# Altium Designer

## Advanced Training with Altium 365

### Embedded Board Array

**Altium**®
TRAINING

**Altium**®
TRAINING

Software, documentation and related materials:

1

# Table of Contents

# Embedded Board Array

## 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.
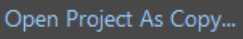
## 2  Shortcuts

Shortcuts used when working with Embedded Board Array

| | |
|---|---|
| F1 | Help |
| Ctrl+S | Save Document |

# 3 Preparation

1. Close all existing projects and documents.

2. Next, create a copy of the Training Project: `Embedded Board Array.`

3. Select **File » Open Project...** to open the *Open Project* dialog.

4. Enable the folder view button ▢ .

5. Navigate to the predefined Training Project `Embedded Board Array`
   `(Top\Projects\Altium Designer Advanced Training Course\...).`

6. Select **Open Project as Copy...** | Open Project As Copy... | .

7. In the new dialog *Create Project Copy:*

   a) Add your name to the project name: `Embedded Board Array – [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  Placing the Board Array into a Panel Template

9.  Open the `1SL1 Xilinx Spartan-11E PQ208.PcbDoc` file. You will panelize this design.

10. Open the `14X18_PanelTemplate_Size_D.PcbDoc` file. This is the panel template that you will use to panelize the design.

11. 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)  Navigate to the Project folder and select the `1SL1 Xilinx Spartan-11E PQ208` PCB document in the `Embedded Board Array` folder.

    e)  Select **Open**. The PCB design will now be on your cursor.
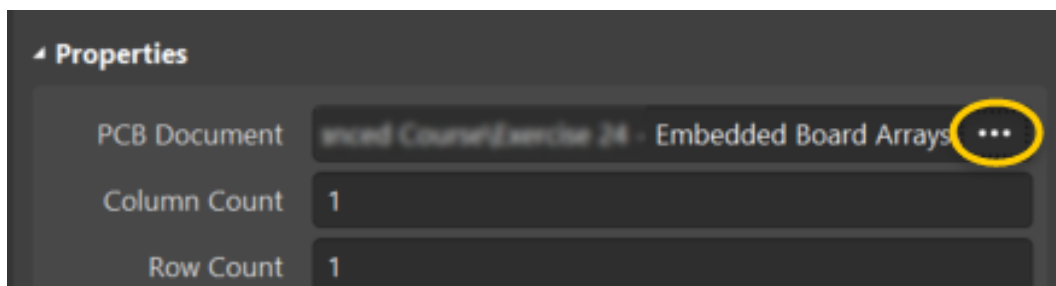


*Figure 1. Source PCB for panelizing*

12. With the PCB design still on your cursor, 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 in the *Properties* Panel.

> Hint: If the *Properties* panel is too narrow, the *Margin* and *Spacing* values may not be shown. If you don't see these values, enlarge the width of the *Properties* panel.
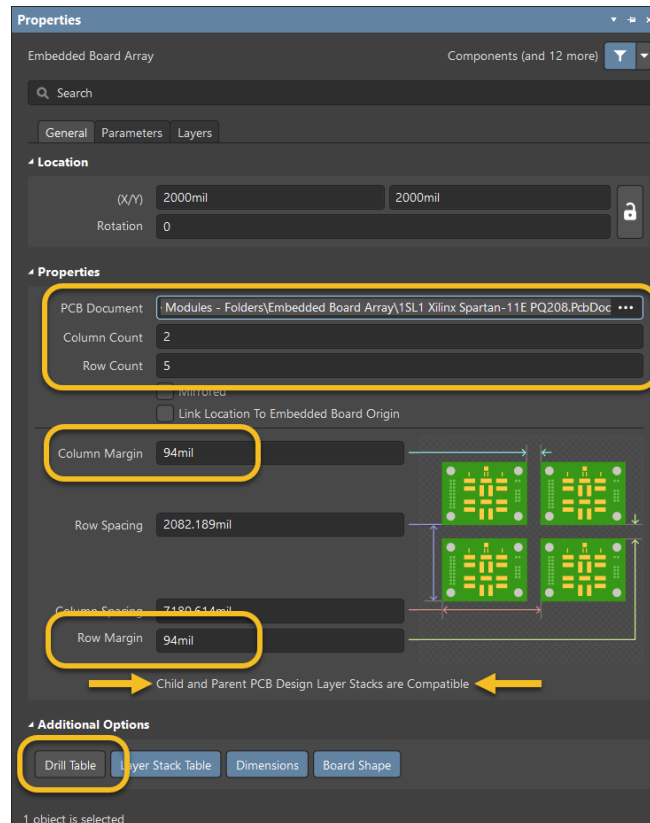
*Figure 2. Embedded Board Array dialog*

13. The mechanical layers of the reference design are still displayed, and these layers will transfer to the panel unless you hide them. Let's configure the display of the Mechanical Layers to hide the layers that are redundant in the panel:

a) With the place command still paused, select the Layers tab from the *Properties* panel. If you've exited the paused placement prematurely, press the **TAB** key.

b) Scroll down to the group of *Mechanical Layers (Type Mechanical)*, select the eye icon 👁 to hide layers `19_FABRICATION_NOTES` and `21_TITLE_BLOCK` as shown in Figure 3.
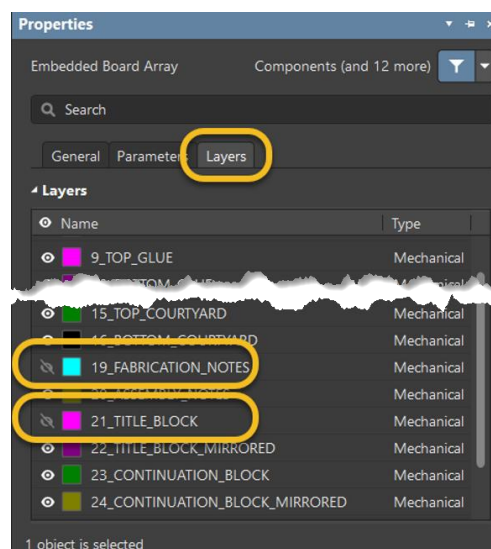


*Figure 3. Mechanical layer display for referenced PCB*

14. Select 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.

> Hint: 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.

