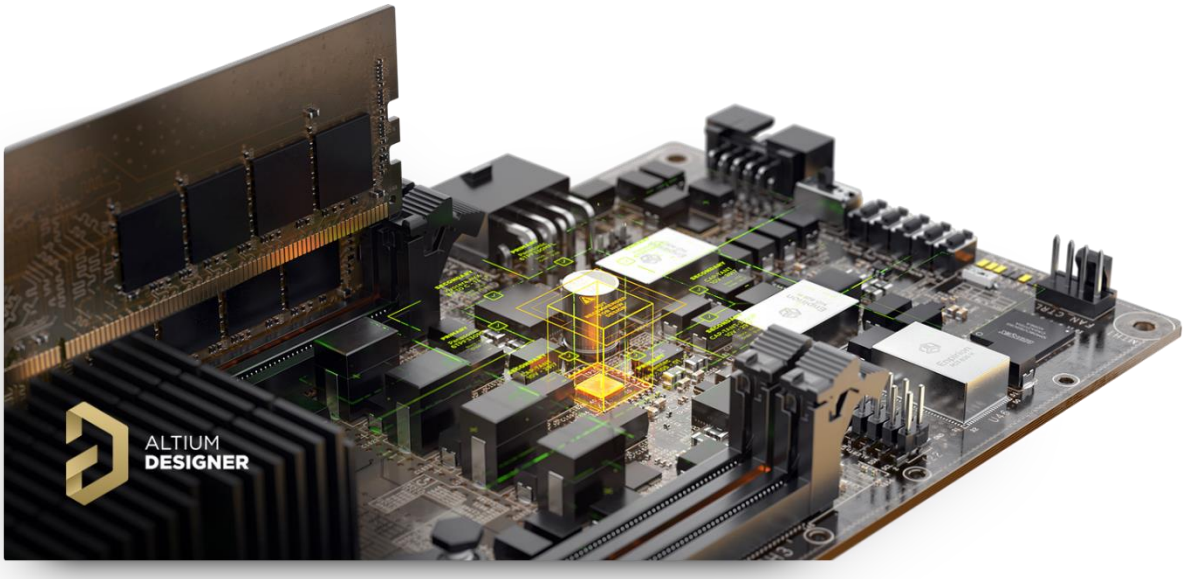


# Altium Designer

## Advanced Course

Module: Rigid-Flex Design
Advanced Mode

Software, documentation and related materials:

Copyright © 2022 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in the whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties.
ACTIVEBOM®, ActiveRoute®, Altium 365™ , Altium Concord ProTM, Altium Designer®, Altium Vault®, Altium NEXUS™, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, Draftsman®, DXP™, Easytrax®, EE Concierge™, xSignals®, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, Upverter™, X2®, XSignals® and their respective logos are trademarks or registered trademarks of Altium LLC or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.

# Table of Contents


1.1 Purpose 3

1.2 Shortcuts 3

1.3 Preparation 3

1.4 Defining Board Regions 4

1.4.1 Accessing Board Planning Mode ..... 4

1.4.2 Creating the Board Outline ..... 4

1.4.3 Assigning Layer Stacks ..... 7

1.5 Add Layer Stack Table 8

1.6 Populating the Design 10

1.6.1 Adding Components ..... 10

1.7 Animating Flex Behavior 11

1.7.1 Adding Bending Lines ..... 11

1.7.2 Action! ..... 14

# Rigid-Flex Design Advanced Mode

## 1.1 Purpose

Altium Designer's Board Planning Mode enables the creation of both flexible and rigid board regions as well as enabling 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 Manger. The Board Planning Mode allows designers to allocate individual stacks to specific sections of the board. In this module we will define 3 regions corresponding to the 3 layer stacks we have available in the PCB, and explore how the Board Planning Mode allows us to define multiple regions in a single board outline.

Rigid Flex advanced mode is available for Altium Designer 22.3 and subsequent versions.

Please check that the following settings are activated in the *Preferences.*
*System - General* > Button **Advanced**:
**PCB.RigidFlex2.0**
**PCB.RigidFlex.SubstackPlaning**

Check that the *Layer Stack* is set to **Advanced Mode**

## 1.2 Shortcuts

Shortcuts when working with Rigid-Flex Design Advanced Mode

| | |
|---|---|
| **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** |

## 1.3 Preparation

1. **Close all existing projects and documents**.
2. Open the `Rigid Flex Design.PrjPCB` project found in its respective Advanced Training folder.
3. Open the PCB document `Flex.PcbDoc`.

# 1.4 Defining Board Regions

## 1.4.1 Accessing Board Planning Mode

4. 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.
5. Select menu **View » Board Planning Mode** (Key **1**) to begin the process of defining individual board regions. The current Board Region / Board Outline will be highlighted in green as shown in Figure 1 below.

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 three-dimensional view.

*Figure 1. Default Board Planning Mode View*

## 1.4.2 Creating the Board Outline

6. Ensure that you are currently in Imperial units. Press the **Q** hotkey to toggle units.
7. 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`. (Use **Ctrl+G** as an alternate method in 2D Layout Mode)
8. 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 at the Heads Up Display. Use the dimensions below as a reference, Figure 2.

