ALTIUM DESIGNER

ALTIUM 365

Altium Designer

Advanced Training with Altium 365

Rigid-Flex 2 – PCB Design

Altium®
TRAINING

Altium®
TRAINING

Software, documentation and related materials:

Table of Contents

Rigid-Flex 2 – PCB Design

1 Purpose

You can use the Board Planning Mode in Altium Designer to create flexible and rigid board regions, and split and bend lines of flexible circuits. How the Board Planning Mode works depends on how the layer stack is defined in the Layer Stack Manager. With the Board Planning Mode, you can allocate individual stacks to specific sections of the board. In this module, you will define three regions, corresponding to the three layer stacks available in the PCB. Also, you will explore how to define multiple regions in a single board outline using the Board Planning Mode.

2 Shortcuts
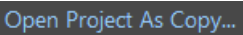
Shortcuts used when working with Rigid-Flex 2 – PCB Design

F1	Help
1	Board Planning Mode
2	2D Mode
3	3D Mode
E » O » S	Set Origin
G	Grid
Ctrl+Shift+G	Set Global Snap Grid...
V » F	View Fit Board
Num 0	PCB 3D View – Isometric View
Num 1	PCB 3D View – Top View
Num 5	PCB 3D View – Fold the Flex-Rigid PCB
CTRL+S	Save Document

3 Preparation

1. Close all existing projects and documents.
2. Next, create a copy of the Training Project: `Rigid-Flex 2 – PCB Design`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button ⬚.
5. Navigate to the predefined Training Project `Rigid-Flex 2 – PCB Design` `(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: `Rigid-Flex 2 – PCB Design – [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 Defining Board Regions

4.1 Accessing Board Planning Mode

10. Open the PCB document.

11. To assist with defining the board shape, set the origin at the bottom left corner of the board using the command **Edit » Origin » Set**. Zoom in to the lower-left board corner and left-click to set.

12. Select menu **View » Board Planning Mode** (Key **1**) to begin the process of defining individual board regions. The current Board Region (Board Outline) is highlighted in green as shown in Figure 1 below.

> Hint: The hotkey for accessing Board Planning Mode is **1**. Once you are in this view, press the **2** key to return to the 2D board view, or **3** for the 3D view.
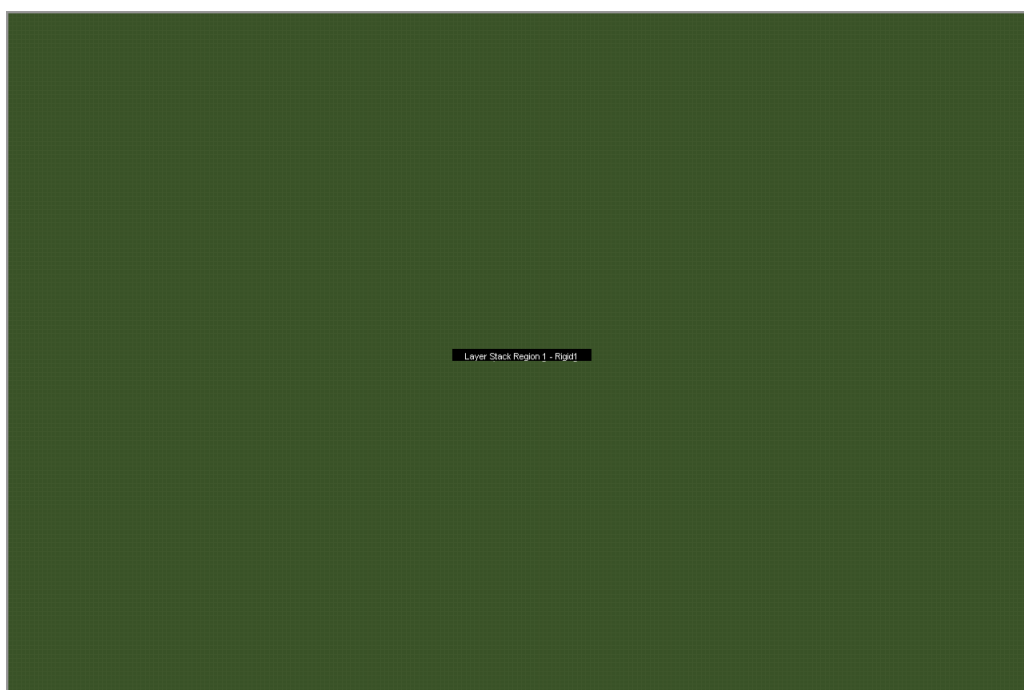
Figure 1. Default Board Planning Mode View

4.2 Creating the Board Outline

13. Ensure that you are currently in Imperial units. Press the **Q** hotkey to toggle units.

14. To make the creation of a board outline easier, set a coarse grid by pressing **Ctrl+Shift+G** and enter a value of `500 mils`.

> Hint: Use **Ctrl+G** as an alternative method in 2D Layout Mode.