15. 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.
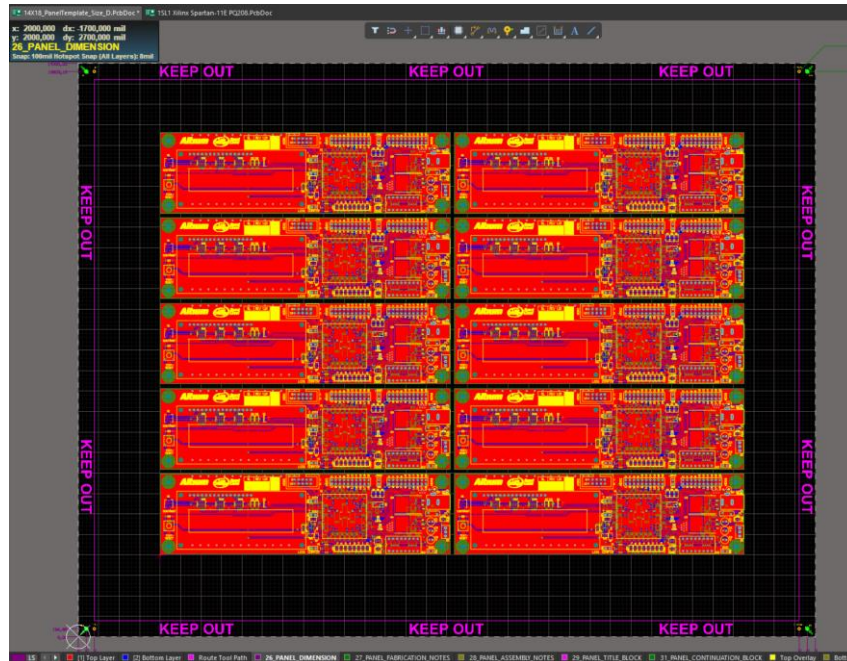


*Figure 4. Placed Embedded Board Array*

16. 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.
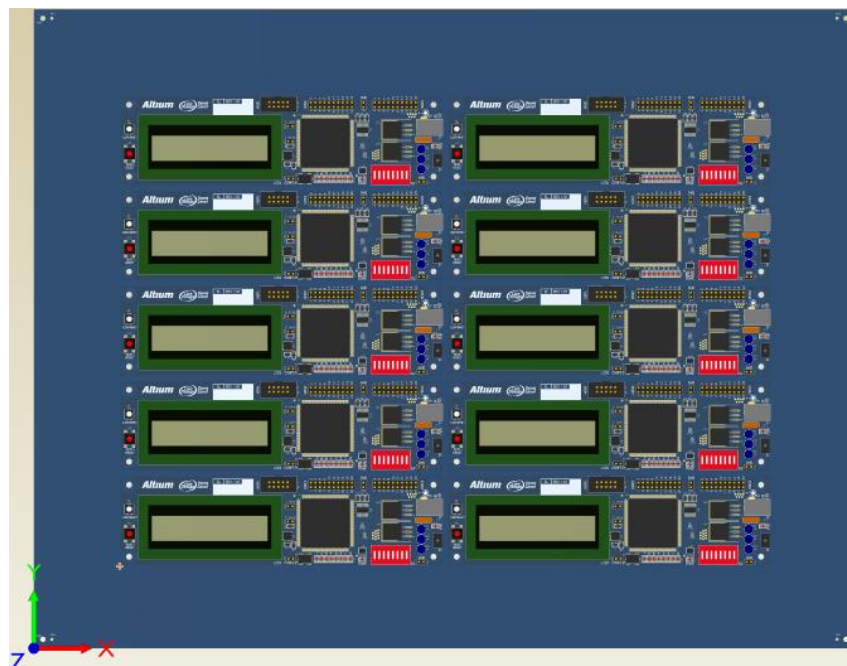


*Figure 5. 3D view of the panel*

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

7

# 5 Placing the Route Tool Path

> Hint: 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.

## 5.1 Route Tool Path

18. Switch the focus to the `1SL1 Xilinx Spartan-11E PQ208.PcbDoc` file.

19. In this PCB, there's an existing Mechanical Layer already defined as the Route Tool Path.

20. Open the *View Configuration* panel. You can click on the bar in the lower left next to the left-right layer selection buttons, or select the *View Configuration* from the *Panels* button.

21. In the `Mechanical Layers (M)` section, right-click on the `Route Tool Path` layer name and select **Edit Layer** as shown in Figure 6.
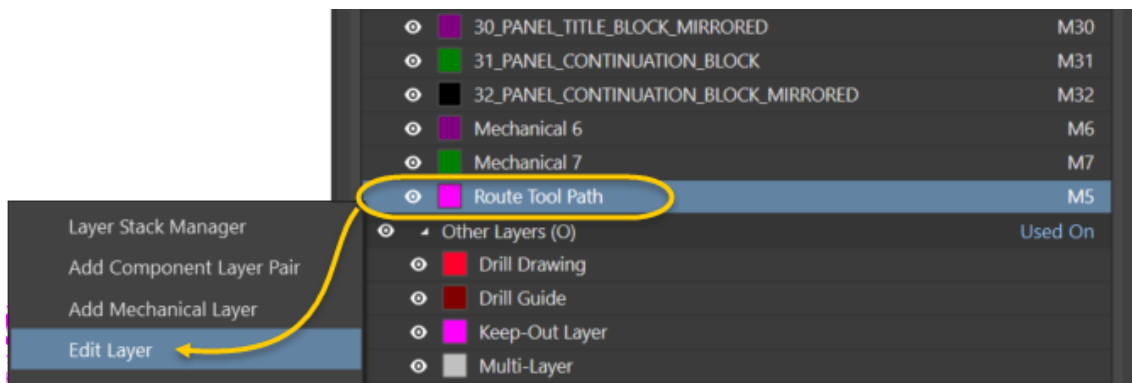


*Figure 6. Edit Route Tool Path layer*

22. The *Layer Type* for this layer is set to **Route Tool Path**, as shown in Figure 7.

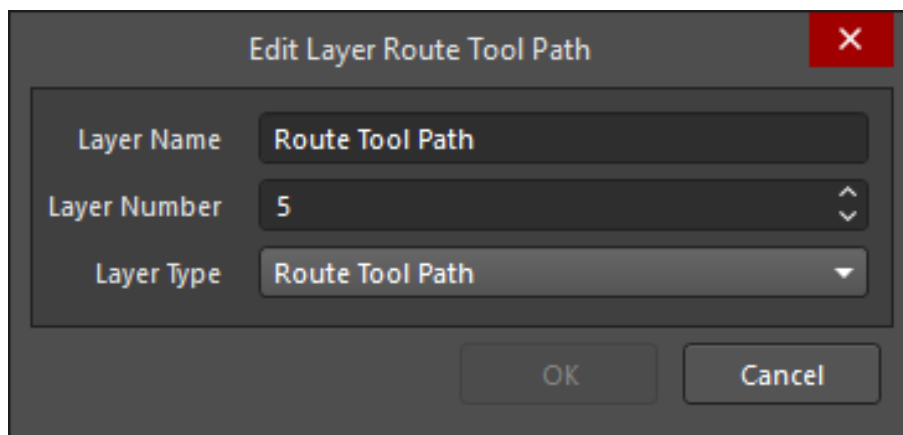23. Close the *Edit Layer Route Tool Path* dialog without making any modifications.