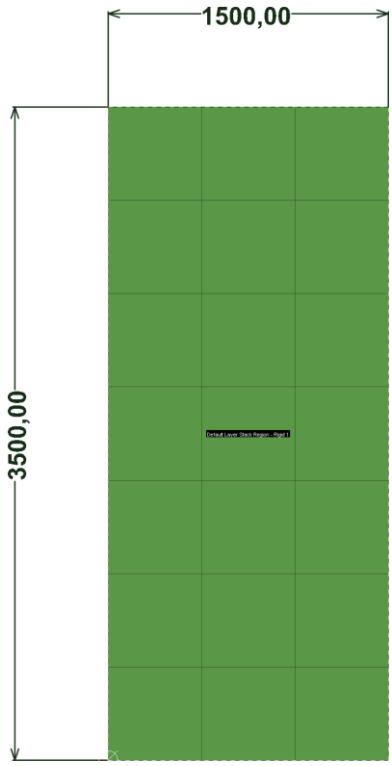


*Figure 2. Defining the first Board Region (Mils)*

9. Use the command **Place » Board Region** to create a second board region with the dimension 2000mil x 1000mils as seen in Figure 3

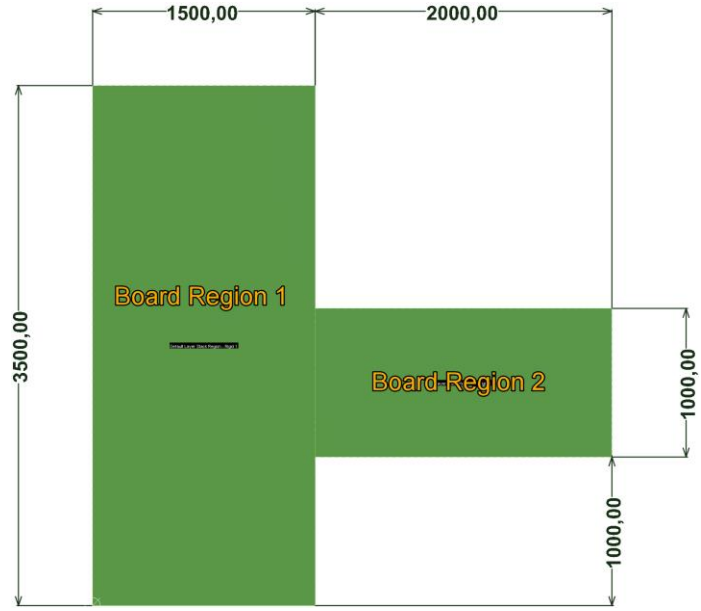


*Figure 3. Defining the second Board Region*

10. Use the command **Place » Board Region** to create a third board region with the dimension 1500mil x 3500mils, as seen in Figure 4.