15. Select the Board to see the handles. Modify the existing Board shape to specify the first board region by selecting and moving the Board outline. Refer to the current cursor position in the Status Bar in the bottom left corner or the x, dx, y, dy information in the Heads-Up Display. Use the dimensions below as a reference, Figure 2.
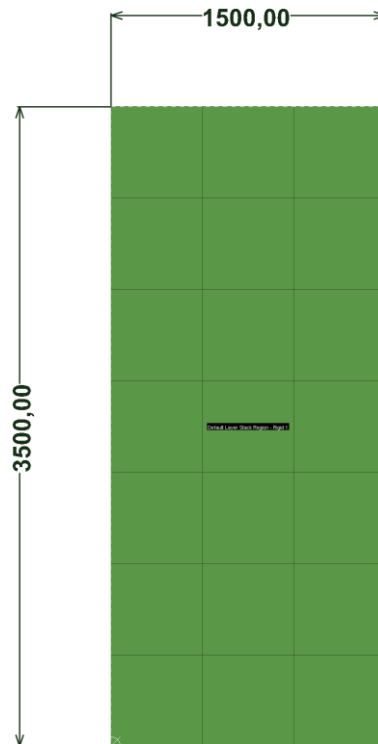
Figure 2. Defining the first Board Region (Mils)

16. Use the command **Place » Board Region** to create a second board region with the dimension 2000mil x 1000mils, as seen in Figure 3. If needed, change the Corner Mode with **Shift+Spacebar** to 90° corner.

Figure 3. Defining the second Board Region
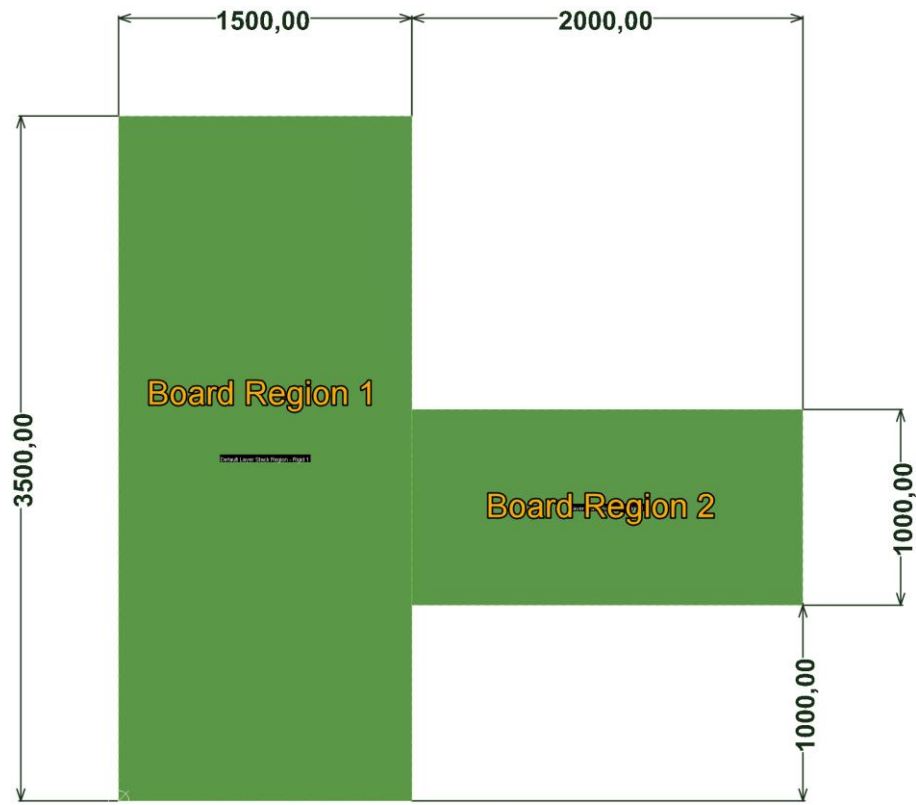
17. Create a third board region with the dimension 1500mil x 3500mils, as seen in Figure 4.
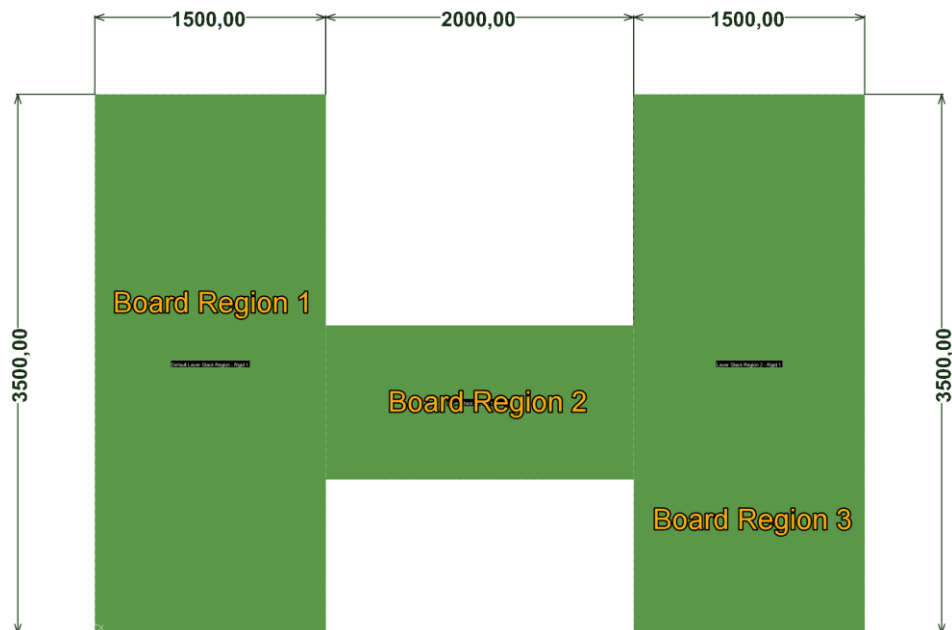

