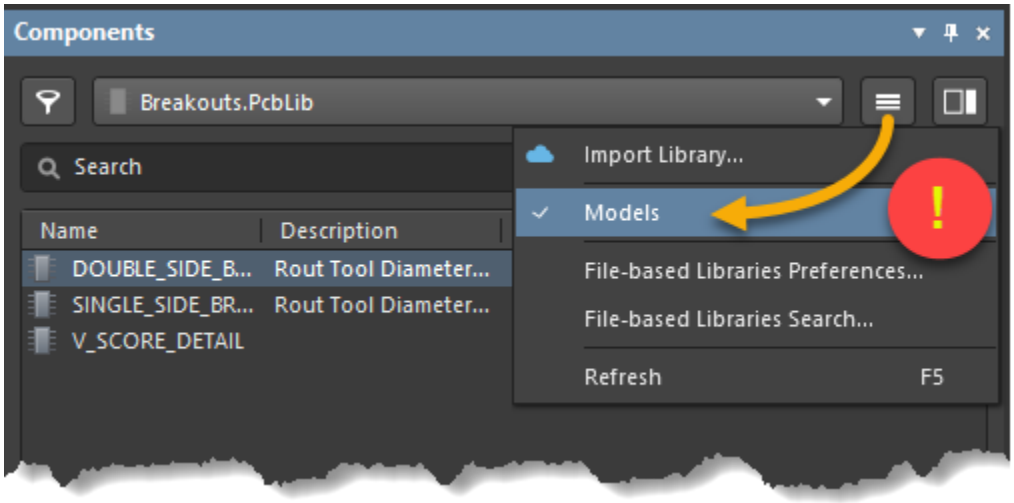


*Figure 20. Enable the visibility of Model libraries*

40. From the library drop-down menu, select `Breakouts.PcbLib` as shown in Figure 21.

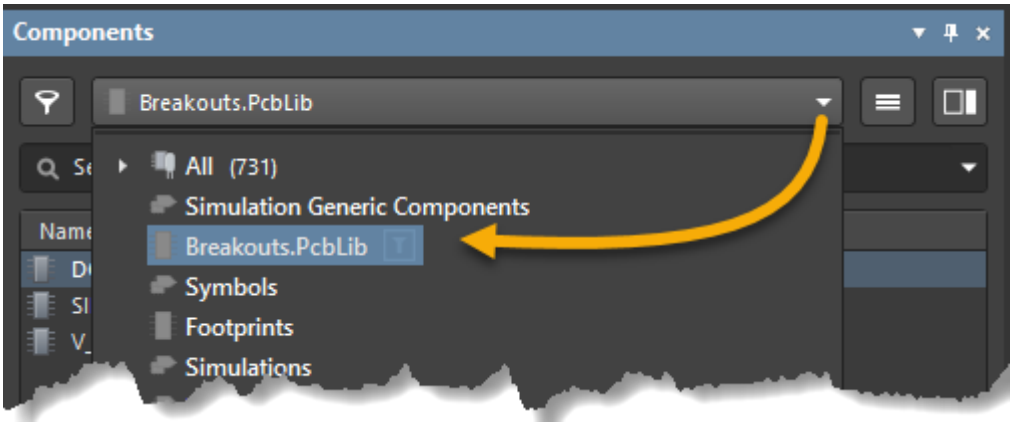


*Figure 21. Breakout display in the Components panel*

41. Right-click on the `SINGLE_SIDE_BREAKOUT` footprint and select **Place** as shown in Figure 22.

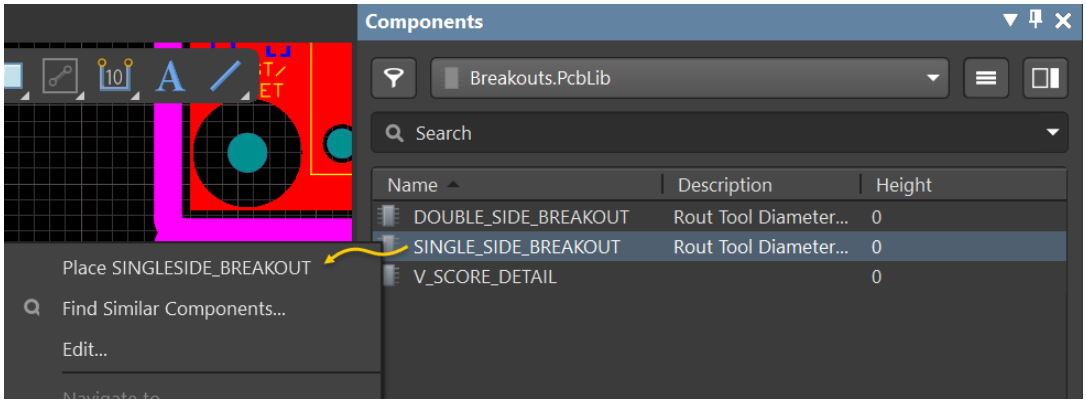


*Figure 22. Placing the breakout footprint*

42. Before placing the footprint, hit the **TAB** key to access the *Properties* panel.