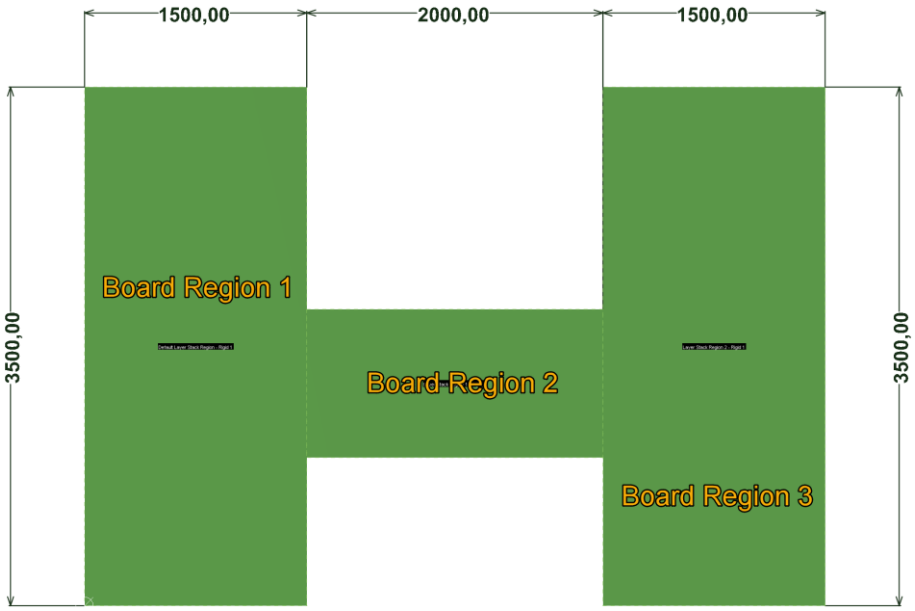


*Figure 4. Defining the third Board Region*

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

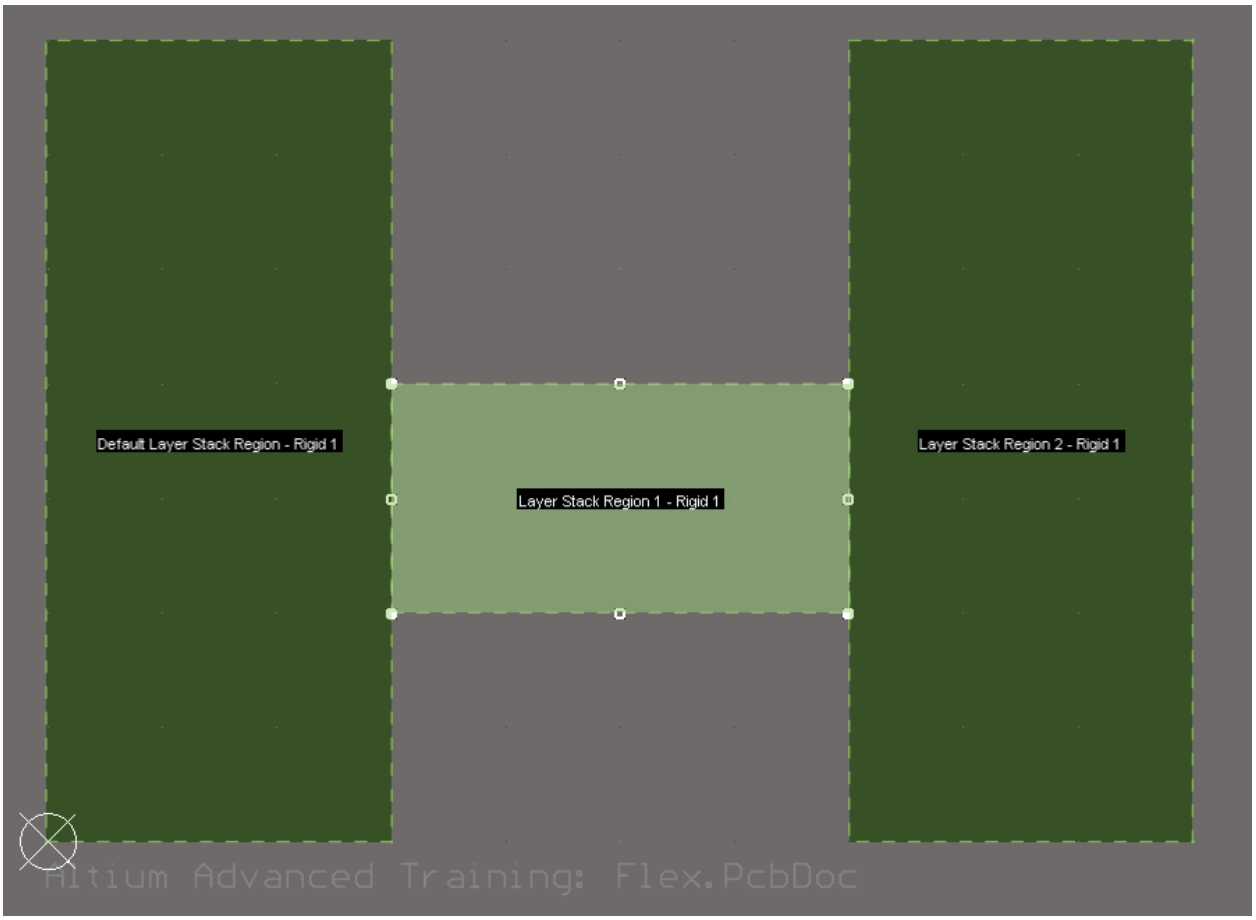


*Figure 5. Board with three Board Regions*

## 1.4.3 **Assigning Layer Stacks**

12. Double click on the left Board Region. In the *Properties* panel dialog that appears:
    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 will not move when animated.
    d) Feel free to change the color (e.g. to blue)