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

## 5.2  Define the Route Tool Path

24. Switch the focus **back** to the `14X18_PanelTemplate_Size_D.PcbDoc` file.

25. From the **Design menu**, select **Board Shape » Create Primitives From Board Shape**.

26. Set the options in the *Line/Arc Primitives From Board Shape* dialog as follows below. Also use Figure 8 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**.
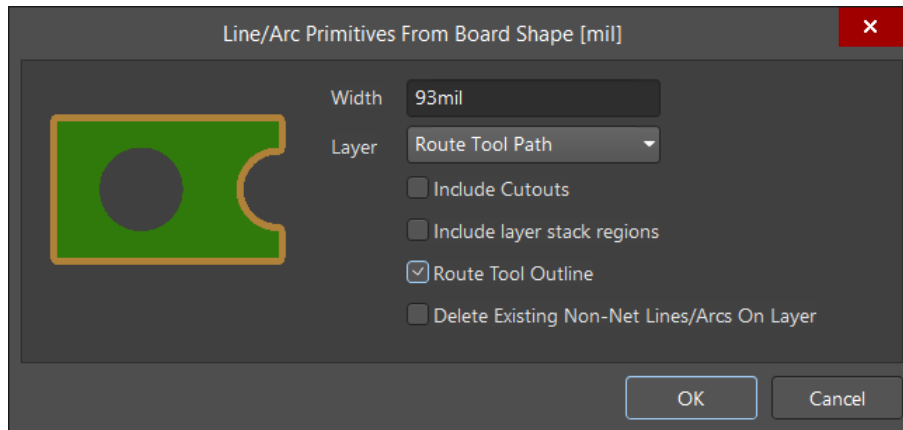


*Figure 8. Line/Arc Primitives From Board Shape dialog*

27. Select **OK** to close the dialog.

28. 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.
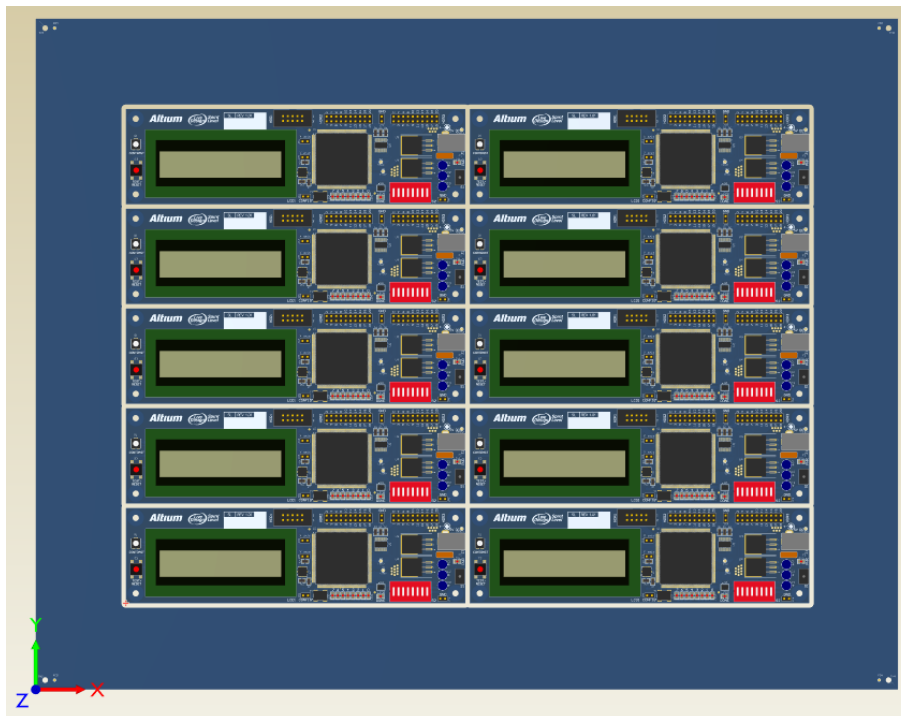


*Figure 9. 3D view showing Route Path cutouts*

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

9

# 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, you will create gaps in the Route Path to add breakout footprints.

> Note: In the training workspace for the Advanced Training with Altium 365, 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.

30. From the **Edit** menu, select **Slice Tracks**. You will notice a crosshair on your cursor.

31. Please start with 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 will notice the solid and dotted lines appear, ready to start the slicing of the tracks.
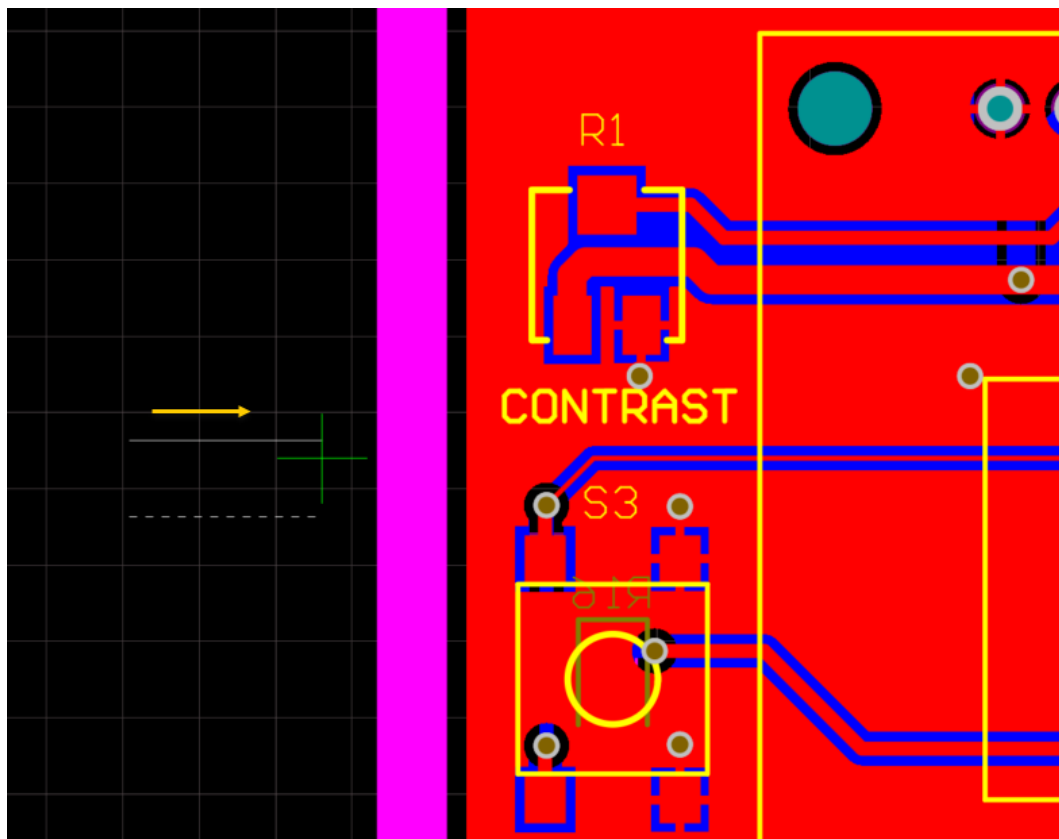


