What is Rigid-Flex?

As the name suggests, a flexible printed circuit is a pattern of conductors printed onto a flexible insulating film. Rigid-flex is the name given to a printed circuit that is a combination of both flexible circuit(s) and rigid circuit(s), as shown in the image above. This combination is ideal for exploiting the benefits of both flexible and rigid circuits - the rigid circuits can carry all or the bulk of the components, with the flexible sections acting as interconnections between the rigid sections.

Flexible circuit technology was initially developed for the space program to save space and weight. They are popular today as they not only save space and weight - making them ideal for portable devices such as mobile phones and tablets - they can also: reduce packaging complexity by substantially reducing the need for interconnect wiring; improve product reliability due to reduced interconnection hardware and improved assembly yields; and reduce cost, when considered as part of the overall product manufacture and assembly costs.

Flexible circuits are normally divided into 2 usage classes: static flexible circuits, and dynamic flexible
circuits. Static flexible circuits (also referred to as use A) are those that undergo minimal flexing, typically during assembly and service. Dynamic flexible circuits (also referred to as use B) are those that are designed for frequent flexing; such as a disk drive head, a printer head, or as part of the hinge in a laptop screen. This distinction is important as it affects both the material selection and the construction methodology. There is a number of layer stack-up configurations that can be fabricated as rigid-flex, each with their own electrical, physical and cost advantages.

**Rigid-Flex Design**

Designing a flex or rigid-flex circuit is very much an electromechanical process. Designing any PCB is a 3 dimensional design process, but for a flex or rigid-flex design the 3 dimensional requirements are much more important. Why, because the rigid-flex board may attach to multiple surfaces within the product enclosure, and this attachment will probably happen as part of the product assembly process. To ensure that all sections of the finished board fit in their folded location within the enclosure, it is strongly recommended that a mechanical mock up (also known as a paper doll cut out) is created. This process must be as accurate and realistic as possible with all possible mechanical and hardware elements included, and both the assembly-time phase and the finished assembly must be carefully analyzed.

For an engaging discussion on the materials, technologies and processes used in rigid-flex design, as well as information about the challenges involved with the production of a rigid-flex board, download and read the free [Rigid-Flex Guidebook](#) by Altium’s Ben Jordan.

**Materials Used in Flexible Circuit Manufacture**

Flex circuits are created from a stackup of flexible substrate material and copper, laminated together with adhesive, heat and pressure.

The most common substrate is polyimide, a strong, yet flexible thermosetting polymer (thermoset). Examples of polyimides often used in the manufacture of flexible circuits include: Apical, Kapton, UPLEX, VTEC PI, Norton TH and Kaptrex. Note that these are registered trade names, owned by their respective trademark holders.

The copper layer is typically rolled and annealed (RA) copper, or sometimes wrought copper. These forms of copper are produced as a foil and offer excellent flexibility. They have an elongated grain, it is important to orient this correctly in a dynamic flex circuit to achieve the maximum flexing lifespan. This is achieved by orienting the dynamic flex circuit along the roll (so the circuit bends in the same way the foil was coiled on the roll). The flex manufacturer normally deals with this during the preparation of fabrication panels, it only becomes an issue if the designer performs their own circuit panelization (referred to as nesting in flex circuit design). The copper foil is typically coated with a photo-sensitive layer, which is then exposed and etched to give the desired pattern of conductors and termination pads.

The adhesive is typically acrylic, and as the softest material in the structure, introduces the greatest number of manufacturing challenges. These include: squeeze-out, where the adhesive is squeezed out into openings cut into the cover layers to access copper layers; Z-axis expansion defects due to the higher CTE (coefficient of thermal expansion) of acrylic adhesive; and moisture out gassing due to the higher rate of moisture absorbance, which can result in resin recession, blow outs and
delamination at plated through hole sites. Alternative adhesives and adhesive-less processes are available, these may be more appropriate in less cost-sensitive applications.

A simplified view of how a flexible circuit is manufactured, the materials are laminated together under heat and pressure.

**Flex and Rigid-Flex Layer Stackup Types**

There are a number of standard stackups available for flex and rigid-flex circuits, referred to as *Types*. These are summarized below.