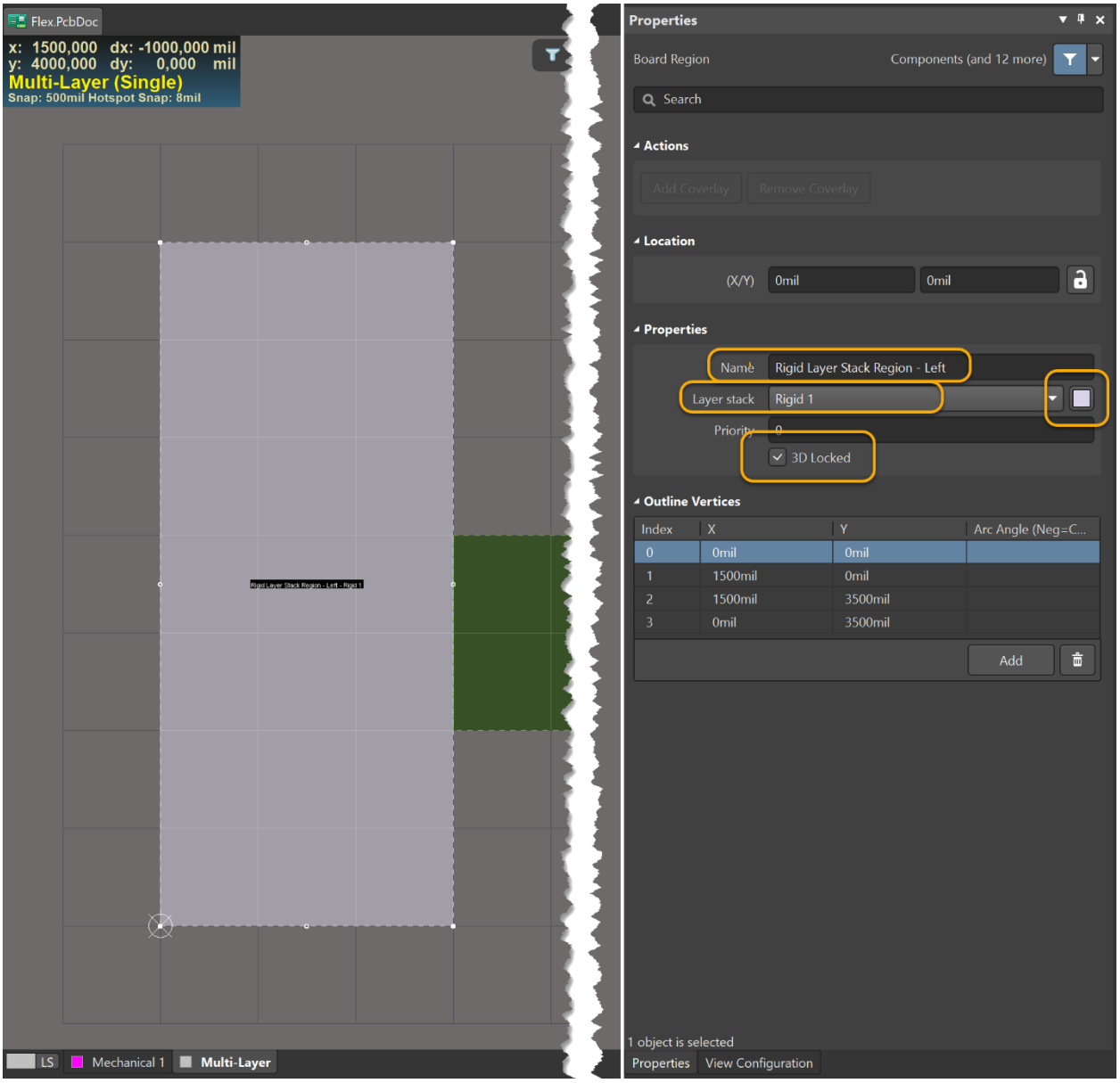


*Figure 6. Left Board Region properties*

13. Double-click 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.

14. Double click on the right board region:
    a) Change the name to `Rigid Layer Stack Region - Right`.
    b) Check that the Layer Stack is set to **Rigid1**.
    c) Check that the *3D Locked* option is not checked.
15. The renamed and configured stacks are shown in Figure 7.

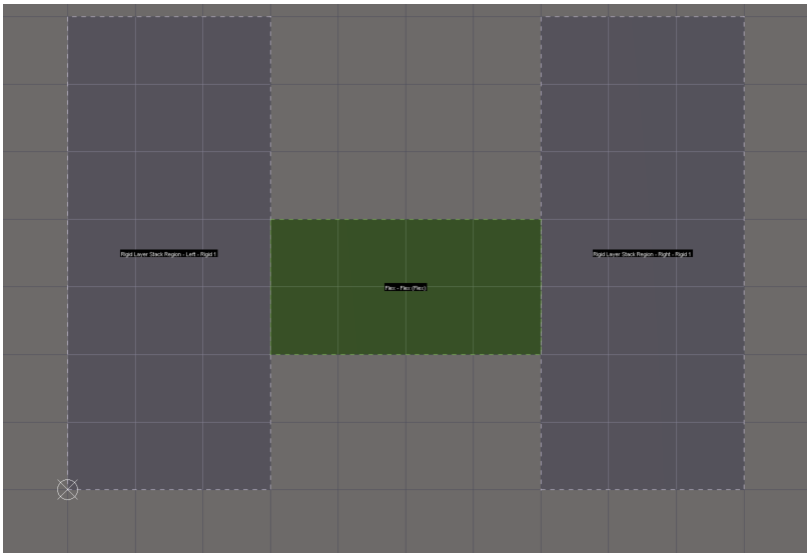

*Figure 7. Layer Stacks assigned for Rigid-Flex design*

# 1.5 Add Layer Stack Table

A layer stack table can be added to your PcbDoc to fully document the details of your layer stacks. This is important to facilitate communication with your board manufacturer, especially with designs that have both rigid and flex regions.
The Documentation for the PCB can also be done with a Draftsman document.

16. Switch to the 2D view by pressing the **2** hotkey.
17. 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**.
    c) Change the name to `Layer Stack`.
    d) Change the *Active Layer* drop-down and select **Layer Stack**.
18. Next, we will 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) Change the slider next to *Show Board Map* to increase/decrease the size. Set it to approximately **25%.**
        v) Activate the **Show Total Board Thickness** option.

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

vii) Click 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.

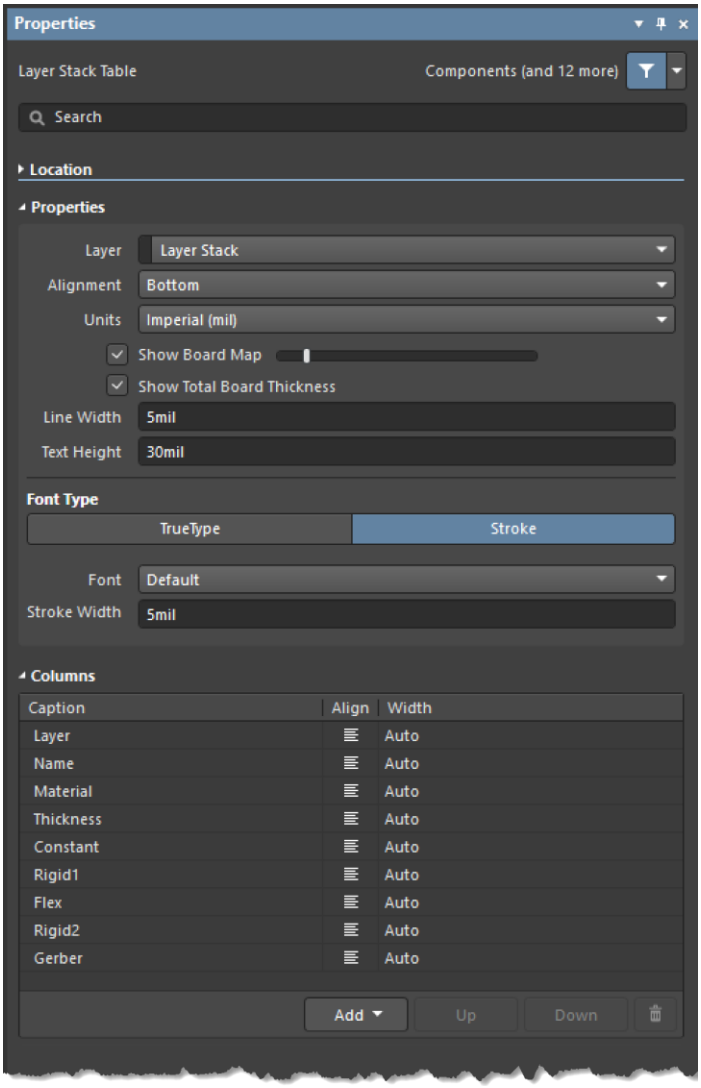


*Figure 8. Modifying the Layer Stack Table*

19. Left click to place the table below your board outline as shown in Figure 9. Notice that there is a diagram below the row listings that provides the legend for each region of the board. In the last 3 columns, the patterns referenced in the legend are used to show which layers exist in the 3 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 | | | | |