Figure 4. Defining the third Board Region

18. You should be able to click on the 3 Board Regions. This indicates that the board outline now has three sections, as shown in Figure 5 below.
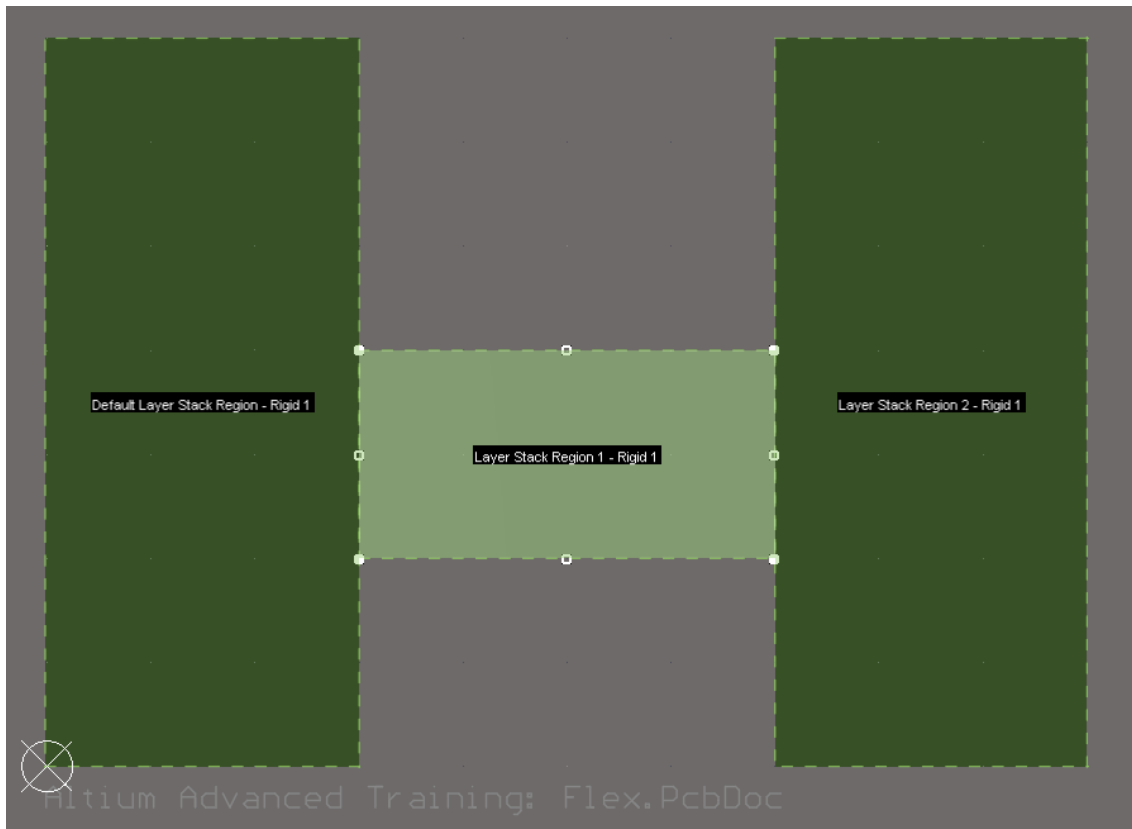
Figure 5. Board with three Board Regions

4.3 Assigning Layer Stacks

19. Double-click on the left Board Region. In the *Properties* panel dialog, follow the steps below:

    a) Change the name to `Rigid Layer Stack Region - Left`.

    b) Set the Layer Stack to **Rigid 1**, as shown in Figure 6.

    c) Ensure the **3D Locked** checkbox is enabled. The 3D Locked option will fix this section of your board outline to the workspace so it won't move when animated.

