PCB DESIGNER'S GUIDE TO FLEX AND RIGID-FLEX

EMA Design Automation®

PCB DESIGNER'S GUIDE TO FLEX AND RIGID-FLEX

Flexible PCBs are becoming increasingly common to adhere with the mechanical requirements of ever shrinking product design. With this, comes additional design requirements that must be adhered to for successful PCB fabrication and assembly including:

- Materials
- Stackup
- Design Rules
- Routing and Pours
- Documentation

This eBook will discuss what needs to be considered when designing flex and rigid-flex PCBs and gives designers the understanding and techniques they need to create a successful product.

Table of Contents

02 Introduction to Flex and Rigid-Flex Designs

04 Specialized Materials

07 Flex
Types of Flex Boards

09 Flex with Stiffener
Types of Flex Boards

11 Rigid-Flex
Types of Flex Boards

13 Combinational
Types of Flex Boards

15 Mechanical Requirements
Key Considerations

17 Component Requirements
Key Considerations

19 Routing Best Practices
Key Considerations

21 Design Rules
Key Considerations

22 Fabrication
Key Considerations

27 Navigating Flex Design Challenges

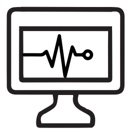
29 Overcoming Flex Design Challenges with OrCAD X

INTRODUCTION TO FLEX AND RIGID-FLEX PCBS

WHERE CIRCUITS GET FLEXIBLE AND INNOVATION TAKES SHAPE

Flex PCBs are designed to bend, twist, or fold while maintaining electrical connectivity. By incorporating thin, flexible materials, flex PCBs are ideal for compact and light-weight applications. Rigid-flex designs combine both rigid and flexible elements into a single board, offering mechanical flexibility and complex 3D interconnect solutions. These designs are widely used in applications where space savings, weight reduction, and durability are important including:

Wearables



Medical Devices

Automotive

Aerospace

While flex or rigid-flex designs can have vast benefits for product development, they also present additional complexities. The pros and cons of rigid flex designs present trade-off decisions which designers must carefully weigh:

Pros	VS	Cons
Increased Versatility		Increased Fabrication Costs
Light Weight		Specialized Materials
Increased Durability and Reliability		Increased Design and Manufacturing Time
Decreased Space Consumption		Higher Design Complexity
Enhanced Electrical Performance		Possibility of Mechanical Stress and Fatigue
Simplified Product Assembly		Limited Ability to Repair
Optimized Power Efficiency		Limited Electrical Performance

In addition to comparing and contrasting the benefits and challenges of rigid-flex designs, engineers should understand how these can affect the PCB design process. Detailed metrics should be analyzed in reference to the scope of the project to ensure teams are selecting the PCB type that will best meet their project requirements and specifications.

INTRODUCTION TO FLEX AND RIGID-FLEX PCBS

DECIDING WHEN TO FLEX

The decision to design a flex or rigid-flex PCB is one that should not be taken lightly. When contemplating this choice, there are some key questions that designers must ask themselves. The following questions will help to determine if your project requirements will be met with a flex or rigid-flex design:

1 Does the design budget allow for a higher cost fabrication?

Flex and rigid-flex designs require a custom fabrication process and specialized materials. This can increase manufacturing costs up to **5 to 10 times greater** than typical rigid boards. The cost trade-off should be weighed between increased fabrication costs and the potential decreased cost for components and cables.

2 Does the design timeline allow for longer fabrication?

While some rigid boards can be fabricated in just a few days, most flex and rigid-flex designs often take **2 to 3 weeks** to produce. Be sure to consider if the project schedule and intended product launch date can accommodate this longer lead time before deciding on a flex design configuration.

3 What development stage is the design in?

Consider what stage of the development process you are in. If you are in the prototype stage, the cost and time savings of designing a rigid board may be worth it- even if you respin the design to be a rigid-flex board in a future development cycle.

Example:
If you are in the development stage and testing circuit functionality for two rigid boards connected with flex it may be better to keep the boards separate and connected with a cable to allow for individual board respins instead of manufacturing the full rigid-flex board each time a modification is required. This will cut down on both the time and cost of fabrication.

4 Can flex meet the electrical needs of the PCB?

When dealing with high-power or sensitive signals, flex boards may limit the ability to meet your design requirements.

- Flex designs typically use ½ oz or 1/3 oz copper weight. If your design has high current, a flex board may not be able to meet the required electrical specifications for your design.
- Materials used in rigid-flex designs will vary between flex and rigid zones. If your design contains sensitive signals which will be impacted by material changes this may affect signal quality and integrity.

Once the decision has been made to incorporate flex elements into the PCB design, the type of flex board required as well as the ideal materials and stackup must be determined.

INTRODUCTION TO FLEX AND RIGID-FLEX PCBS
SPECIALIZED MATERIALS

Material selection is critical to the operation, cost, and success of your flex designs. To determine if a flex material will meet your design requirements, always consult your flex fabricator to understand:

- In-Stock and Available Materials
- Material Lead Time
- Material Thickness
- Design Properties
- Temperature Rating
- Cost
- Flexibility Properties

Each part of a flex stackup has multiple material options based on the needs of the design. Materials should be carefully considered for the dielectric, soldermask, and stiffener used in the PCB stackup.

1 Dielectrics

Dielectric materials are used as insulating layers between conductive copper layers. These materials help control electrical performance by influencing signal integrity, impedance, and overall PCB reliability. The choice of dielectric material affects the PCB's ability to manage high-frequency signals, power distribution, and thermal performance. The following are common dielectric materials for flex designs:after a product is launched a formal Engineering Change Order (ECO) must be issued to manage revisions, improvements, corrections, or updates to the existing product. With new board spins easily costing 10's of thousands of dollars, both unexpected redesigns and ECOs result in additional design hours and expenses that were not accounted for in the initial proposal.

Polyimide VS Polyester

Polyimide

Polyimide is a high-performance polymer known for its thermal stability, mechanical strength, and electrical insulation properties. The most common polyimide is Kapton but other options like Isola are available. This type of dielectric is commonly used in flex designs providing a balance between flexibility and tensile strength. Thickness can range for the polyimide core between a 1/2 inch and 3 mils.

Pros

- Various Thicknesses
- High-Temperature Applications
- Flexible But Durable

Cons

- More Expensive

Polyester

Polyester (PET) is a cheaper dielectric material used in the fabrication of flex designs. This material is typically used for consumer electronics and low-cost applications. Polyester can have limitations for the design application and should not be used for high-end, high-temperature performance PCBs as it has a low temperature resistance (up to 150° C) and is susceptible to moisture.