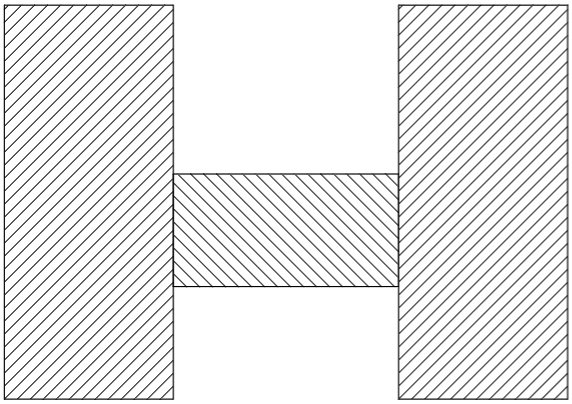

*Figure 9. Stack Layer Table added to PCB Document with region legend*

# 1.6 Populating the Design

## 1.6.1 **Adding Components**

20. Prior to importing components, change the grid by pressing **G** and selecting **25 mils**.
21. Synchronize components from the schematic using the command **Design » Import Changes** from **Rigid Flex Design.PrjPcb** and click **Execute Changes**.
    Note: If the ECO will display a warning for compiler errors this can be ignored at this time.
22. 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.
23. Place the remaining components on the top side of the `Rigid Layer Stack Region - Left` board as shown (as example) in Figure 10.

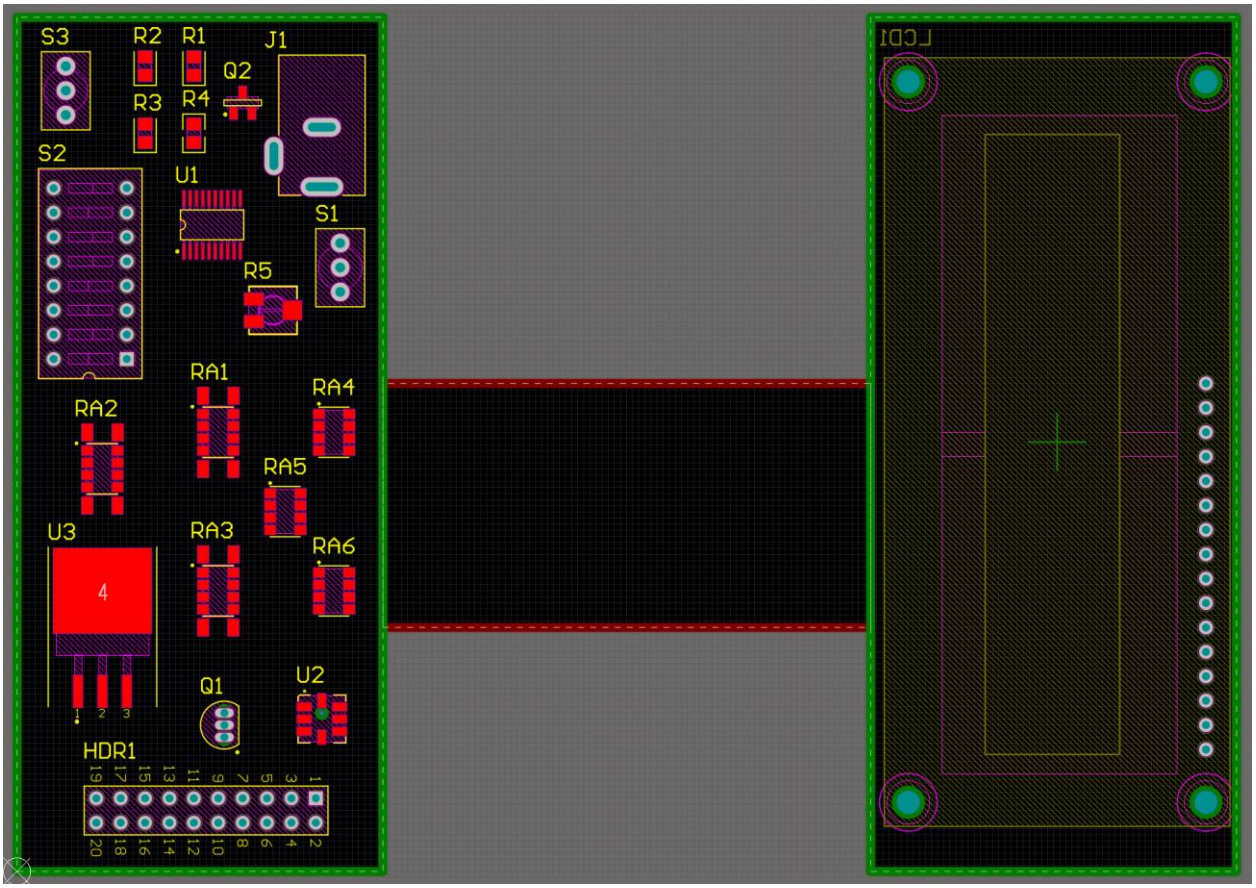


*Figure 10. Components position on Rigid1 (top side) and Rigid2 (bottom side)*

# 1.7 Animating Flex Behavior

## 1.7.1 **Adding Bending Lines**

Now we need to specify how and where the Flex portion will bend. This is done with bending lines.