    d) Feel free to change the color, for example to blue.
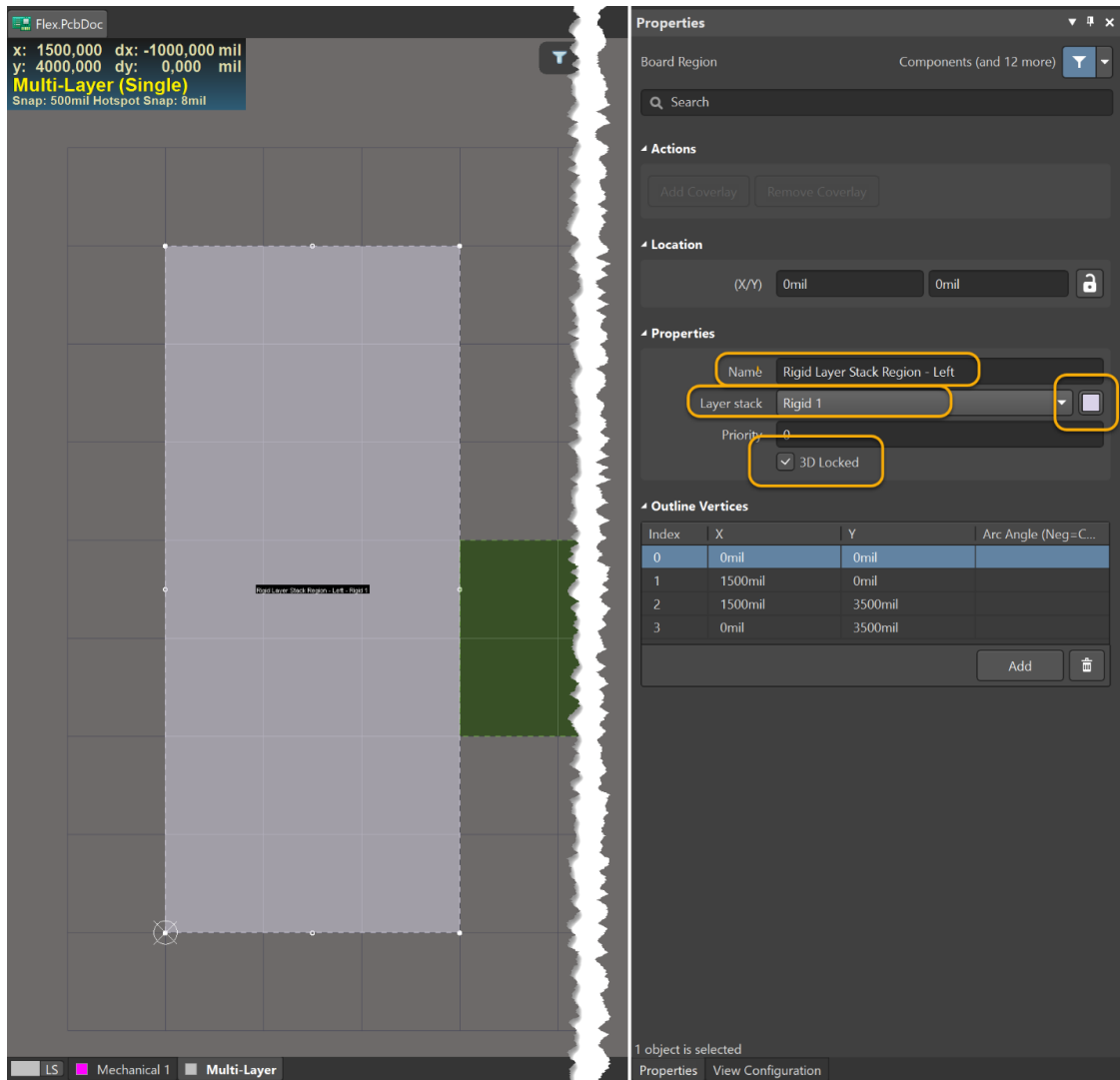
Figure 6. Left Board Region properties

20. Select the middle board region:

    a) Change the name to `Flex`.

    b) Set the Layer Stack to **Flex**.

    c) Control that the *3D Locked* option is not checked.

    d) Feel free to change the color.

21. Select the right board region:

    a) Change the name to `Rigid Layer Stack Region - Right`.

    b) Check that the Layer Stack is set to **Rigid 1**.

    c) Check that the *3D Locked* option is not checked.

22. The renamed and configured stacks are shown in Figure 7.
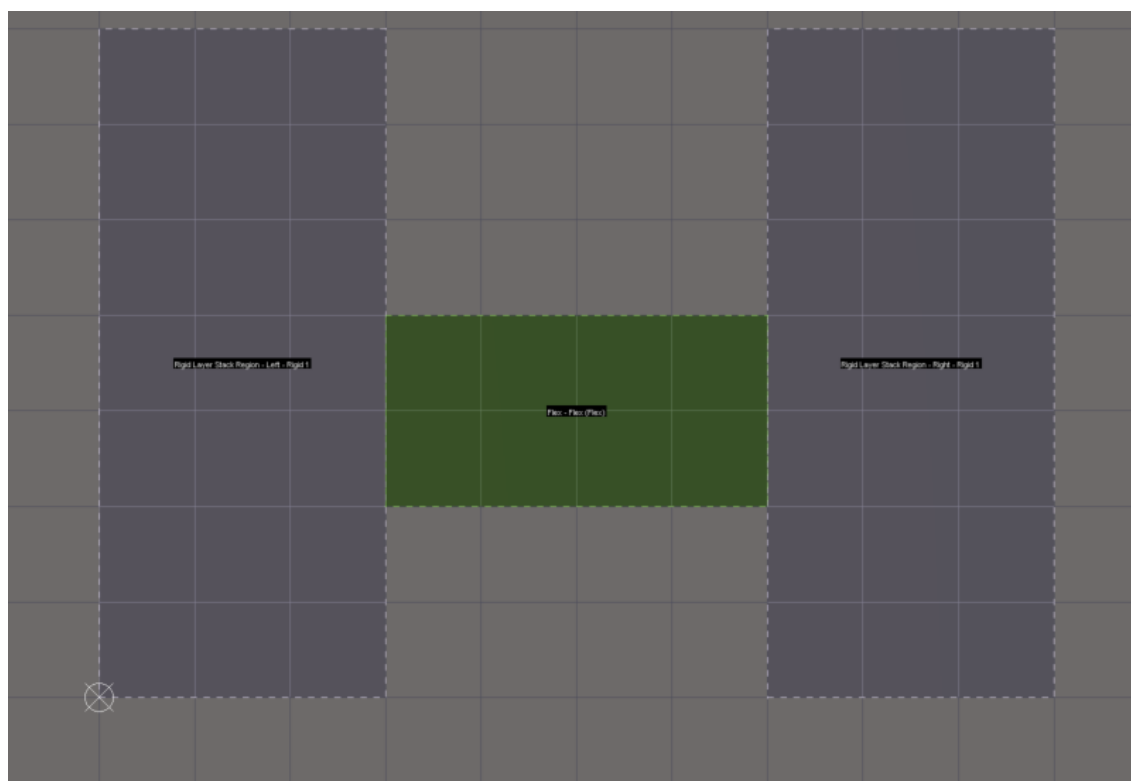
Figure 7. Layer Stacks assigned for Rigid-Flex design