Pros

- Cheaper

Cons

- Temperature Limitations
- Susceptible to Moisture

INTRODUCTION TO FLEX AND RIGID-FLEX PCBS

SPECIALIZED MATERIALS

2

Soldermask

In rigid PCB design, soldermask or solder resist is used as a protective polymer coating applied on the topmost insulating layer over the copper traces of a PCB, except for the areas where components are soldered. It prevents solder bridging, oxidation, and contamination, ensuring the board functions properly and lasts longer. For flex PCBs, traditional soldermask is not suitable as it becomes rigid, and brittle once cured. Since flex PCBs bend, twist, and fold, using standard soldermask would lead to cracking, delamination, and mechanical failure. Instead, the following materials are available for use in flex design stackups:

Coverlay and Adhesive

VS

A combination of coverlay and adhesive is most commonly used in flex design. During fabrication this is added as a film on top of the flex region, typically 1 mil coverlay and 1 mil adhesive. This can create design challenges; since this is applied as a film not a liquid, the coverlay sliver that you are able to achieve is much higher than with a soldermask, approximately 10 mils instead of 4 mils with a typical soldermask application. Coverlay and Adhesive application enables the use of a stiffener if required for your PCB design.

Pros

- ✓ Minimal Thickness
- ✓ Can Use Stiffener

Cons

- ✗ Increased Soldermask Sliver

Flexible Soldermask

Flexible soldermask is similar to the standard soldermask used in rigid designs. This material is often used when needing to mount more components on a flexible circuit because smaller slivers can be achieved. Flexible soldermask also comes with limitations to the design. Stiffeners cannot be used in conjunction with this material without other materials or additional assembly processes required. Additionally, this material cannot support shielding materials typically used for EMI shielding.

Pros

- ✓ Decreased Soldermask Sliver

Cons

- ✗ No Stiffener Use
- ✗ No EMI Shielding

INTRODUCTION TO FLEX AND RIGID-FLEX PCBS
SPECIALIZED MATERIALS

3

Stiffener

A stiffener is a rigid material which is attached to specific areas of a flex PCB to provide mechanical support, reinforcement, and stability. Stiffeners are typically used for structural enhancement, such as reinforcing components and connectors or increasing the mechanical strength for mounting. The following materials are commonly used as a stiffener in flex PCB stackup design:

FR-4

VS

FR-4 is the cheapest and most readily available material for stiffener as it is commonly stocked as a standard material used in rigid designs. FR-4 is available in various thicknesses, allowing you to conveniently achieve your stackup requirements but provides no flexibility.

Pros

- Inexpensive
- Readily Available
- Various Thicknesses

Cons

- Provides No Flexibility

Polyimide

VS

Polyimide (PI) materials can also be used as a stiffener. These materials are typically more expensive- in fact the most expensive material presented here. This material is not typically stocked and will have longer lead time associated with it; however, it has the lowest conductivity and minimal thickness of the stiffener materials.

Pros

- Low Conductivity
- Minimal Thickness
- Some Flexibility

Cons

- Most Expensive
- Longer Lead Time

Metal

Metal can be used as a stiffener in flex designs and include materials such as aluminum and stainless steel. These materials have high conductivity. While this can be beneficial in scenarios where performing heat sinking is desired, it may be detrimental for typical circuit operation. Using metal material does have benefits to the stackup design. Metals are typically available in thinner thicknesses allowing you to achieve the optimal balance between high rigidity and design thickness.

Pros

- Minimal Thickness
- High Rigidity

Cons

- High Conductivity
- Increased Temperatures

Now that you have achieved an understanding of the varying material choices that are involved in flex designs, it is important to understand how these materials come together to create the different types of flex boards and stackups you can design.

TYPES OF FLEX BOARDS

FLEX

Flex PCBs are thin, bendable circuit boards designed to fit into compact, dynamic, or irregularly shaped electronic devices. Unlike rigid PCBs, which use FR-4 material, flex PCBs use flexible materials, allowing them to bend, twist, and fold without breaking. Flex boards are typically used when no components need to be installed.

Flex PCB at a Glance

Use Case:
Designs with no components

Average Manufacturing Time:

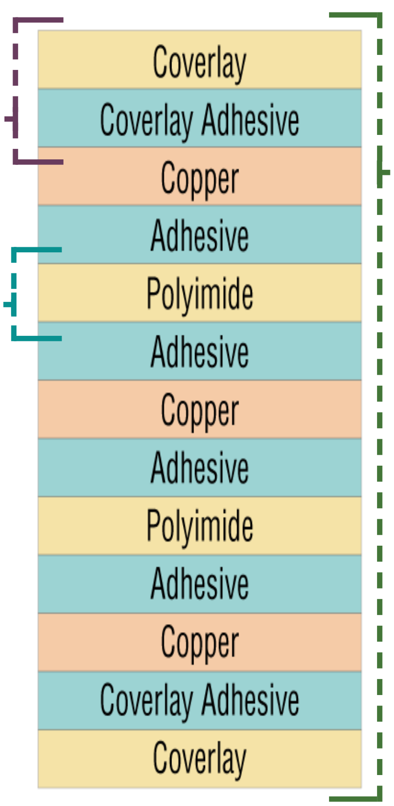
Average Cost:
$ $

Flex Stackup Configuration

Flexible stackups differ from rigid stackups due to the materials used and the fabrication process. Below is a sample 3-layer flex stackup detailing common differences between flex and rigid designs.

Coverlay and Adhesive

Coverlay and adhesive are used in place of soldermask. Flexible soldermask is an option but has limitations that must be weighed.



Polyimide Materials

Polyimide or similar materials are used instead of FR-4 or prepreg to provide flexibility.

Layer Count

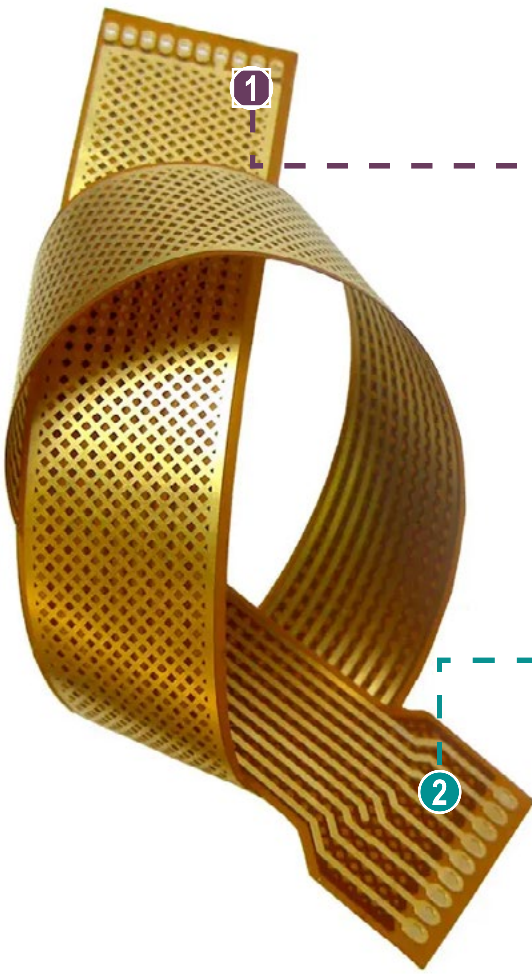
Unlike rigid boards, flex boards can have an odd number of layers. During the fabrication process it is much easier to add copper layers one at a time, resulting in ability to create stackups with 3 and 5 layers.