43. Click on the eye icon next to the *Designator* field to hide it as shown in Figure 23. You'll notice that the `Designator1` value will no longer be visible in the PCB.

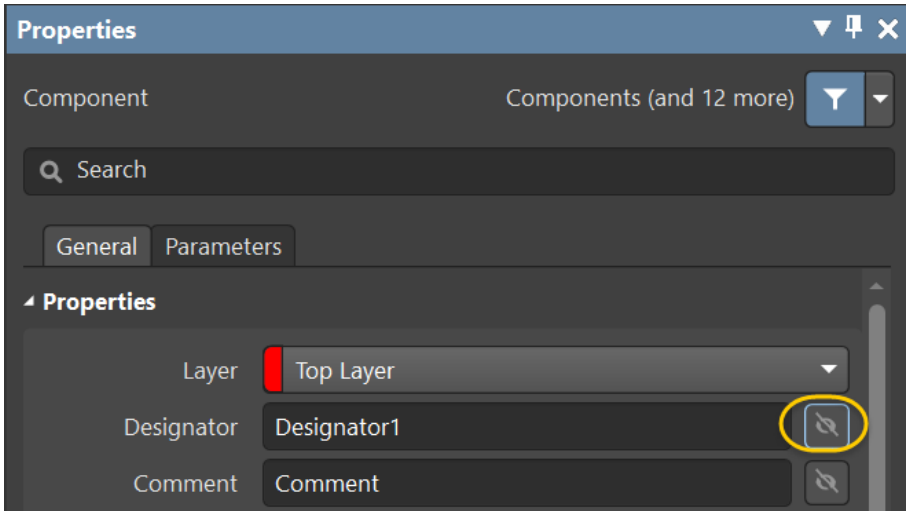


*Figure 23. Hide footprint designator*

44. Click the **Pause** icon or the **ESC** key to resume the placement.

45. With the `SINGLE_SIDE_BREAKOUT` footprint on your cursor, place the origin of the footprint onto the edge of the route path gap as shown in Figure 24. Use the **Spacebar** to rotate the footprint as necessary.