The Types are defined in the following standards:

- **IPC 6013B** - *Qualification and Performance Specification for Flexible Printed Boards*
- **MIL-P-50884E** - *Military Specification: Printed Wiring Board, Flexible or Rigid-Flex, General Specification for* (available [here](#))

### Type 1 - Single Layer

Single-sided flexible wiring containing one conductive layer and one or two polyimide outer cover layers.

- One conductive layer, either laminated between two insulating layers or uncovered on one side.
- Access holes to conductors can be on either one or both sides.
- No plating in component holes.
- Components, stiffeners, pins and connectors can be used.
- Suitable for static and dynamic flex applications.

### Type 2 - Double Layer

Double-sided flexible printed wiring containing two conductive layers with plated through holes, with or without stiffeners.

- Two conductive layers with an insulating layer between; outer layers can have covers or exposed pads.
- Plated through-holes provide connection between layers.
- Access holes or exposed pads without covers can be on either or both sides; vias can be covered on both sides.
- Components, stiffeners, pins and connectors can be used.
- Suitable for static and dynamic flex applications.

Type 3 - Multilayer

Multilayer flexible printed wiring containing three or more conductive layers with plated-through holes, with or without stiffeners.

- Three or more flexible conductive layers with flexible insulating layers between each one; outer layers can have covers or exposed pads.
- Plated through-holes provide connection between layers.
- Access holes or exposed pads without covers can be on either or both sides.
- Vias can be blind or buried.
- Components, stiffeners, pins and connectors can be used.
- Typically used for static flex applications.

Type 4 - Multilayer Rigid-Flex

Multilayer rigid and flexible material combinations (Rigid-Flex) containing three or more conductive layers with plated-through holes. Rigid-flex has conductors on the rigid layers, which differentiates it from multilayer circuits with stiffeners.

- Two or more conductive layers with either flexible or rigid insulation material as insulators between each one; outer layers can have covers or exposed pads.
- Plated through-holes extend through both rigid and flexible layers (apart from blind and buried vias).
- Access holes or exposed pads without covers can be on either or both sides.
- Vias or interconnects can be fully covered for maximum insulation.
- Components, stiffeners, pins, connectors, heat sinks, and mounting brackets can be used.
A Type 4 rigid-flex structure, the rigid sections are formed by adding rigid layers to the outside of the flex structure.

# How Rigid-flex is Supported in Altium Designer

Altium Designer’s PCB editor is a layered design environment, supporting up to 32 signal layers and 16 plane layers. These copper layers are separated by insulation layers. In a traditional rigid PCB these insulating layers are typically fabricated using FR4 and pre-preg, although there is a range of materials available, each with properties that suit different applications. For a traditional rigid PCB these copper and insulating layers exist across the entire PCB, so a single layer stack can be defined for the entire board area.

FR4 is **Flame Retardant**, type 4 woven glass reinforced epoxy laminate - a strong, rigid insulator that retains its high mechanical and electrical insulating properties in both dry and humid conditions, and also has good fabrication properties.

Pre-preg - short for preimpregnated, is a flexible material, typically also containing woven glass, which is supplied to the PCB fabricator partially cured (not completely cooked). It is included between the rigid layers in the layer stack during fabrication, and then heated to perform final curing, after which it becomes rigid, helping to join the layers and form the overall structure of the finished board.
The layer stack for an eight layer rigid circuit, as it was configured in earlier versions of Altium Designer.

A rigid-flex design does not have a consistent set of layers across the entire circuit design, the rigid section of the board will have a different set of layers from the flexible section. And if the rigid-flex design has a number of rigid sections joined by a number of flex sections, then there may be a different set of layers used in each of these sections. A PCB editor with a single layer stack cannot support this design requirement. To support this, Altium Designer's layer stack management system has been enhanced to support the definition of multiple stacks, as shown below.

The Layer Stack Manager supports the definition of any number of layer stacks.

**Multiple Layer Stacks**

*Main article: Defining the Layer Stack*

To support the need to define a different set of layers in different areas of the board design, Altium Designer supports the concept of multiple layer stacks. This is achieved by having an overall master layer stack that defines the total set of layers available to the board designer in this design. From this master layer stack, any number of sub-stacks can be defined, using any of the layers available in the master stack. Each sub-stack is defined and named, ready for use in the rigid-flex design.