TYPES OF FLEX BOARDS

FLEX: DESIGN CONSIDERATIONS

When deciding to design a flex or rigid-flex PCB, there are certain design choices that must be considered to foster flexibility. The stackup must be adjusted to contain more flexible materials such as coverlay, adhesive, and polyimides which allow the flex board to bend.

To encourage flexibility, remember to keep the copper weight lower. For optimal flexibility, 1/3 oz or ½ oz copper weight is widely used. While 1oz and 2oz copper can be used, this limits flexibility- which is the advantage of designing a flex PCB.



1 Specialized Flex Materials

Leverage materials that are specialized for flex designs such as:

- Coverlay
- Adhesive
- Polyimide

2 Reduced Copper Weight

Minimize the copper weight used to foster flexibility such as:

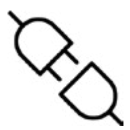
- 1/2 oz
- 1/3 oz

This basic flex stackup and design can be expanded upon, incorporating additional materials and design elements, to meet the complexity requirements for your PCB. Understanding the basics of a flex stackup, materials, and design considerations will create a solid foundation when additional complexities are introduced.

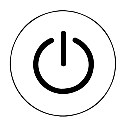
Image Source: Multi-cb

TYPES OF FLEX BOARDS
FLEX WITH STIFFENER

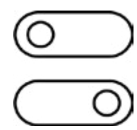
A flex board with stiffener is a fully flexible board with some stiffened or supported regions. This configuration is ideal if you have a small number of components needed in the design as these stiffened regions can provide stability and reinforcement, allowing the installation of:



Connectors



Buttons



Switches

Using a flex with stiffener stackup configuration is also applicable for a Zero Insertion Force (ZIF) connector. A ZIF connector enables easy insertion and removal of a flexible flat cable (FFC) or ribbon cable without requiring significant force. These connectors are commonly used in applications where delicate or thin conductors need to be connected, such as in laptop keyboards, LCD displays, and other compact electronic devices.

Flex with Stiffener PCB At a Glance

Use Cases:
Designs with some components
ZIF Connectors

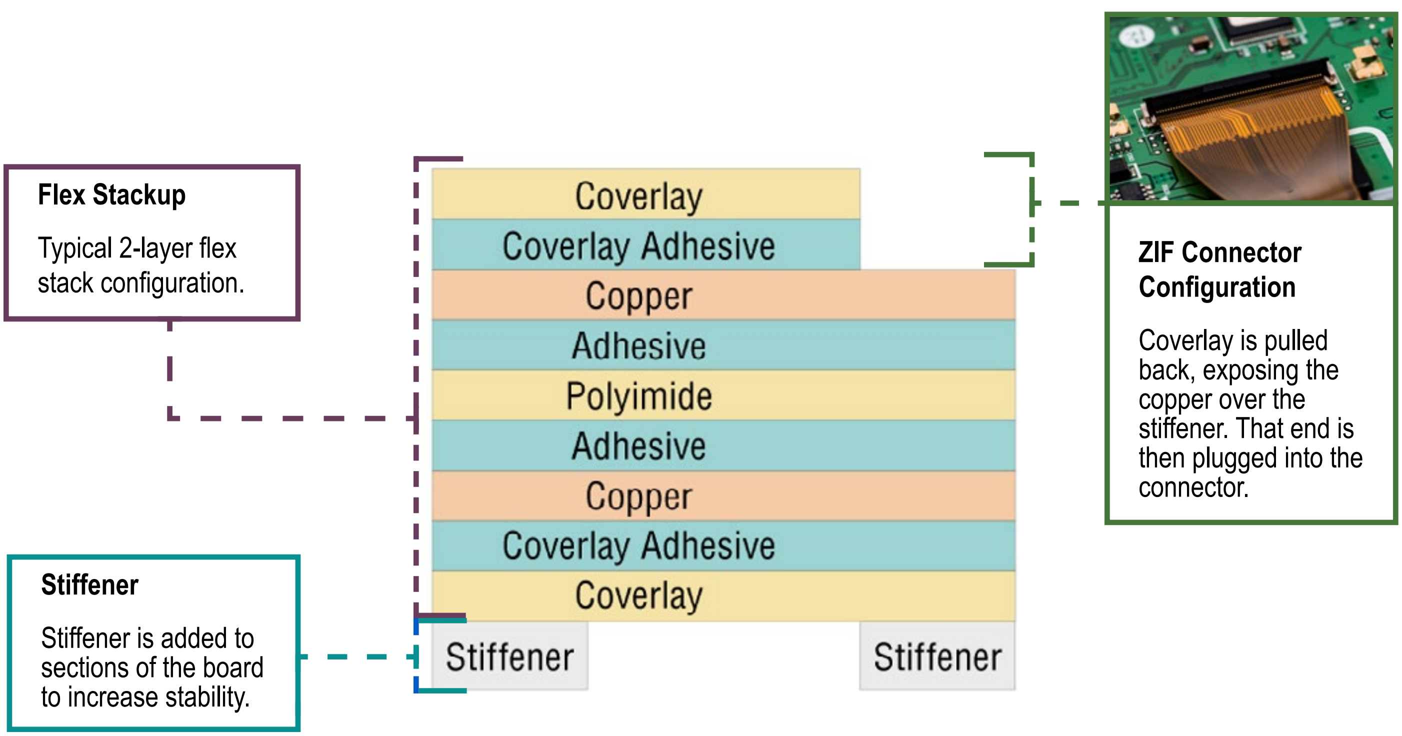
Average Manufacturing Time:

Average Cost:
$ $

> Configuring a stackup for a ZIF connector requires additional attention as there are specific rules based on the copper pattern, thickness, and size of the stiffener that need to be used to successfully plug it into the connector.