*Figure 10. Slick Tracks command near the board edge*

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.
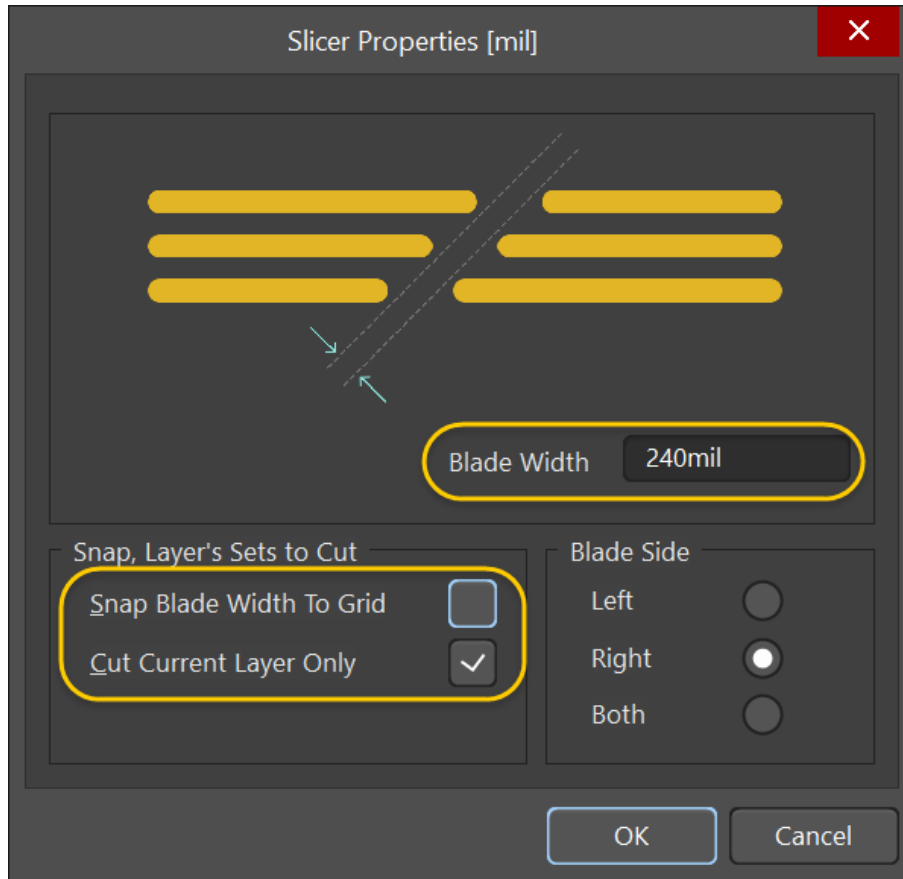
i) Select **OK** to return to the PCB.



*Figure 11. Slicer Properties*

32. 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.
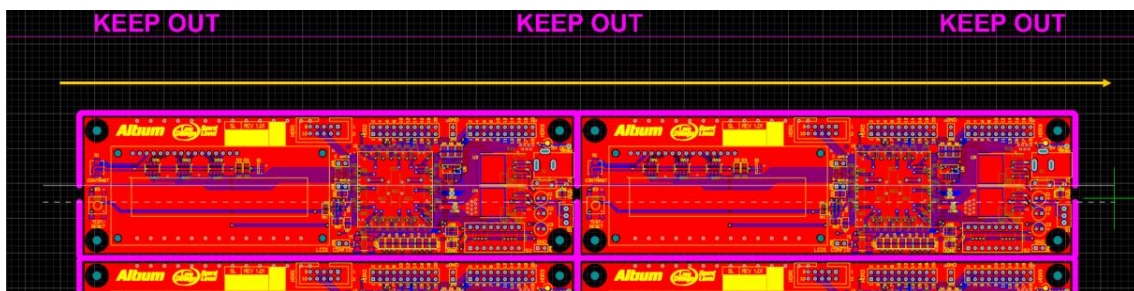


*Figure 12. Horizontal slice across entire panel*

33. Left-click to terminate the first cut. You will 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.

34. With the **Slice** command still active on the cursor, repeat this process 4 more times for the remaining rows of panelized boards. The `Route Tool` Path layer should look similar to Figure 13.



*Figure 13. Horizontal cuts for breakout tabs*

35. Now you 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, you can find it in the **Edit** menu.
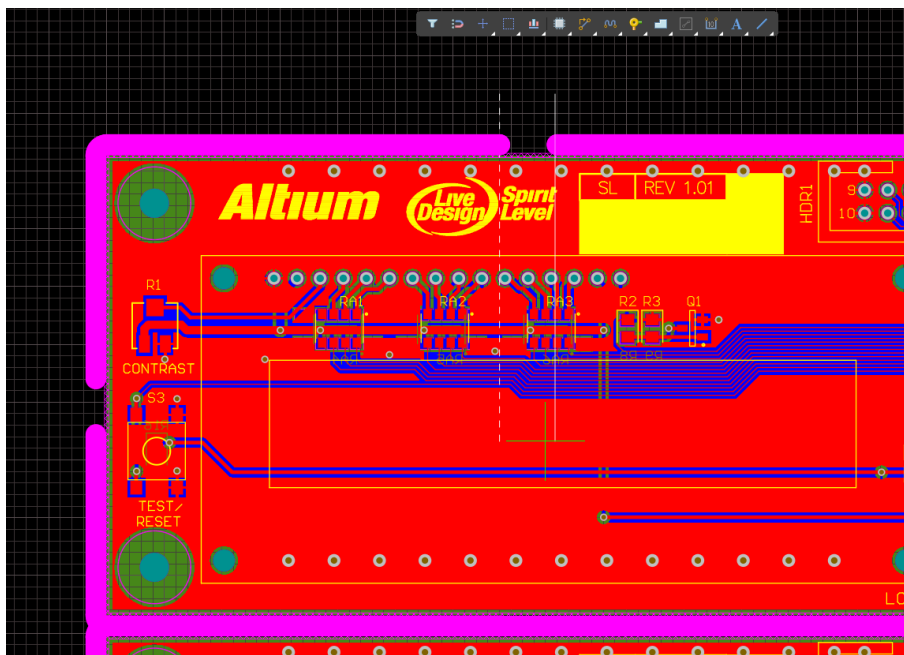