**The Board Shape**

*Main Articles: Defining the Board Shape, Defining Board Regions and Bending Lines*

The layer stack defines the board design space in the vertical direction, or Z plane. In Altium Designer, the board space is defined in the X and Y planes by the *Board Shape*. The board shape is a polygonal region of any shape, with straight or curved edges that lie at any angle, which can also include cutouts (internal holes) of any shape. The board shape is a fundamental concept in Altium Designer's PCB editor, it defines the area available for design - where the components and routing can be placed - and all of Altium Designer’s intelligent analysis engines, such as the design rule checker or the autorouter, operate with the boundaries of the board shape.

Note that there is a single, overall board shape for the entire circuit design, including rigid-flex. Within this board shape, any number of board regions can be defined by placing Split Lines to divide the board into separate regions. The image below shows a board shape that has been divided into 3 regions, by the placement of the 2 horizontal blue Split Lines. Use the links above to learn more about splitting a board into multiple regions.
An unusual board shape, note the horizontal dashed blue Split lines, these divide the board into 3 separate regions.

The board shape is often defined by a mechanical designer in an MCAD application. It can be transferred to Altium Designer using one of the industry-standard interchange formats, including DXF or STEP.

**Assigning a Layer Stack to a Region of the Board**

Main Article: [Defining Board Regions and Bending Lines](#)

As mentioned, in a traditional rigid PCB the copper and insulating layers exist across the entire PCB, so a single layer stack can be defined for the entire board shape. For a rigid-flex design made up of a number of rigid and flex regions, where each region needs a different layer stack, an alternative approach is needed. In Altium Designer this is achieved by supporting the ability to assign a layer sub-stack to a specific region of the board shape. To do this, double click on the region to open the Board Region dialog, then select the required **Layer stack** in the dropdown, as shown in the image below.
Double-click on a region to open the Board Region dialog and assign the required layer stack.

**Placing and Managing Flex Bend Lines**

Main article: [Defining Board Regions and Bending Lines](#)

If a region has a layer stack assigned, and that stack has the **Flex** option enabled, then Bending Lines can be placed across that region. Each Bending Line has a: **Radius**, **Bend Angle** and an **Affected Area Width** property, allowing them to be displayed in their folded state, as they would be in a real-world situation.

![Diagram of Bending Lines](image)

2 Bending Lines have been defined, allowing this rigid-flex board to be displayed in its folded state.

**Displaying and Folding a Rigid-Flex Design in 3D**

Altium Designer includes a powerful 3D rendering engine, which allows the presentation of a highly realistic 3 dimensional representation of the loaded circuit board. This engine also supports rigid-flex circuits, and when it is used in combination with the **Fold State** slider allows the designer to examine their rigid-flex design in the flat state, the fully folded state, and anywhere in between.

To switch to the 3D display mode, press the 3 shortcut key (press 2 to return to 2D, or 1 to return to Board Planning Mode). The board will be displayed in 3D, and if the component footprints include 3D Body Objects that define the mounted component, then these will also be displayed. In the image below you can see that the board includes a battery and a battery clip.

![3D Display of Rigid-Flex Board](image)

To apply all of the Bending Lines, slide the **Fold State** slider - it's in the PCB panel when set to Layer Stack Regions mode - as highlighted in the image below. Note that the bends are applied in the
order defined by their sequence number. Bending Lines can share the same sequence number, it simply means that those bends will be folded at the same time when the Fold State slider is used. The board can also be folded/unfolded by running the View » Fold/Unfold command (press the 5 shortcut).

You can only fold a board if one of the rigid sections has the 3D Locked checkbox enabled in the Board Region dialog. Altium Designer needs this to know which section of the board must remain fixed during the folding process.

3D Movie Maker Support for Rigid-Flex Designs

Main article: PCB 3D Video

The ability to fold a rigid-flex design can also be captured as a 3D movie. It is very simple to do and does not require the use of movie key frames during the folding sequence.

Refer to the main article referenced above for a detailed description of how to make a 3D movie. As a basic guide:

1. Switch the PCB editor to 3D mode.
2. Display the PCB 3D Movie Editor panel, and create and name a new Movie Title in the top section of the panel.
3. Create an initial Key Frame, showing the board in its unfolded state.
4. Slide the Fold State slider to show the rigid-flex design in its folded state, then position the folded board as required.
5. Now create a second Key Frame for this view, and set the time. Consider how long you want it
to take to fold the rigid-flex design (the **Duration** setting), typically this would be a few seconds.