24. Enter Board Planning Mode by pressing the **1** hotkey.
25. Change the grid size to `100 mils`, **Ctrl+Shift+G**.
26. 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.
27. Add a second bending line between (3300, 2000) and (3300, 1000). Right click to exit the command
28. 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.

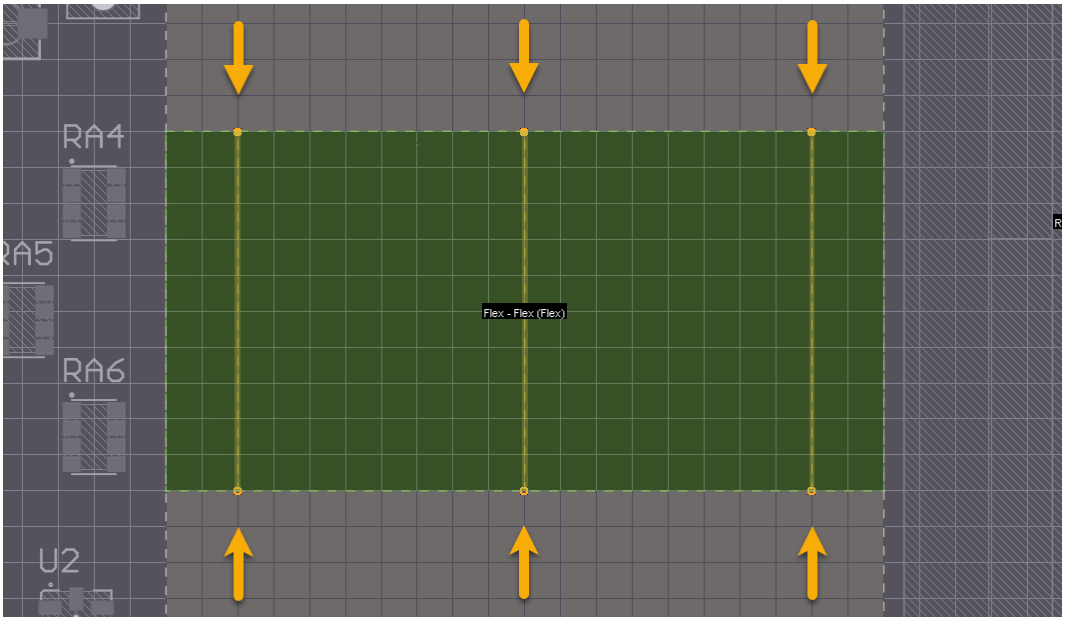


*Figure 11. Three Bending Lines added to Flex region*

29. 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, click on the *<All Stacks>* Layer Stack.
    d) In the *Stackup Regions* section, click on Region **Flex**.
    e) Clicking on a Bending line, the corresponding bend will highlight in the PCB as shown in Figure 12.

If you do not see the Bending Lines at the PCB panel.
Instead of using the *PCB* panel to configure the bending lines you can select the bending lines at the PCB and change the values in the *Properties* panel.