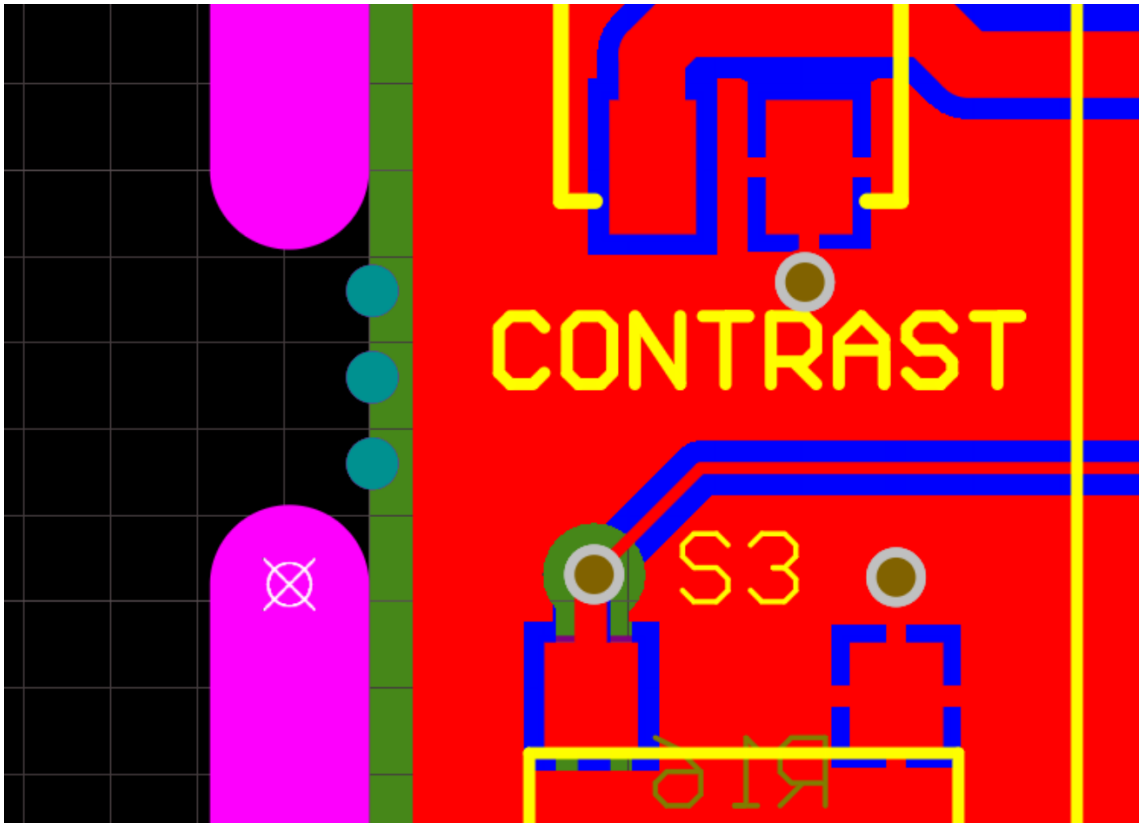


*Figure 24. Placing the breakout tab on the route path edge*

46. Once you've placed the first breakout footprint, rotate it 180 degrees and place another on the right-edge of the route path gap.
47. Right-click to exit the command once finished.
48. From the `Breakouts.PcbLib`, right-click on the `DOUBLE_SIDE_BREAKOUT` footprint and select **Place**. You'll notice that this footprint is double-sided, allowing us to place it in between the board instances of our panel.
49. With the footprint on your cursor, place it on one of the voids on the inner route tool path as shown in Figure 25.
50. Right-click to exit the command once placed.

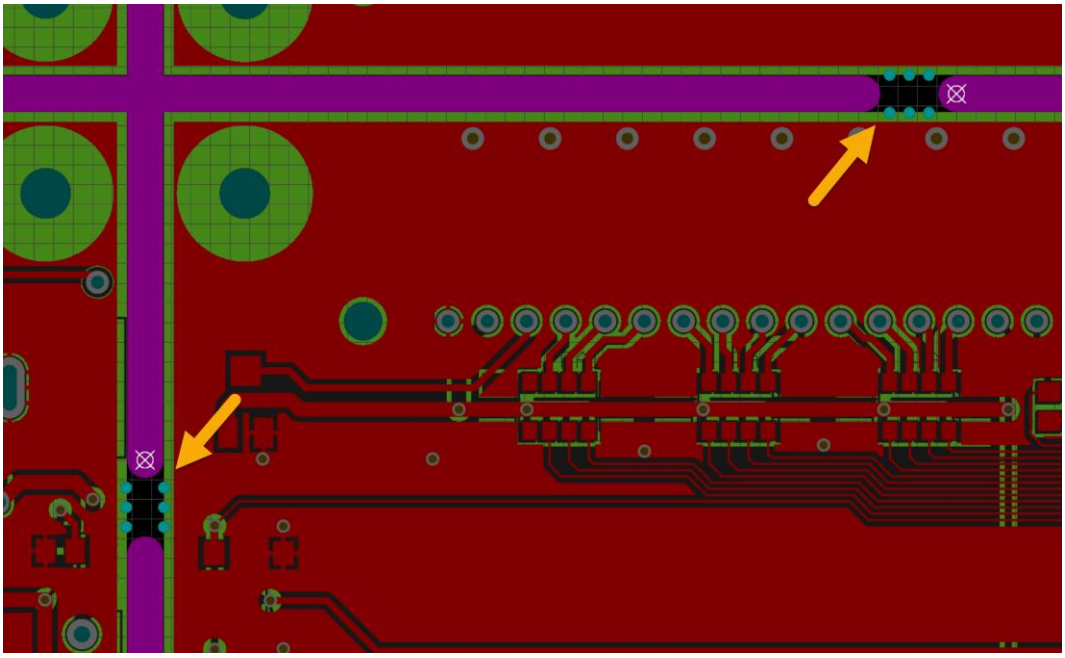

*Figure 25. Placement of breakout footprints, snapping reference to track ends*

51. View the panel in 3D and observe the new route tool path with the breakouts.
52. Switch back to 2D using the **2** key.