5 Adding Layer Stack Table

You can add a layer stack table to your PcbDoc to document the details of your layer stacks. This is important to facilitate communication with your board manufacturer, especially with designs that have rigid and flex regions.

Also, you can prepare the documentation for the PCB using a Draftsman document.

23. Switch to the 2D view by pressing the **2** hotkey.

24. Rename Mechanical 1 layer to `Layer Stack` and make it the active layer as follows:

   a) Open the *View Configuration* panel by pressing the **L** hotkey.

   b) In the Mechanical Layers section, right-click on `Mechanical 1` and select **Edit Layer**.

      i) Change the name to `Layer Stack`.

      ii) Close the dialog with **OK**.

   c) Back to the *View Configuration* panel, change the *Active Layer* drop-down and select the layer **Layer Stack**.

25. Next, place the Layer Stack Table. See Figure 8 as a reference.

   a) From the menus, select **Place » Layer Stack Table.**

   b) Press the **Tab** key to modify the Table parameters in the *Properties* panel, as shown in Figure 8.

      i)   Verify that the **Layer Stack** is the selected layer.

      ii) Change the **Line Width** to `5mil`, **Text Height** to `30mil`, and **Stroke Width** to `5mil`.

      iii) Ensure the **Show Board Map** checkbox is enabled.

      iv) Activate the **Show Total Board Thickness** option.

      v)   Take a look at the *Columns* section to see the possible information that could be activated or deactivated for the table. Use the buttons below to add, delete, and move (up and down corresponds to right and left) columns as needed.

      vi) Select the **Pause** icon ⏸ to resume placement.

   c) Position the table outside the board area and left mouse click to add the table.

   d) Right mouse click to end placement mode.

26. From the *Properties* panel, you can change the **Slider** position, next to **Show Board Map** option, to increase or decrease the size, for example, set it to approximately 25%.
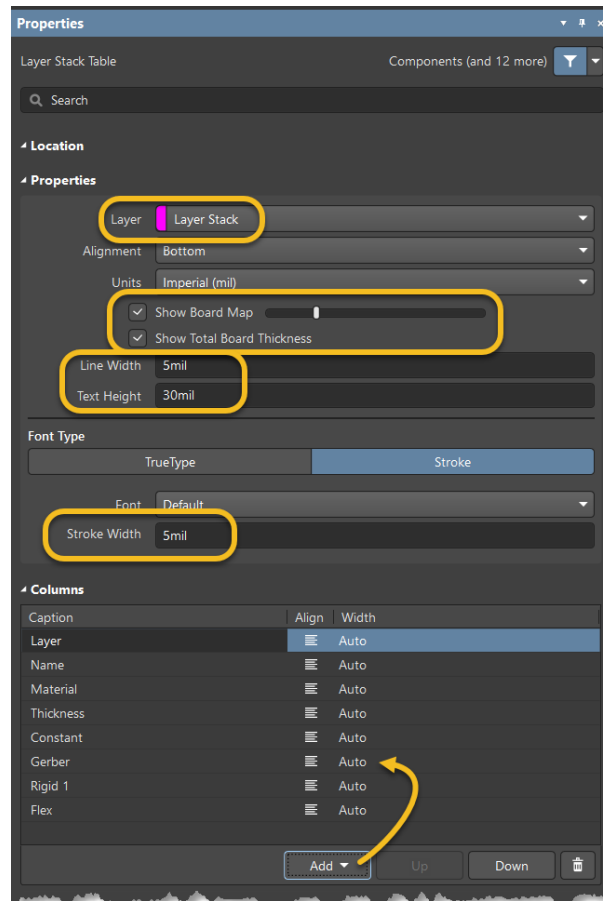
Figure 8. Modifying the Layer Stack Table

27. Below the row listings, there's a diagram that provides the legend for each region of the board, Figure 9. In the last two columns, the patterns referenced in the legend are used to show which layers exist in the three regions of the design.