*Figure 14. Position of the Slice Tracks curser for first vertical cut*

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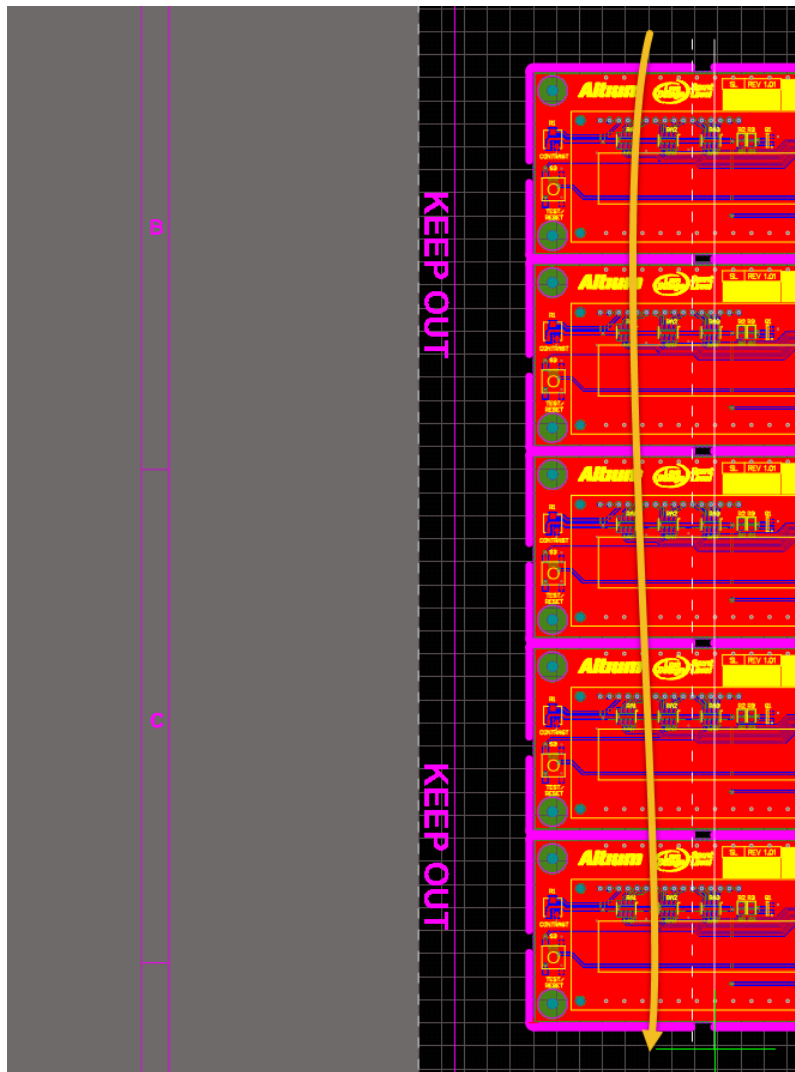


*Figure 15. Vertical cut across entire panel*

c) Left-click to complete the slice. The crosshair is still on your cursor.

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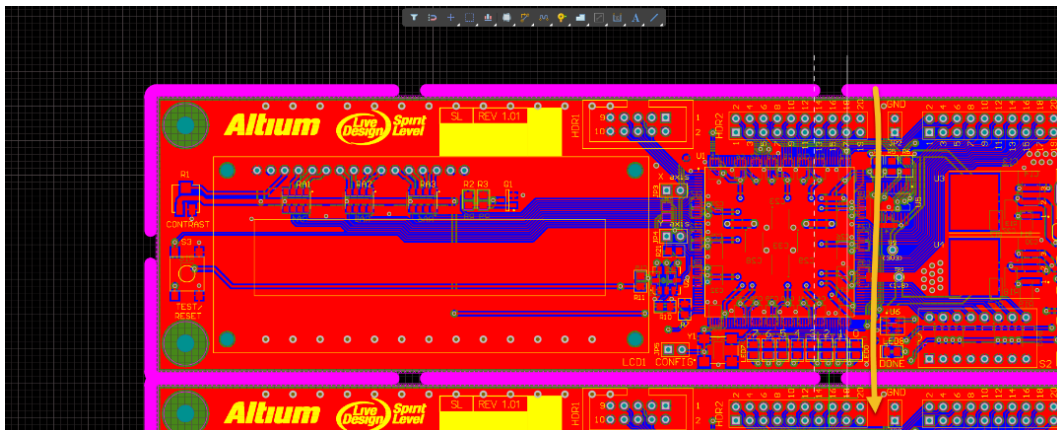


*Figure 16. Position of the Slice Tracks curser for second vertical cut*

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

b) Repeat the two vertical cuts for the second column of boards in the panel, Figure 17.

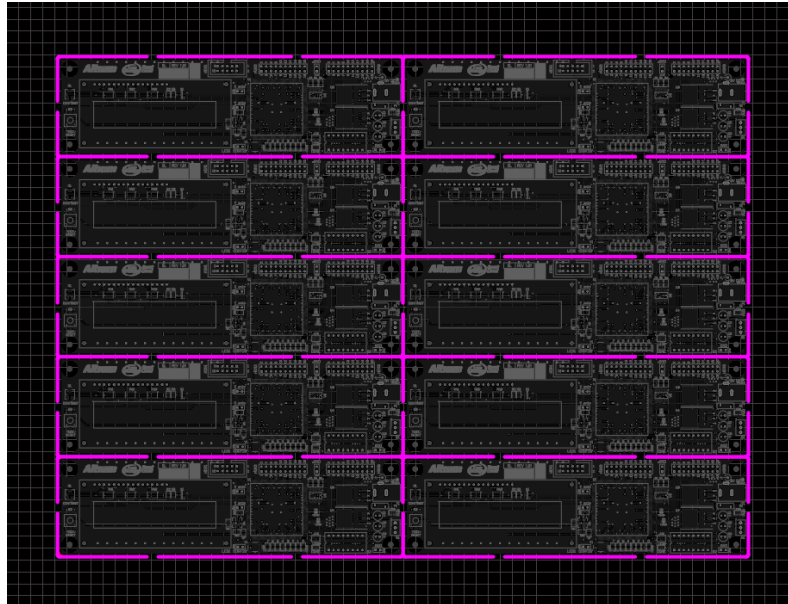c) Right-click to exit the command once completed.



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

37. View the panel in 3D and observe the new route tool path.

38. Switch back to the 2D view.

39. Open the *Preferences* ⚙ and follow the steps below:

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** to easily place the breakout tabs.
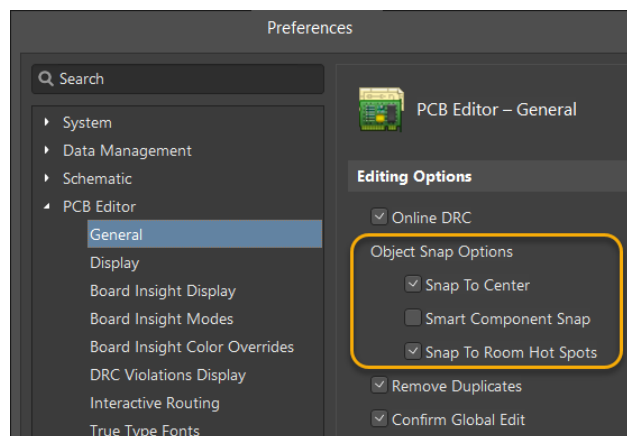
d) Select **OK** to save and close the *Preferences*.