Flex with Stiffener Stackup Configuration

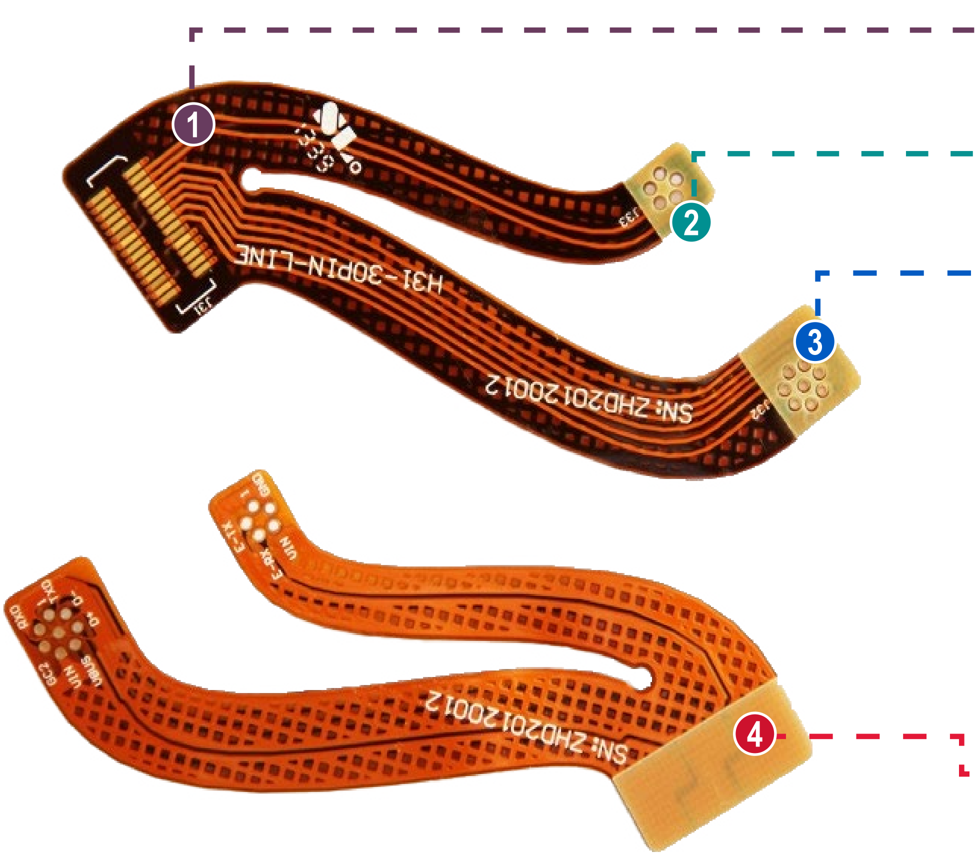
Flex with stiffener stackups expand upon the flex stackup configuration. Below is a sample 2-layer flex with stiffener stackup detailing common differences between flex and flex with stiffener.



TYPES OF FLEX BOARDS

FLEX WITH STIFFENER: DESIGN CONSIDERATIONS

When designing a flex board and incorporating stiffener into the stackup, special consideration needs to be given to the placement of the stiffener. To determine the appropriate location of the stiffener in the stackup design, consider where the components will be placed on the PCB and what type of component (surface mount or through-hole) will be used. Stiffener placement will vary between surface mount and through-hole components to accommodate for soldering techniques during the assembly process.



1 **Flex Stackup**

2 **Stiffener**

3 **Stiffener Installation: Through-Hole**

Stiffener installed on the same side of the board as the component for through-hole devices.

4 **Stiffener Installation: Surface Mount**

Stiffener installed on the reverse side of the board for surface mount devices.

Image Source: GT Smart Technology

TYPES OF FLEX BOARDS
RIGID-FLEX

A rigid-flex board is a combination of full rigid boards and flexible elements. Typically, this configuration is selected when a designer is replacing a cable between two rigid PCBs. When replacing a cable with a flex section, designers eliminate connectors, cables, and assembly steps. While a rigid-flex design is more expensive based on the fabrication and lead time, transitioning to this configuration can help designers:

Increased Durability

Meet Mechanical Requirements

Simplify Assembly

Transitioning to a rigid-flex configuration will increase the complexity of the design- requiring additional attention to detail for the stackup configuration and PCB layout.

Rigid-Flex PCB at a Glance

Use Case:
Replacing a cable between two rigid PCBs

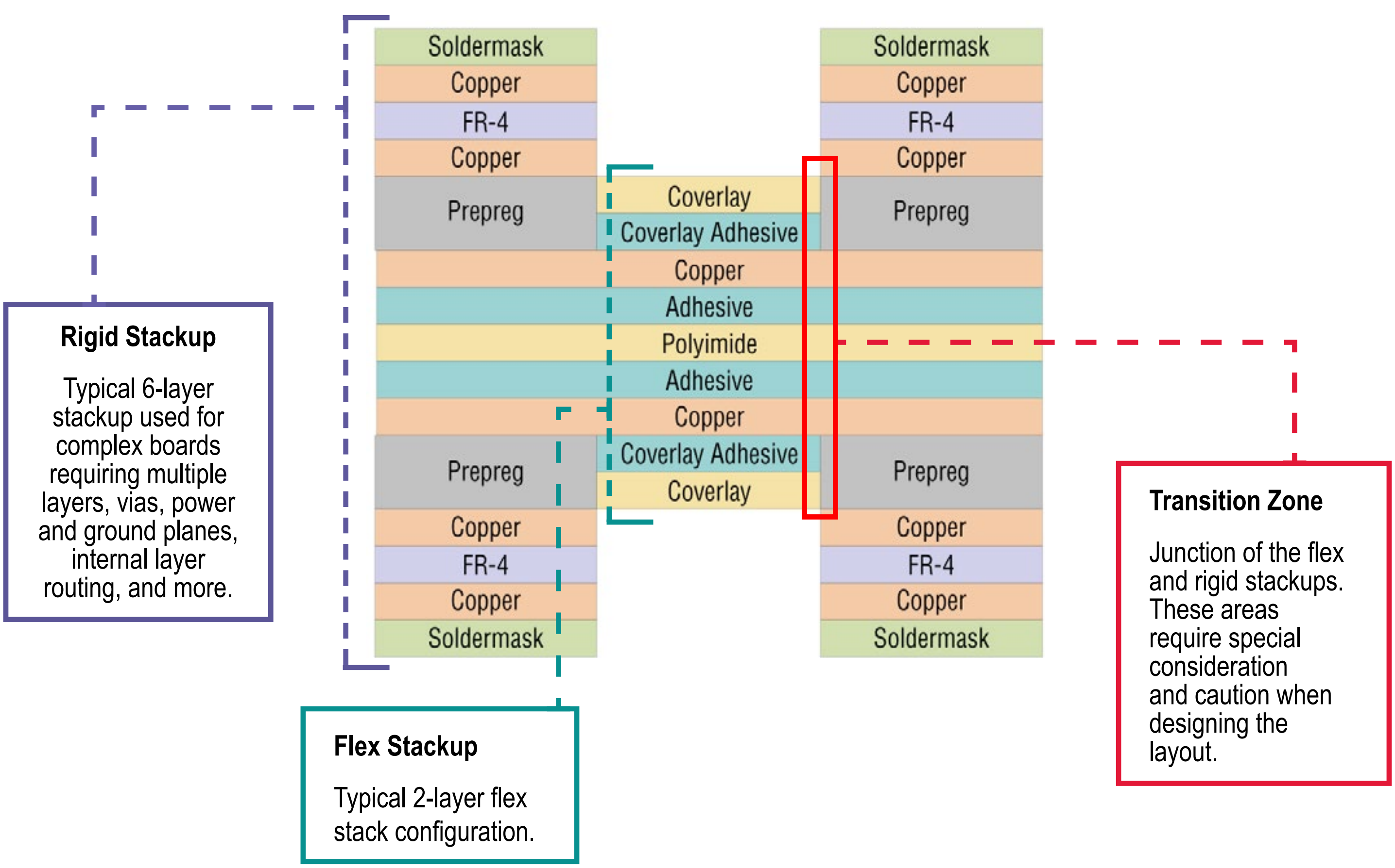
Average Manufacturing Time:

Average Cost:

$ $ $

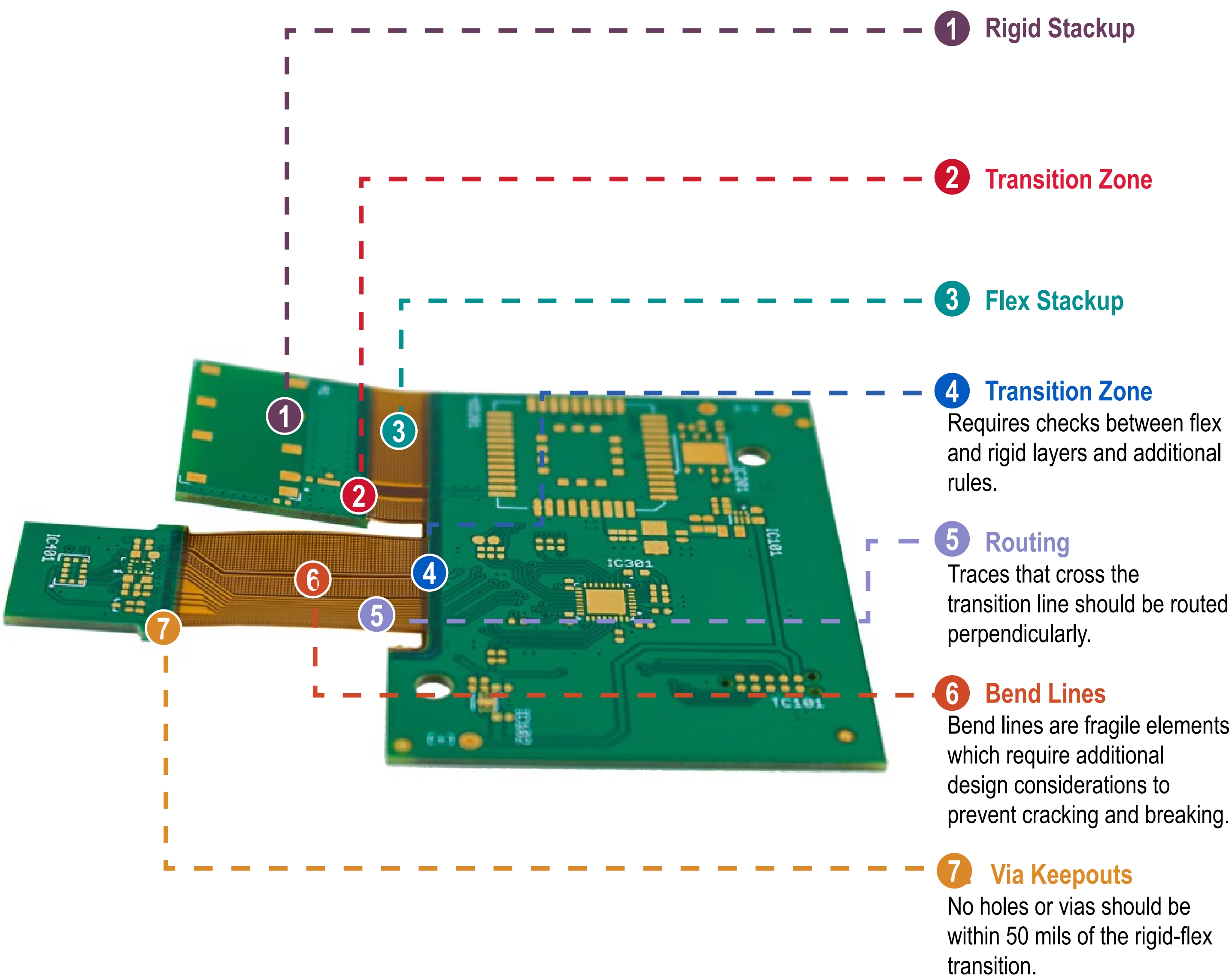
Rigid-Flex Stackup Configuration

Rigid-flex stackups contain two separate stackups put together and combine the typical stackup of a rigid board with flexible elements. Below is a sample rigid-flex stackup detailing key elements of the cross section.



TYPES OF FLEX BOARDS
RIGID-FLEX: DESIGN CONSIDERATIONS

Multi-layer, rigid flex stackups which consist of two separate stackups put together require careful consideration during the design process. These designs contain junction points, referred to as transition zones, where the flex and rigid materials in the stackup meet. It is important to consult your manufacturer to understand how the materials work together and incorporate design rules to ensure reliability, durability, and proper operation. The diagram below outlines a rigid-flex design, and some of the design considerations required when working with transition and bend zones.



Understanding the increased design complexity for rigid-flex PCBs up-front will help designers properly manage the required stackups, zones, and rules throughout the design process to increase the likelihood of success. Each of these previously discussed stackups can be combined into a single PCB, referred to as a combinational or hybrid PCB, which integrates multiple types of PCB structures and stackups into a single design.

Image Source: EPEC

TYPES OF FLEX BOARDS
COMBINATIONAL

A combinational PCB is a combination of all the different types of flex designs previously discussed. This configuration can contain any arrangement of previously discussed stackups including:

- Rigid
- Flex
- Flex with stiffener
- Different stiffener thickness
- Stiffener on the top or bottom
- Multiple arms

With the ability to contain three or more unique stackups, combinational boards are often used for complex designs; however, careful consideration needs to be given to the stackup design, management, and communication to ensure success.