Layer	Name	Material	Thickness	Constant	Gerber	Rigid 1	Flex
	Top Overlay				GTO		
	Flex Top Overlay				GTO		
	Flex Top Solder	SM-001	1,00mil	4	GTS		
	Top Solder	Solder Resist	0,40mil	3.5	GTS		
1	Top Layer		1,40mil		GTL		
	Dielectric 4	PP-006	2,80mil	4.1			
2	GND	CF-004	1,38mil		GP1		
	Dielectric 2	PP-006	2,80mil	4.1			
3	Mid 1	CF-004	1,38mil		G1		
	Dielectric 1	FR-4	12,60mil	4.8			
4	Mid 2	CF-004	1,38mil		G2		
	Dielectric 3	PP-006	2,80mil	4.1			
5	PWR	CF-004	1,38mil		GP2		
	Dielectric 5	PP-006	2,80mil	4.1			
6	Bottom Layer		1,40mil		GBL		
	Bottom Solder	Solder Resist	0,40mil	3.5	GBS		
	Flex Bottom Solder	SM-001	1,00mil	4	GBS		
	Flex Bottom Overlay				GBO		
	Bottom Overlay				GBO		

Total board thickness (Rigid 1): 32,91mil

Figure 9. Stack Layer Table added to PCB Document with region legend

6 Populating the Design

6.1 Adding Components

28. Prior to importing components, press **G** and select **25 mils** to change the grid.

29. Synchronize components from the schematic using the command **Design » Import Changes from Rigid Flex Design.PrjPcb** and select **Execute Changes**.

> Note: If the ECO displays a warning for compiler errors, you can ignore it this time.

30. Position the LCD display on the bottom layer of the `Rigid Layer Stack Region – Right` section of the board. Press **L** while moving `LCD1` to flip it to the bottom layer.

31. Place the remaining components on the top side of the `Rigid Layer Stack Region – Left` board, as in the example shown in Figure 10.
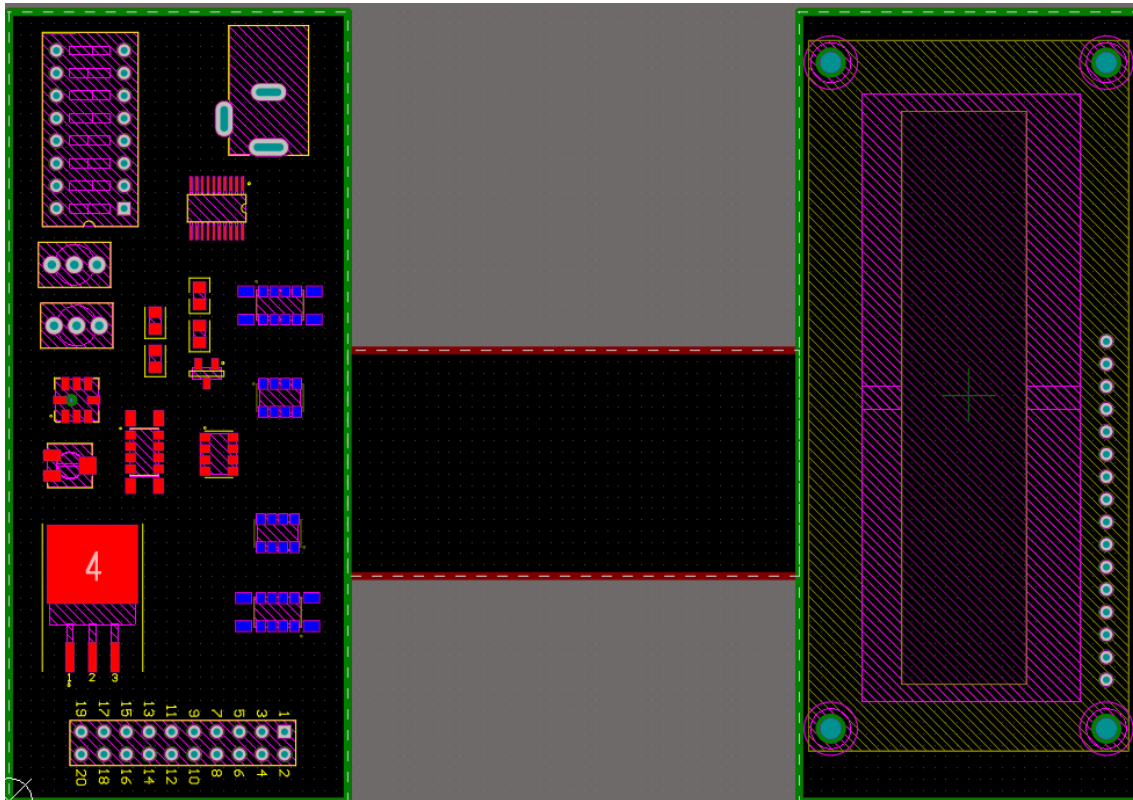

Figure 10. Components position on Rigid1 (top side) and Rigid2 (bottom side)

7 Animating Flex Behavior

7.1 Adding Bending Lines

Next, you need to specify how and where the Flex portion should bend. You will use bending lines.

32. Enter Board Planning Mode by pressing the **1** hotkey.

33. Change the grid size to `100 mils`, **Ctrl+Shift+G or Shift+G**.

34. Select **Place » Define Bending Line** then click at (1700, 2000) and (1700, 1000) to place a bending line. When placing the bending line, the first click defines the starting point, and the second click define end point.