*Figure 18. PCB Preferences*

40. From the layer tab bar, ensure that the `Route Tool Path` layer is still the active layer.

41. With nothing selected in the PCB, hit the **G** key, and change the grid to `50 mil`.

42. Open the *Properties* panel, General tab.

43. 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 it's disabled.

b) In the **Snapping section**, ensure it's set to **Current Layer.**

c) In the *Objects for Snapping*, enable **Track/Arcs Vertices** and **Track/Arcs Lines**. Please note that you will likely have other options enabled in this section.
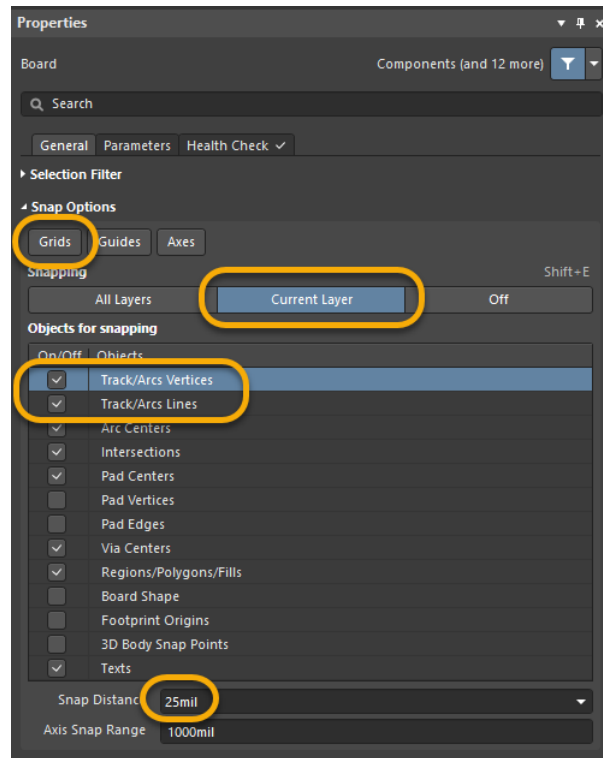
d) Set the *Snap Distance* to `25mil.`



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

44. Open the *Components* panel from the **Panels** button.

45. From the library drop-down menu, select category `Mechanical`, Figure 20.
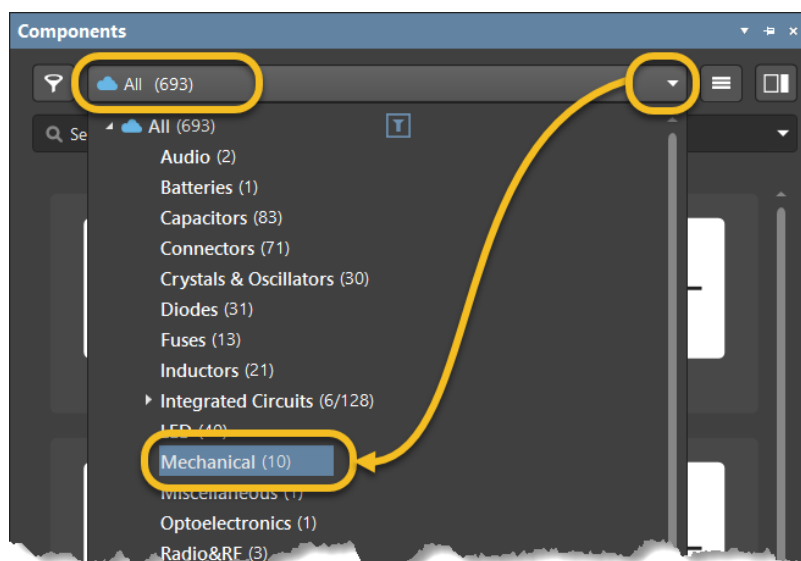


*Figure 20. Mechanical components in the Components panel*

46. If needed, add the word `Breakout` to the search field. As a result, the list is reduced to two search results.

47. Right-click on the `BREAKOUT SINGLE SIDE` footprint and select **Place**, Figure 21.
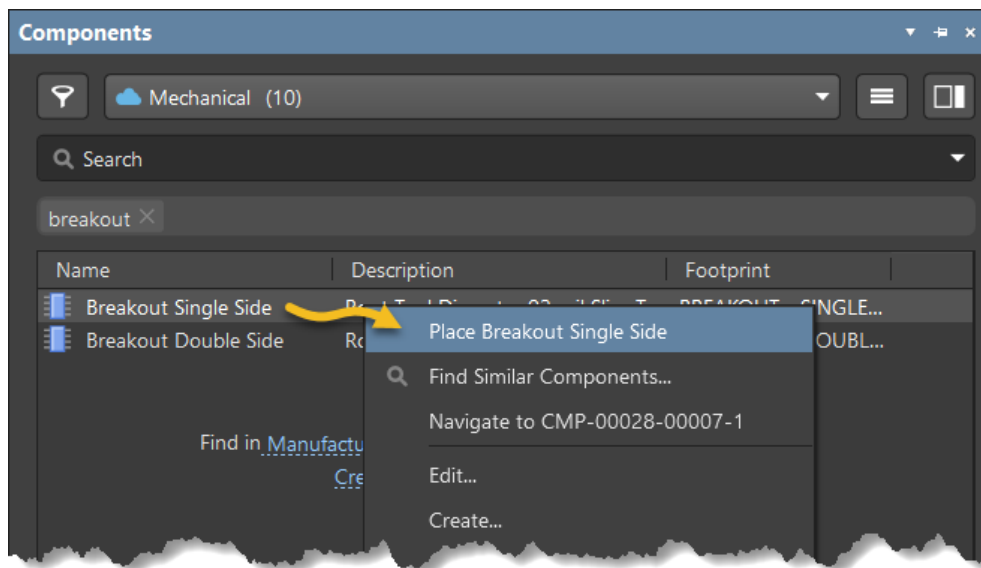


*Figure 21. Placing the breakout footprint*

48. Before placing the footprint, hit the **TAB** key to access the *Properties* panel.

49. Click on the eye 👁 icon next to the *Designator* field to hide it, as shown in Figure 22. You will notice that the `Designator1` value will no longer be visible in the PCB.
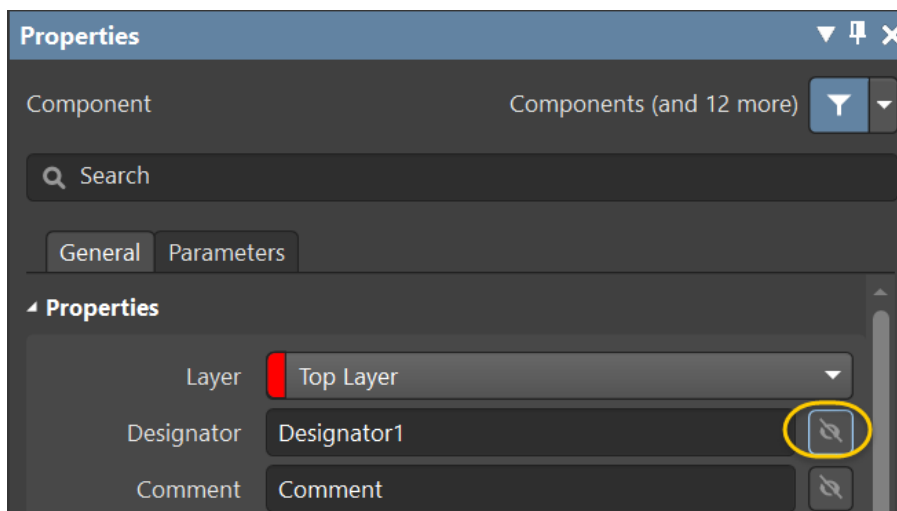