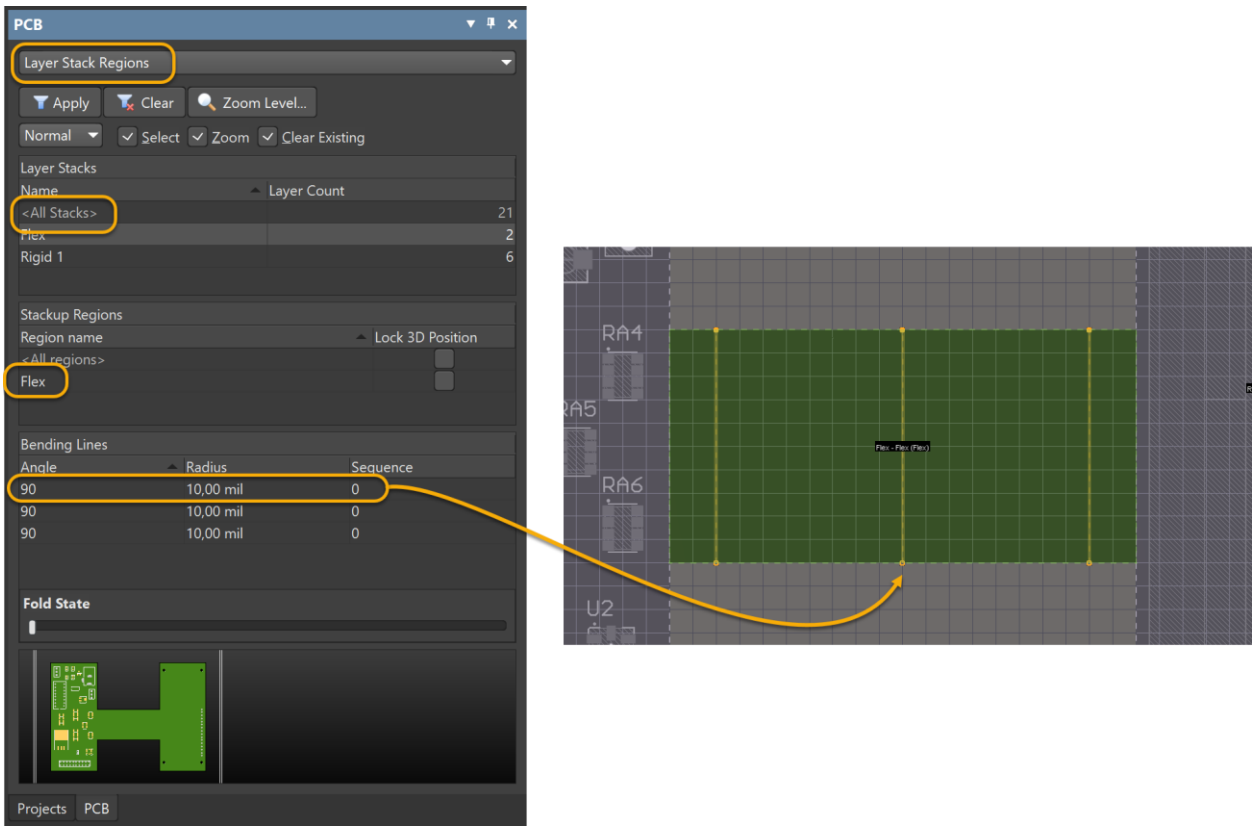


*Figure 12. PCB panel to modify behavior of bending lines*

30. Modifying the Flex Bending radius and bend sequence:
    a) To change the Bending Line values, double-click on the Bending line in the panel and change the values in the dialog.
    b) Select the far-right bending line assign it a Sequence value - **Fold Index** of `1`, **Angle** of `25` and **Radius** of `100mil` as shown in Figure 13.

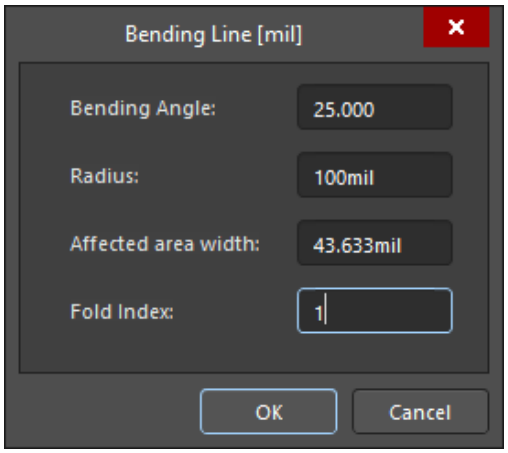


*Figure 13. Bending Line Properties*

    c) Select the middle bending line: assign it a **Fold Index** of `2`, **Angle** of `130`, and **Radius** of `100`.
    d) For the left bending line: assign a **Fold Index** of `3`, **Angle** of `25`, and **Radius** of `100`. Notice that a yellow width of the bending line indicates how much local bending will occur.