35. Add a second bending line between (3300, 2000) and (3300, 1000). Right-click to exit the command.

36. Add another bending line in the middle of the flex board between (2500, 2000) and (2500, 1000). The three lines should appear, as shown in Figure 11.
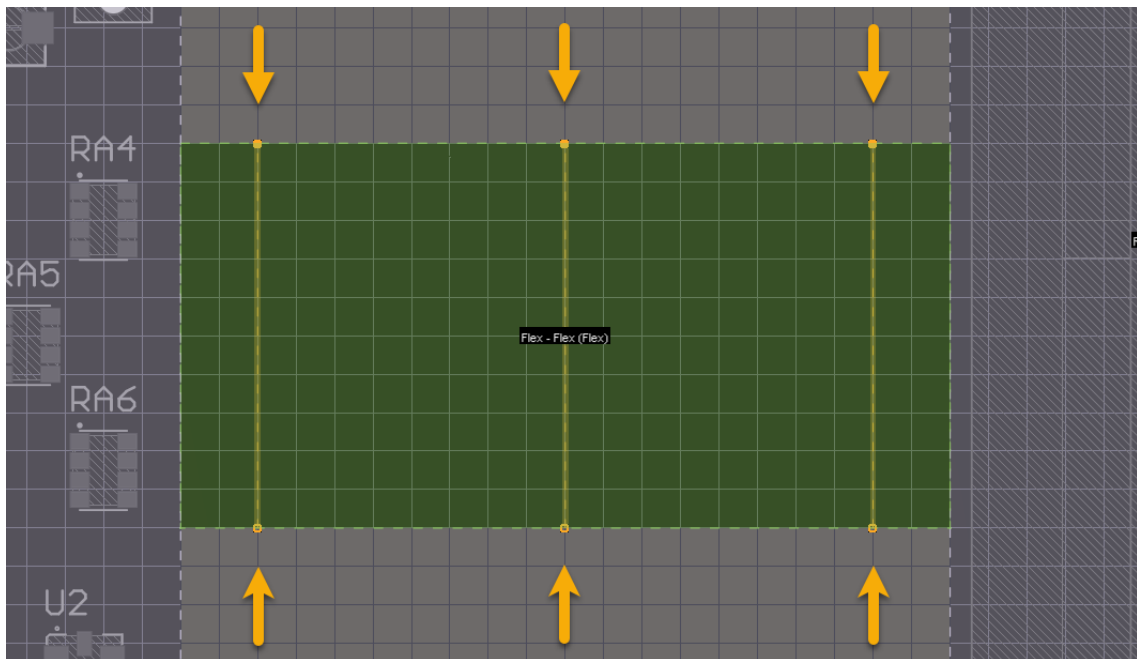

Figure 11. Three Bending Lines added to Flex region

37. Next, access Layer Stack Region section:

   a) Open the *PCB* panel for modifying the bending line parameters.

   b) Select **Layer Stack Regions** from the top drop-down list.

   c) In the *Layer Stacks* section, select the *<All Stacks>* Layer Stack.

   d) In the *Stackup Regions* section, click on Region **Flex**.

   e) If you click on a Bending line, the corresponding bend will be highlighted in the PCB, as shown in Figure 12.

Hint: If you don't see the Bending Lines in the PCB panel, try the following solution:
Select the bending lines in the PCB and change the values in the *Properties* panel, instead of using the *PCB* panel to configure the bending lines
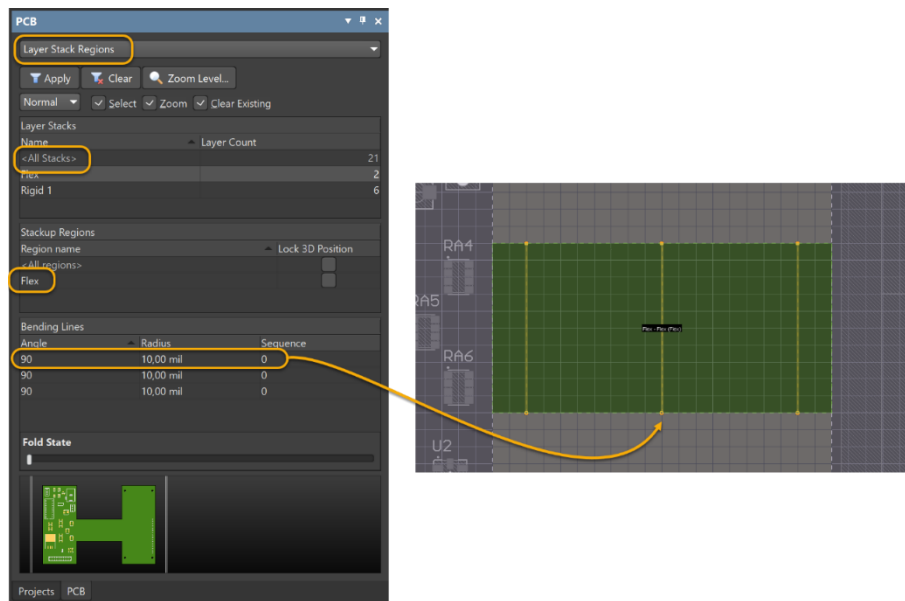

Figure 12. PCB panel to modify behavior of bending lines

38. Modify the Flex Bending radius and bend sequence as described below:

   a) Double-click on the Bending line in the panel and change the values in the dialog to change the Bending Line values.

   b) Select the far-right bending line and add the following values in the Bending Line dialog:

      i) **Fold Index** of 1

      ii) **Bending Angle** of 25

      iii) **Radius** of 100mil, Figure 13.