*Figure 22. Hide footprint designator*

50. Select the **Pause** icon ⏸ or the **ESC** key to resume the placement.

51. 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 23. Use the **Spacebar** to rotate the footprint as necessary.
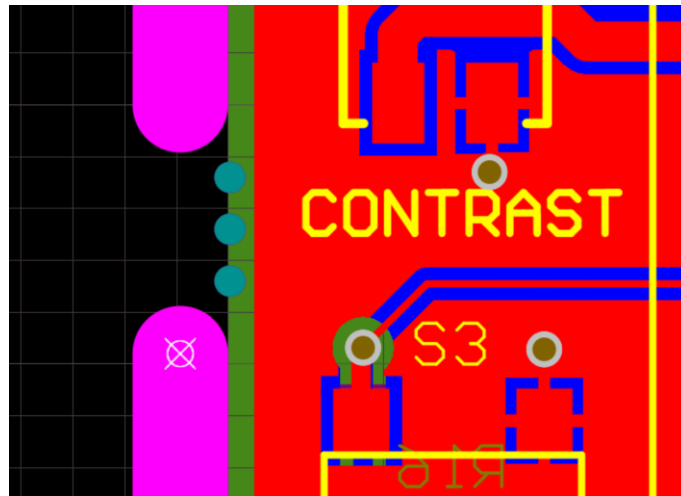


*Figure 23. Placing the breakout tab on the route path edge*

52. Once you've placed the first breakout footprint, rotate it 180 degrees and place another on the right-edge of the route path gap.

53. Once finished, right-click to exit the command.

54. From the `Breakouts` category, right-click on the `BREAKOUT DOUBLE SIDE` component and select **Place**. You will notice that this footprint is double-sided, allowing you to place it in between the board instances of our panel.

55. With the footprint on your cursor, place it on one of the voids on the inner route tool path, as shown in Figure 24.

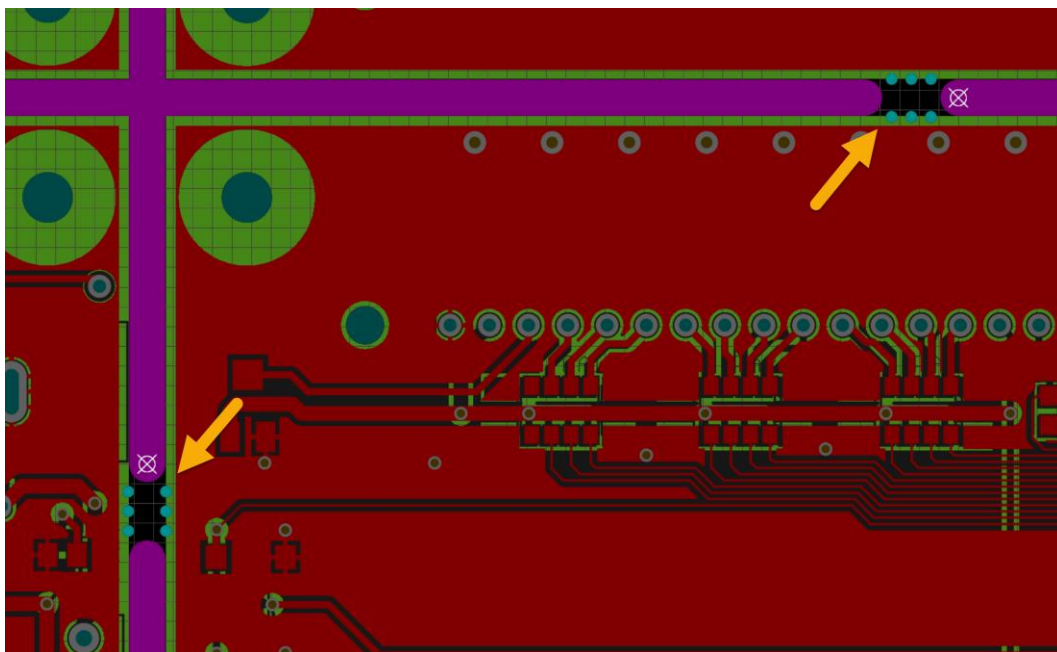56. Once placed, right-click to exit the command.



*Figure 24. Placement of breakout footprints, snapping reference to track ends*

57. View the panel in 3D and observe the new route tool path with the breakouts.
58. Switch back to 2D using the **2** key.
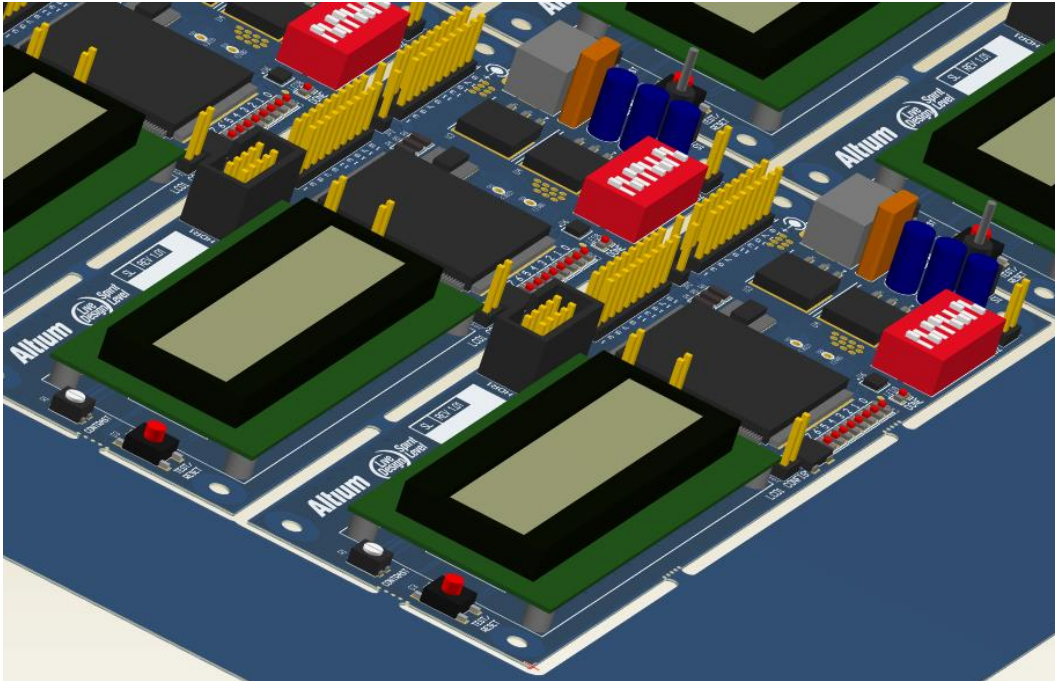59. Finish placing the remaining Breakout elements.



*Figure 25. Example of final panel with route guides and mouse bites*

# 7 Fabrication Files

60. Expand the Setting folder, then expand the Output Job Files folder to see Output Job File `Embedded Board Array.OutJob`. Open it with a double-click.