Combinational PCB At a Glance

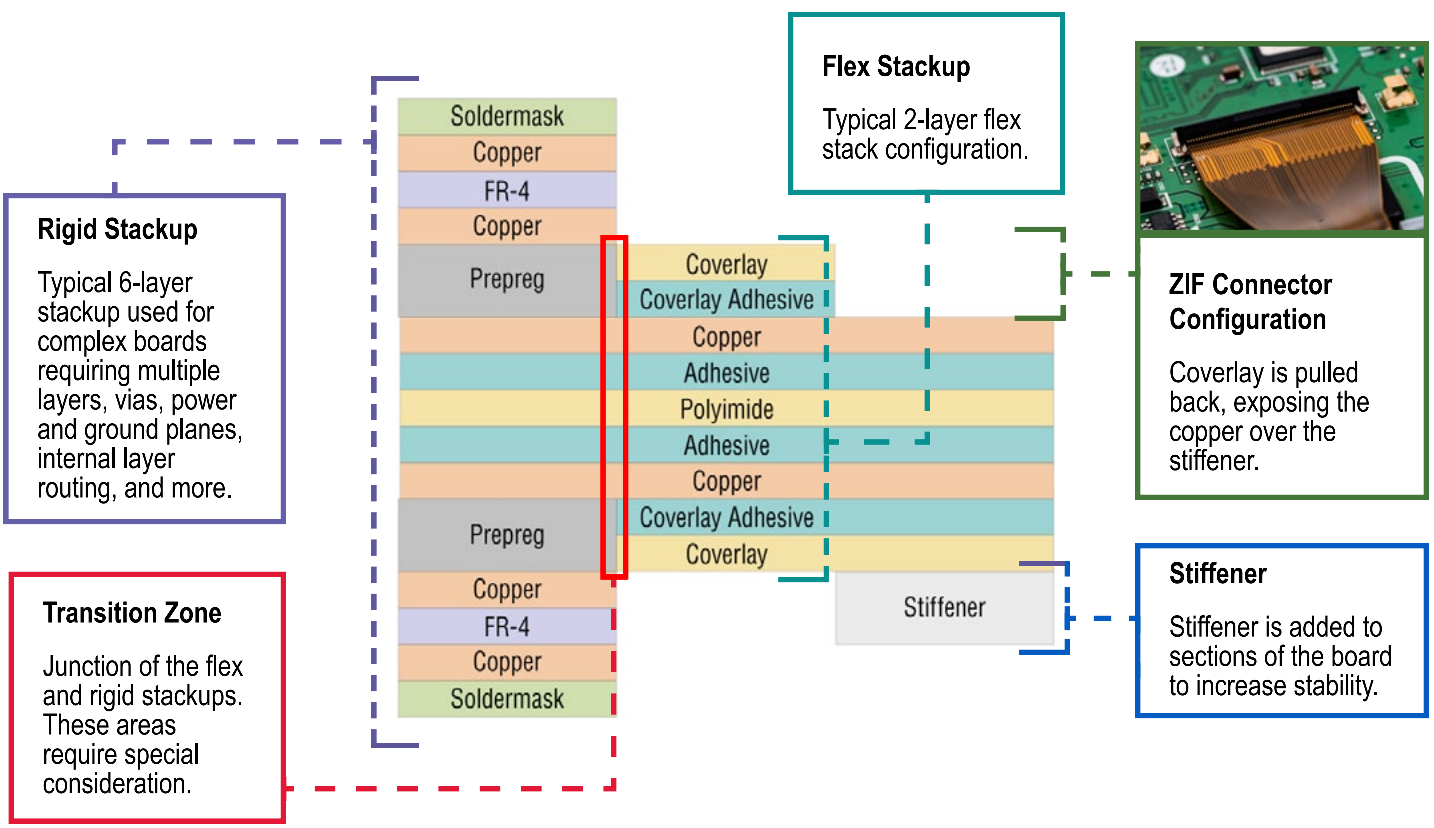
Use Case:
Complex Designs

Average Manufacturing Time:

Average Cost:
$ $ $ $

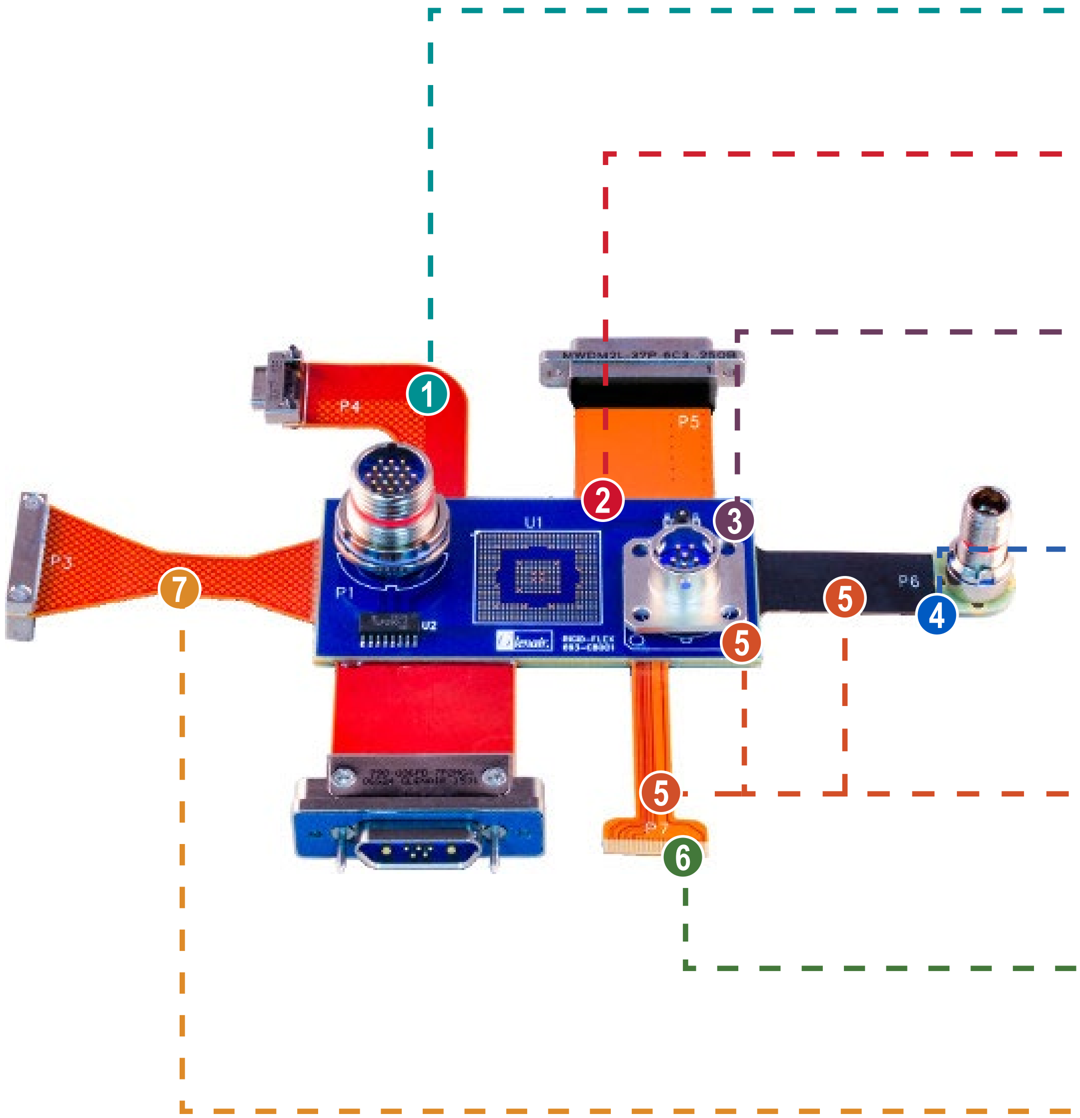
Combinational Stackup Configuration

Combinational stackups contain all or a combination of the stackups previously discussed. Below is a sample combinational stackup detailing key elements of the cross section.



TYPES OF FLEX BOARDS
RIGID-FLEX: DESIGN CONSIDERATIONS

Combinational designs have increased complexity and therefore the most amount of design considerations due to the number of stackups typically included in the PCB. In each of these stackups, fabrication processes and capabilities will vary, resulting in specific rules that must be assigned and managed for each section or zone. To ensure manufacturing success, the multiple stackups and zones defined in combinational PCBs must be properly managed and clearly documented.



1 Flex Stackup
Flex stackup and rules based on manufacturing capabilities.

2 Transition Zones
Transition zones require checks between flex and rigid layers.

3 Rigid Stackup
Rigid stackup and rules based on manufacturing capabilities.

4 Stiffener Installation
Place stiffener according to component types and locations.

5 Multiple Stackups
Multiple stackups and zones should be defined and managed within CAD tools.

6 ZIF Connectors
Consult datasheets for ZIF Connector configuration.

7 Bend Lines
Bend lines are fragile elements which require additional design considerations to prevent cracking and breaking.

It is critical to understand these difficulties and areas of concern upfront to ask the right questions during the development process, define appropriate design rules, minimize manufacturing errors or delays and increase the success of your rigid-flex design. Let's explore these key design considerations in greater detail.

Image Source: Glenair

KEY CONSIDERATIONS FOR FLEX DESIGNS
MECHANICAL REQUIREMENTS

For rigid-flex designs, there are increased mechanical requirements compared to standard rigid PCBs. It is important to collaborate with the mechanical team up-front to understand these specifications and their effect on the PCB layout. ECAD and MCAD teams should discuss the following questions before beginning the PCB:

1

Are the rigid and flex elements fixed, or can they be adjusted?

When transitioning to a rigid-flex board from a rigid configuration, it gives the designer ability to modify and improve the design. Often removing connectors and cables provides additional flexibility. This allows you to optimize the design, reduce rigid board size or adjust the flex size and/or width to develop the creative solutions necessary to meet your design requirements.

2

Where are the bend lines for the flex?

Bend lines can impact placement and routing. The bend lines as well as the corresponding via and placement keepouts should be incorporated into your CAD software from the beginning of the design. To determine the bend lines, work with the mechanical team to obtain knowledge of the final assembly.

3

Is the size and shape of the stiffener clear?

For proper fabrication, the stiffener needs to touch at least two edges of the flex. Additionally, it is important to define and verify that stiffener exists in all flex areas that will have components installed.

4

How is the flex board going to be installed?

There are two main methods for the flex board to be installed- dynamic and static. In dynamic flex, the flexible portion of the design is bending multiple times. In static flex, the flexible portion of the design is bending once and being locked into place and doesn't move in the future. Boards that will have a dynamic flex installation should consider creating a more robust flexible design to minimize potential breakage.

5

What is the required bend radius?

The bend radius is dependent on the layers of the flex stackup. Determine how many layers are required to carry the signals and current in the design and if the layer count is going to adhere to the bend radius requirements. Work with your mechanical team to determine the appropriate bend radius and consult industry standards such as IPC-2223 for bend radius guidelines. As a rule of thumb, the bend radius for 1-to-2-layer designs is approximately 6 times the flex width and the bend radius for boards with three or more layers is 12 times the flex width or more.

To understand the importance of collaboration between ECAD and MCAD teams, let's review a real-world example in which teams must work together to develop creative solutions for critical design choices.

KEY CONSIDERATIONS FOR FLEX DESIGNS

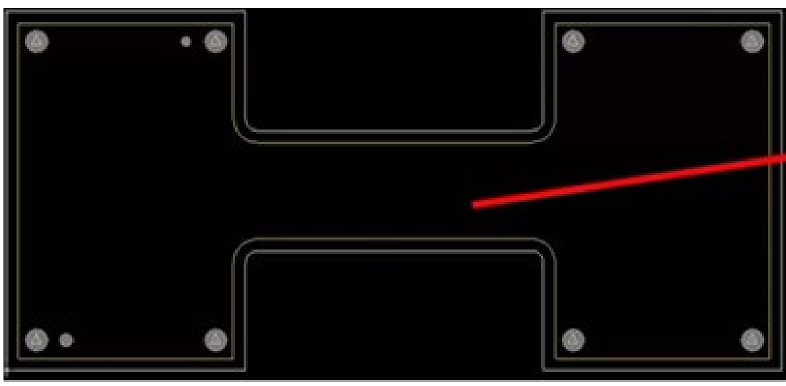
MECHANICAL REQUIREMENTS: REAL-WORLD EXAMPLE

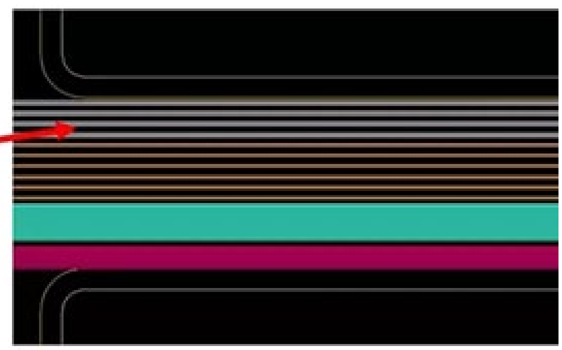
Design Scenario

When transitioning to a rigid-flex board from a rigid configuration, it gives the designer ability to modify and improve the design. Often removing connectors and cables provides additional flexibility. This allows you to optimize the design, reduce rigid board size or adjust the flex size and/or width to develop the creative solutions necessary to meet your design requirements.