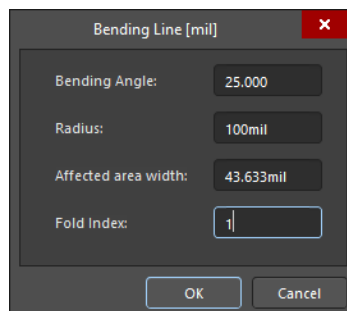*Figure 13. Bending Line Properties*

c) Select the middle bending line and add the following values in the bending Line dialog:

    i) **Fold Index** of 2

    ii) **Angle** of 130

    iii) **Radius** of 100

d) For the left bending line, add the following values in the bending Line dialog:

    i) **Fold Index** of 3

    ii) **Angle** of 25

    iii) **Radius** of 100

Notice that a yellow width of the bending line indicates how much local bending will occur.

Altium
TRAINING

7.2 Action!

39. Go to 3D viewing mode.

40. Position the board, as shown below in Figure 14.

Figure 14. Position Rigid-Flex Design in 3D view

41. Select menu **View » 3D View Control » Fold/Unfold** or press the **5** key. The board should fold up, as shown in Figure 15 with the LCD above the lower rigid region. Press the **5** hotkey to unfold the design.

Figure 15. Folded Boards in 3D folded one on top of the other

42. In the *PCB* panel, slide the **Fold State** slider to manually fold the board, see Figure 16.
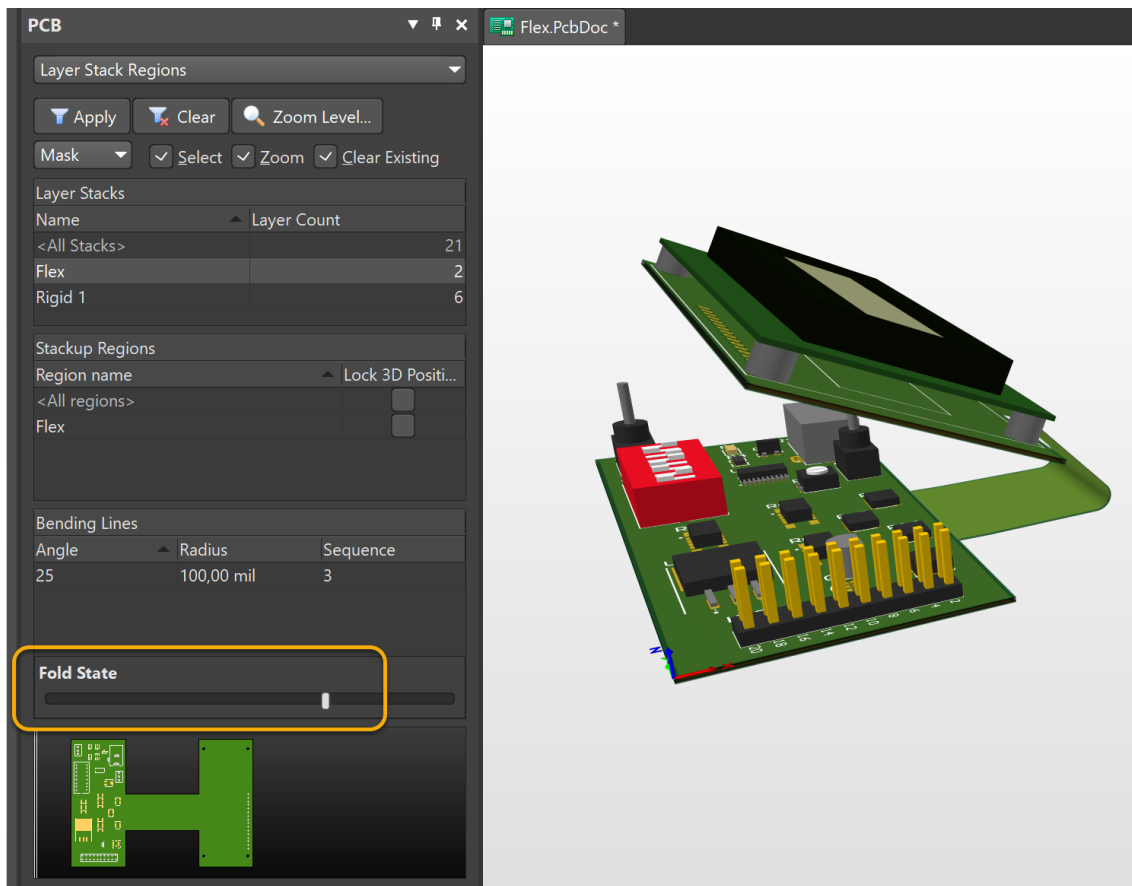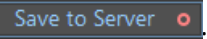

Figure 16. Fold State Slider

43. Feel free to experiment with different Line bending parameters in the *PCB* panel.

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

45. 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 `Rigid-Flex 2 – PCB Design – [Add Your Name] – Finished`.

        ii) Select **OK**.

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

Congratulations on completing the Module!

Rigid-Flex 2 – PCB Design

from

Altium Designer Advanced Training
with Altium 365

Thank you for choosing **Altium Designer**

ALTIUM®
TRAINING