For reference, the `14X18_Panel_RouteTool_D.PcbDoc` is an example for a completed cut route path and the added breakout footprints.

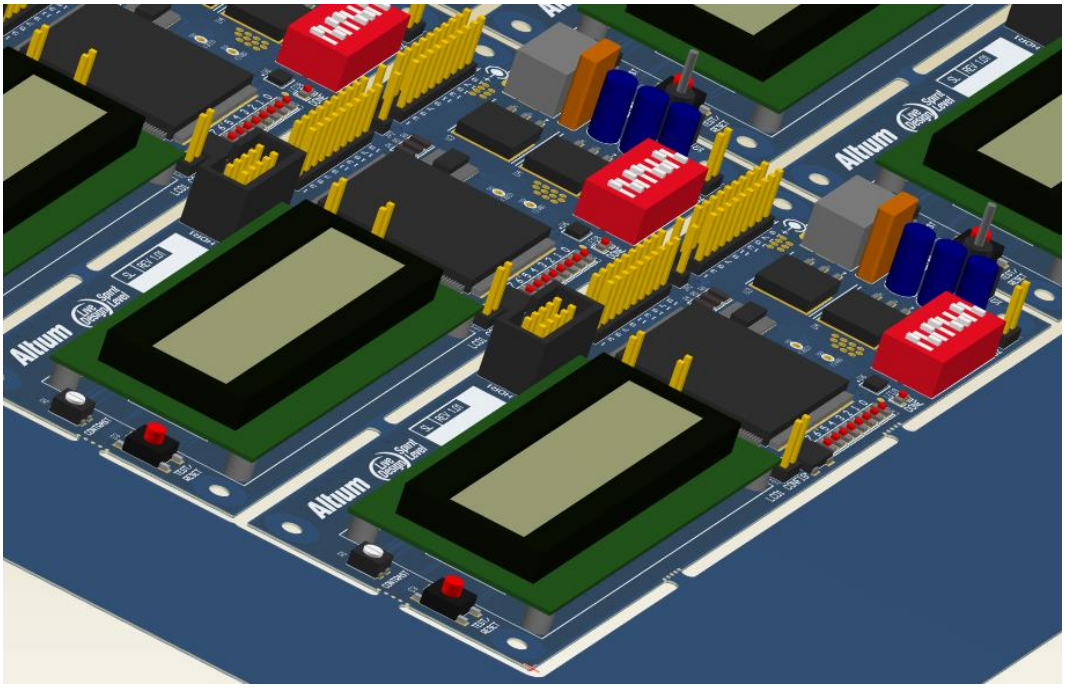

*Figure 26. Example  of final panel with route guides and mouse bites*

# 1.7 Fabrication Files

53. Open the Output Job File `Embedded Board Array.OutJob`.
54. Open the configuration for Gerber to see the Gerber settings and the Layer Table from the Panel PCB and the Source PCB, to check that the layer are compatible.