61. In the **Fabrication Outputs** section, single right-click on the **Gerber Files** header and select to see the Gerber settings, the Layer Table from the Panel PCB, and the source PCB. You can verify if the layers are compatible with the specifications required for fabrication.
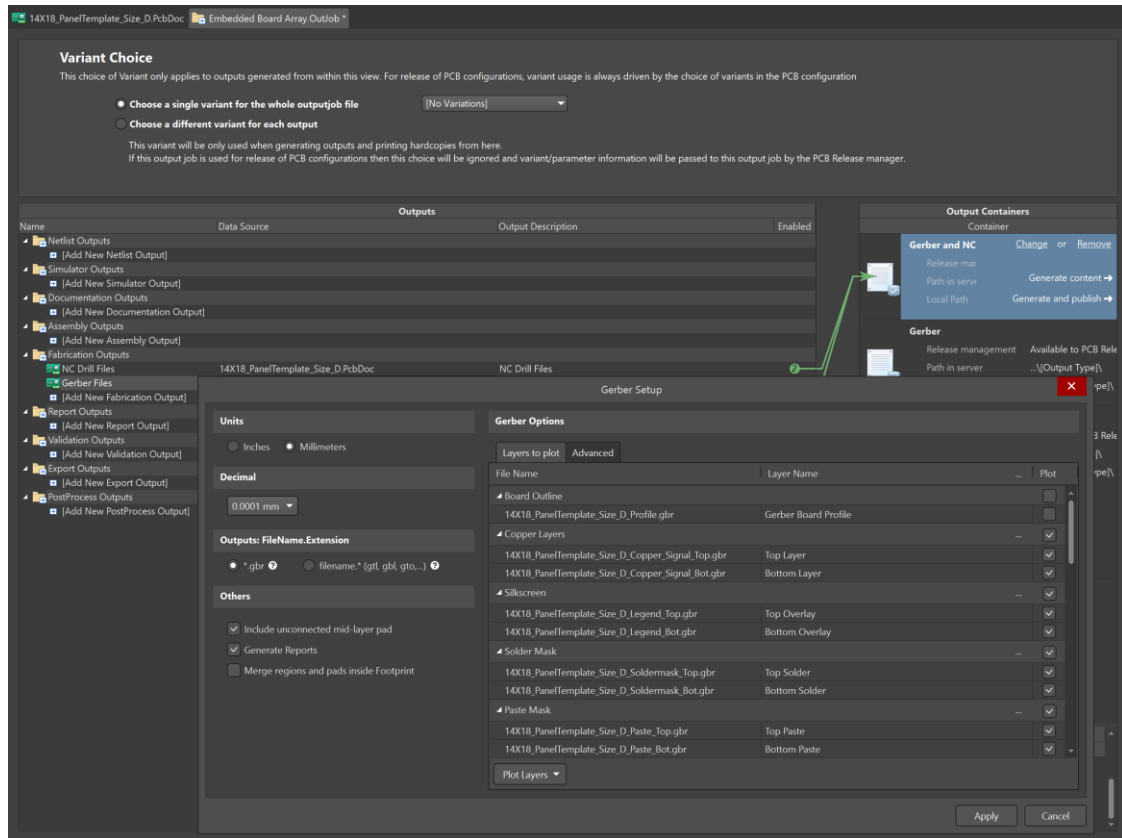


*Figure 26. Layer Table - Gerber generation*

# 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's always necessary to consult the fabrication and assembly houses for recommendations of the detail in the V-Score geometry.

62. Open the `14X18_Panel_VScore_D.PcbDoc` document.

63. Notice under the *Notes* section, just to the upper right of the board, that VSCORE details have been added, as shown in Figure 27.

64. In the training workspace, 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.

65. 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.
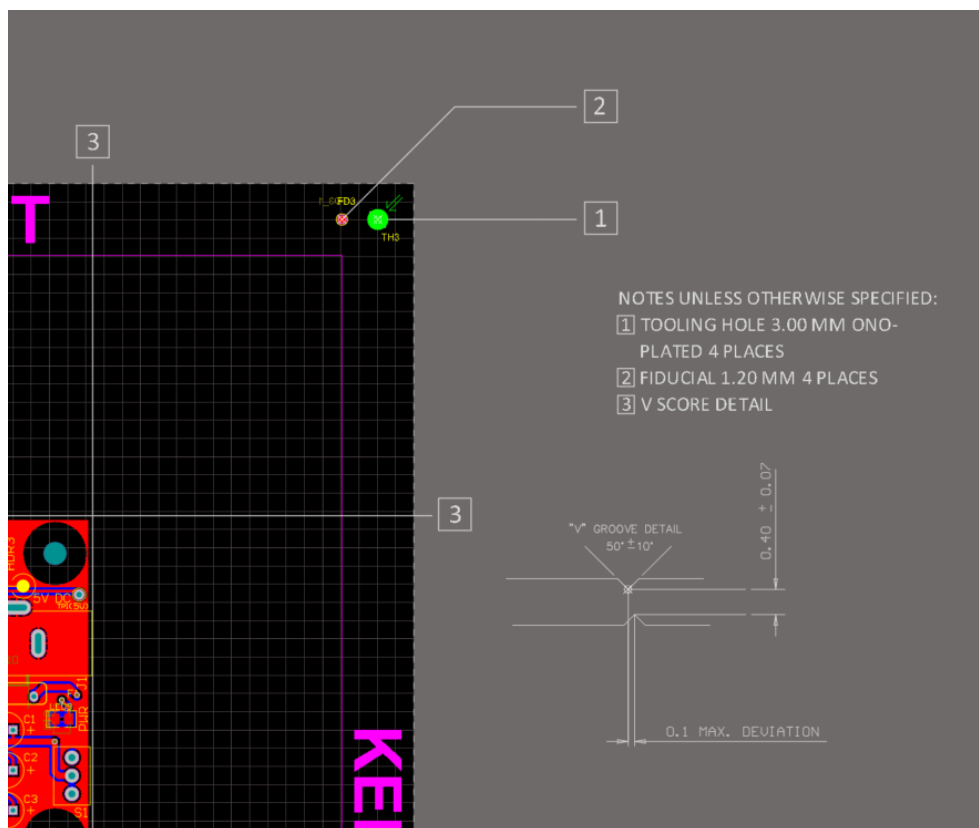


*Figure 27. VSCORE line details and callout*

Hint: With Altium Designer it is also possible to define a Mechanical Layer with the Layer Type "V Cut".

66. Save all documents using **File » Save All**.

67. Save the modifications to the server:

    a) In the *Projects* panel, next to the Project name you find the command **Save to Server** `Save to Server`.

    b) Select **Save to Server**.

    c) In the dialog *Save [Project Name]:*

        i) Add the comment `Embedded Board Array - [Add Your Name] - Finished`.

        ii) Select **OK**.

68. When ready, close the project and any open documents, **Window » Close All**.

# Congratulations on completing the Module!

## Embedded Board Array

## from

## Altium Designer Advanced Training
## with Altium 365

Thank you for choosing **Altium Designer**

**ALTIUM**
**TRAINING**