## 1.7.2 **Action!**

31. Go to 3D viewing mode.
32. Position the board as below in Figure 14.

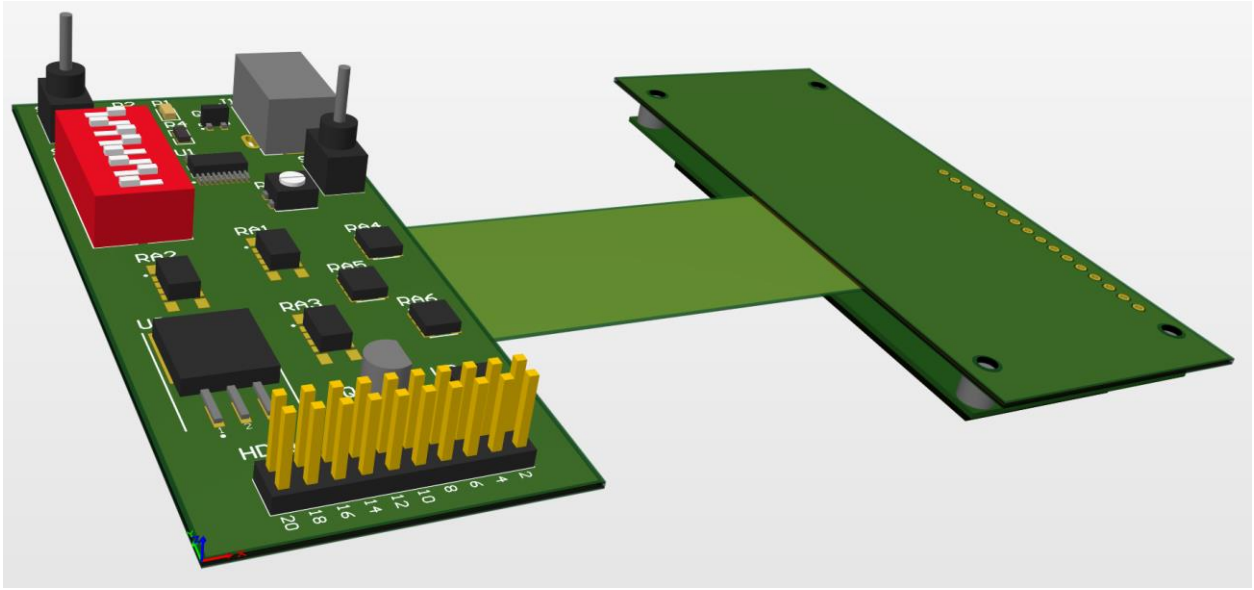

*Figure 14. Position Rigid-Flex Design in 3D view*

33. Select menu **View » 3D View Controll » 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.

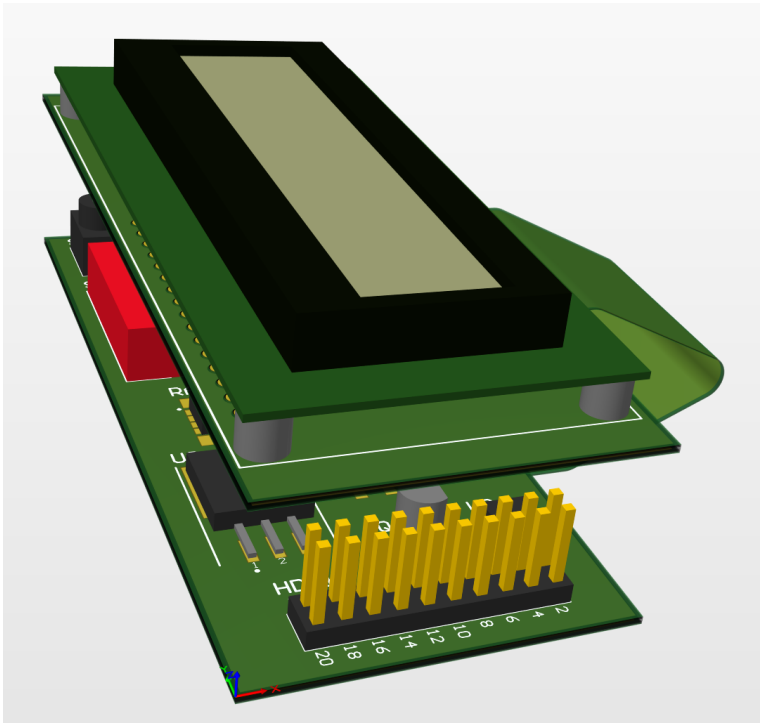

*Figure 15. Folded Boards in 3D folded one on top of the other*

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

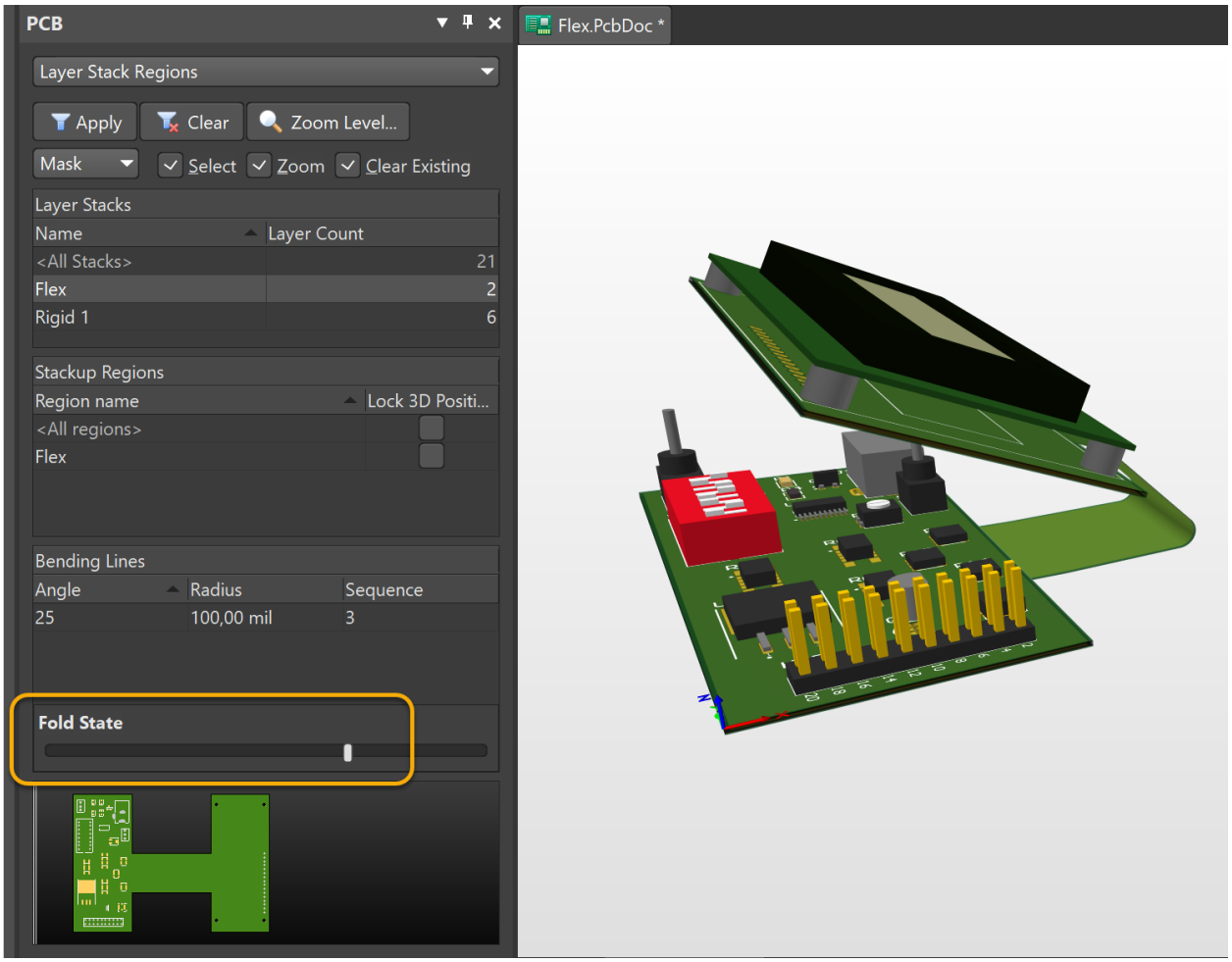


*Figure 16. Fold State Slider*

35. Feel free to experiment with different Line bending parameters in the *PCB* panel.
36. Feel free to save your modifications.
37. **Close the project and any open documents.**

**Congratulations on completing module**

Rigid-Flex Design Advanced Mode

**from the**

**Altium Designer Advanced Course**

**Thank you for choosing Altium Designer**