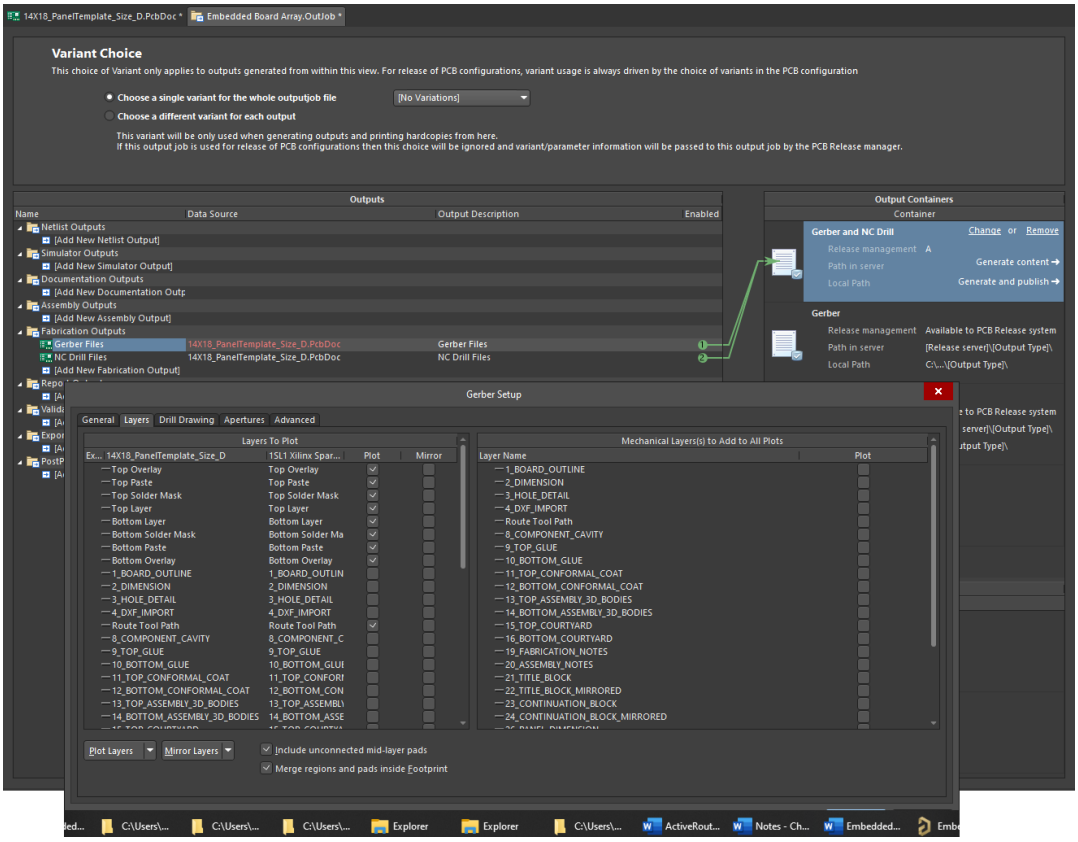


*Figure 27. Layer Table - Gerber generation*

55. Feel free to generate Gerber and NC Drill by  pressing  Nenerate Content from the Gerber and NC drill output Container.

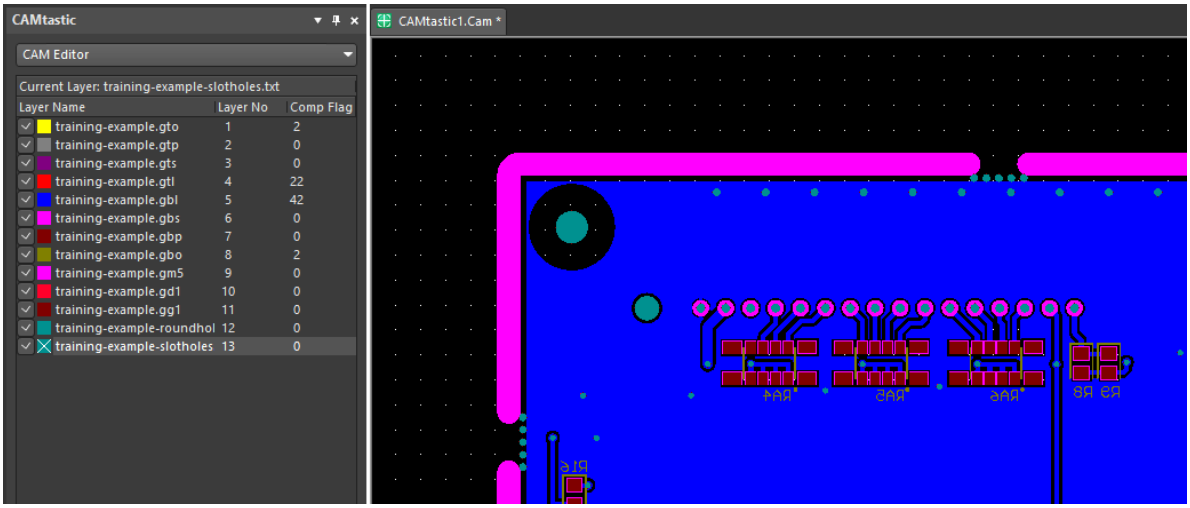


*Figure 28. Gerber and NC Drill from Panel*

# 1.8 Optional: V-Score Tool Path

V-Scoring is another method to provide a means of breakout of the individual PCB assemblies from the panel. Effectively, the fabrication house will route a score line along the edge of each instance of the PCB in the panel, to allow the individual assemblies to be “snapped” out of the panel.