6. To check that the video captures the folding process correctly, press the **Play** button.
7. To generate a movie file, add a **PCB 3D Video** Documentation Output in an Output Job file. Remember to configure the video format options in the **Video settings** dialog.
8. Click the Generate Content link the Output Job file to create the movie file.

The video shown below was created using this process, it has the 2 key frames described above, plus 1 additional key frame that was added at the end to hold the final position for a second.

---

**Design Considerations**

Below is a summary of key design areas that must be considered when design a rigid-flex PCB:

- **Conductor routing** - choice of corner style for routes traveling over a flex region is important, avoid sharp corners, use a curve for least stress.
- **Pad shape and area** - use fillets (teardrops), with rabbit ears (anchoring spurs) for single sided flex, the objective is to capture some of the pad shape with the coverlayer.
- **Through holes** - attempt to avoid through holes in the bend area, particularly in a dynamic application.
- **Coverlayer** - avoid stress risers (exposing the incoming track), reduce opening in coverlayer 250um.
- **Planes** - crosshatched if possible.
- **Staggered lengths** - to avoid bookbinding (layer buckling when flexed), stagger the layer lengths by approx 1.5 times the layer's thickness.
- **Service loop** - make the flex region slightly longer to help with assembly/disassembly, and to allow for product dimensional variations (the extra length is referred to as the service loop).
- **Conserve copper** - consider how the flex circuit will be panelized, it might be better to adjust the design to ensure best material usage.
- **Panelization** - orient the flex regions to suit grain of material (bend along the grain).
- **Tear resistance** - curved corners, drilled hole at corner, hole in slit, leave metal in corners.
- **Routing** - stagger the routes on 2-layer boards to avoid I beaming and widen the routes through the bending zone (especially important for permanent bends).
- **Static Bend Ratio** - the ratio of the bend radius to the circuit thickness. Ideally, multi-layer
circuits should have a bend ratio of at least 15:1. For double-sided circuits, the minimum ratio should be at least 10:1. For single-layer circuits, the minimum ratio should also be at least 5:1. For a dynamic application, aim for a bend ratio of 20-40:1.

- Rolled annealed copper is more ductile, plated copper is not the best choice for flexible regions.

### Documentation and Drawing Requirements

Typical suggested documentation requirements, include:

1. The Flex PCB shall be fabricated to IPC-6013, class (your requirement here) standards.
2. The Flex PCB shall be constructed to meet a minimum flammability rating of V-0 (if required).
3. The Flex PCB shall be RoHS compliant (if required)
4. The rigid material shall be GFN per IPC-4101/24 (if using epoxy material)
5. The rigid material shall be GIN per IPC-4101/40 (if using polyimide material)
6. The flexible copper clad material shall be IPC 4204/11 (flexible adhesiveless copper clad dielectric material)
7. The covercoat material shall be per IPC 4203/1.
8. The maximum board thickness shall not exceed (your requirement here) and applies after all lamination and plating processes. This is measured over finished plated surfaces.
9. The thickness of acrylic adhesive through the rigid portion of the panel shall not exceed 10% of the overall construction. See comments on this above.
10. Pouch material can be used for ease of manufacturing and must be removed from the flexible portion of the board prior to shipping.
11. The flexible section thickness shall be (your requirement here, do not add this note if this thickness is not critical).
12. Minimum copper wall thickness of plated through holes to be (your requirement here) {.001” average is recommended} with a minimum annular ring of (your requirement here). {.002 is recommended}
13. Apply green LPI soldermask (if required) over bare copper on both sides, in the rigid sections only, of the board. All exposed metal will be (specify your surface finish requirement here).
14. Silkscreen both sides of the board (if required) using white or yellow (most common) non-conductive epoxy ink.
15. Marking and identification requirements.
16. Electrical test requirements.
17. Packaging and shipping requirements.
18. Impedance requirements.

### Additional Drawing Detail

1. A drill table detailing finished hole size, associated tolerances and plated/not plated.
2. A dimensional drawing, including reference datum(s), critical dimensions, rigid to flex interfaces, bend location and direction markers.
3. Panelization detail, if required.
4. Construction and Layer detail, detailing material used for each layer, thicknesses and copper weights.

### References

- Flex Circuit Design Guide - Epec Engineering Technologies
- Flexible Circuit Technology - Joe Fjelstad