V-Scoring allows a smoother final board assembly edge. The notes section of the fabrication drawing should contain detail of the V-Score geometry. It is always necessary to consult the fabrication and assembly houses for recommendations of the detail in the V-Score geometry.

56. Open the `14X18_Panel_VScore_D.PcbDoc` document.
57. Notice under the *Notes* section, just to the upper right of the board, VSCORE details have been added as shown in Figure 29.
58. In the `Breakouts.PcbLib`, the VSCORE detail has been added in the `V_SCORE_DETAIL` footprint to the `27_PANEL_FABRICATION_NOTES` mechanical layer in the library. This is a way of creating the detail of the V-Score geometry to be used in subsequent designs.
59. In the `14X18_Panel_VScore_D.PcbDoc`, observe how the lines representing the V-Score locations were placed using **Place » Line** on the `27_PANEL_FABRICATION_NOTES` mechanical layer. These are referenced in the Notes unless otherwise specified.

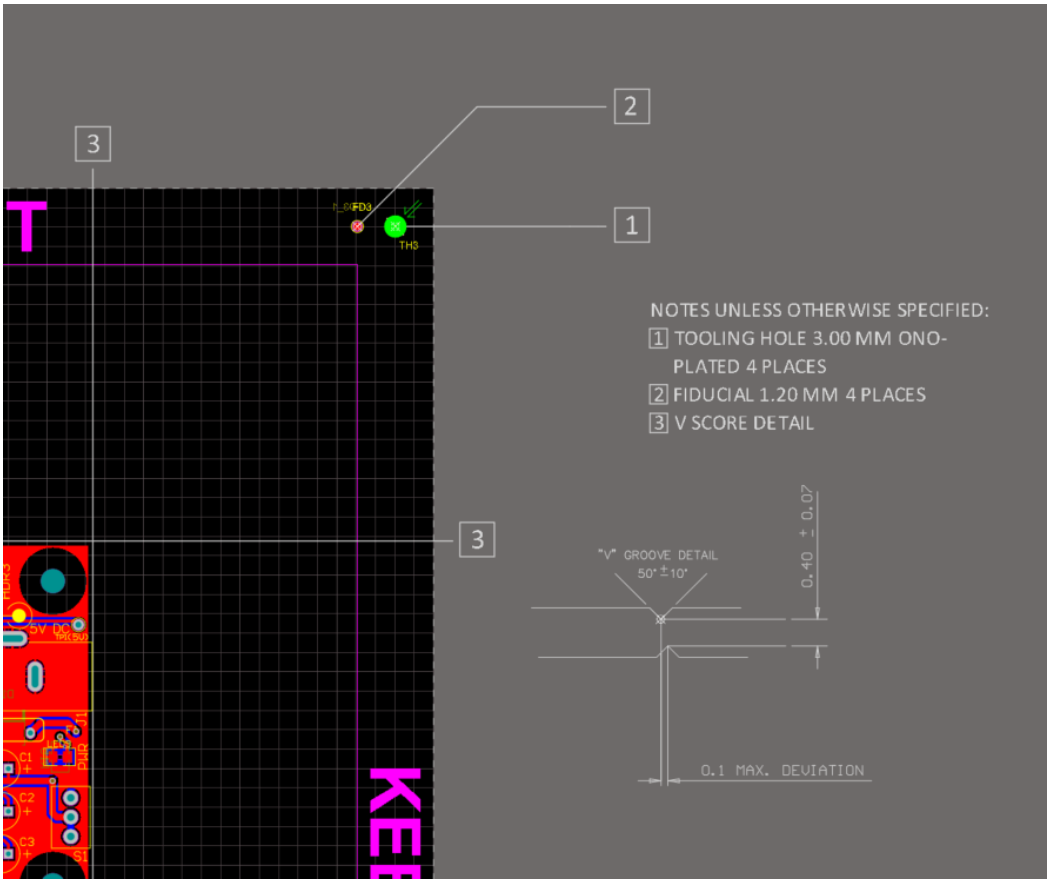


*Figure 29. VSCORE line details and callout*

With Altium Designer it is also possible to define a Mechanical Layer with the Layer Type “V Cut”.

60. **Close the project and any open documents.**

**Congratulations on completing module**

Embedded Board Array

**from the**

**Altium Designer Advanced Course**

**Thank you for choosing Altium Designer**