Problem

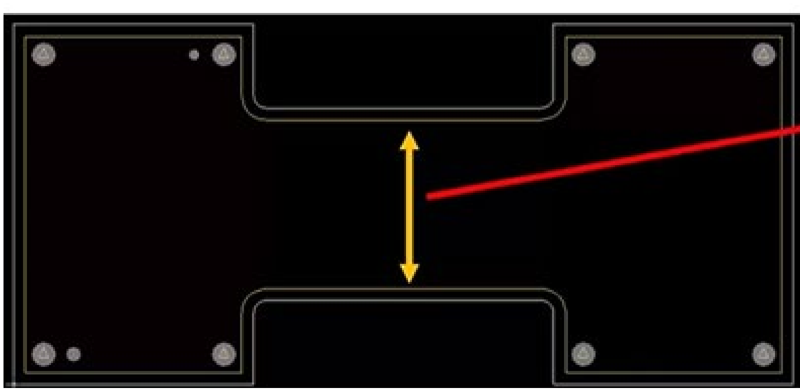
The flex stackup was already utilized to capacity and adding layers to the flex stackup to accommodate the additional signals would compromise the bend radius. For this design, modifying the bend radius was non-negotiable as it was a critical element for mechanical durability and final assembly success.

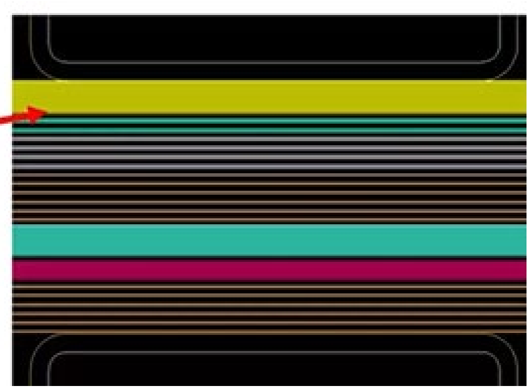




Solution

The mechanical team was consulted, and it was determined that the flex width was not a critical aspect in the design assembly. Therefore, the width of the flexible portion could be increased to accommodate the additional signals without affecting the bend radius.

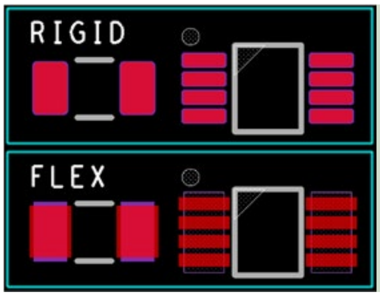




Determining the mechanical specifications at the beginning of the PCB layout will provide the fundamental knowledge (board size, flex width and size, bend lines, bend radius, etc.) required to set your rigid-flex design up for success. With this information defined, the PCB layout process can begin with PCB footprint creation.

KEY CONSIDERATIONS FOR FLEX DESIGNS
COMPONENT REQUIREMENTS: PCB FOOTPRINT MODIFICATIONS

When placing components on the rigid section of a rigid-flex design, footprints can remain the same; however, if components are required on a flex region, the PCB footprint needs to be adjusted to make assembly easier and increase reliability of the end product.

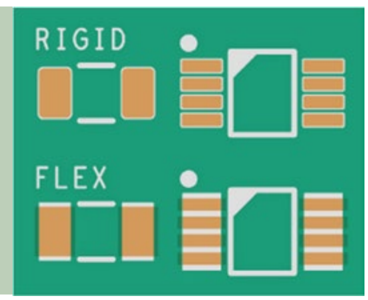


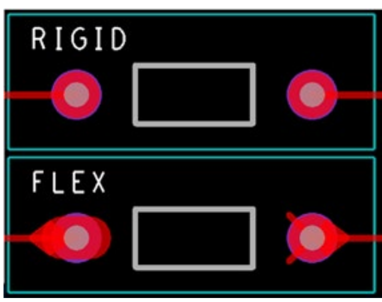
Surface Mount (SMD) Pads

Due to the flexibility of these types of designs, special consideration needs to be taken to ensure pads and footprints stay connected to the PCB with repetitive bending. Anchor pads by having the coverlay extend of the pads on at least two sides. This prevents the component from peeling off the flex.

Coverlay Sliver

With soldermask on a rigid board, you can typically achieve a 3-4mil sliver. For flex boards, based on the materials used, a 10-mil sliver is required for the coverlay. This is difficult to achieve due to the pitch of many SMD footprints; therefore, gang relief is commonly used on IC packages. Gang Relief is a technique used to create relief areas around multiple pads simultaneously, rather than individually. This method helps in improving manufacturability and reliability.





Through-Hole (TH) Pads

Incorporate at least a 30-mil annular ring oversize with 15 mils on each side of the through-hole. This is a good rule of thumb but may not always be possible with the space in the design but the larger the annular ring oversize the better. Anchor the pads with anchor tabs, adjusted pad shapes, and/or teardrops to prevent any peeling of pads off the board during assembly.

ZIF Connector Definition

ZIF connectors are typically placed at the edge of a flex design with a stiffener underneath and will be plugged into a connector on a mating board. Coverlay should be mostly removed but should anchor the end of the pads. Stiffener thickness is critical for installation. Copy copper, coverlay, stiffener, and thickness exactly from the datasheet so that the part will install correctly.

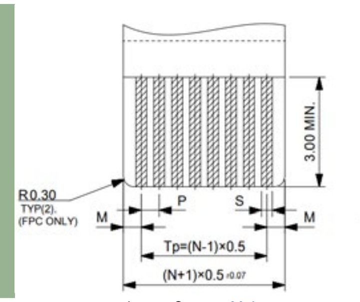


Image Source: Molex

With PCB footprint modifications complete to accommodate for the flexibility of this type of design, the PCB development process can continue onto component placement.

KEY CONSIDERATIONS FOR FLEX DESIGNS
COMPONENT REQUIREMENTS: PLACEMENT

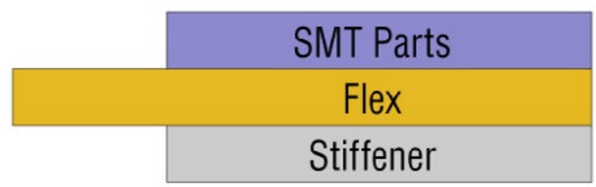
Similar to the PCB footprint modifications, if you are working in a rigid area, placement is the same as always; however, if you are working in a flex area, special consideration needs to be given to component placement. When placing components in a flex design, the most important rule is to avoid placing parts (as well as vias) on or near a bend line. This should ***never*** occur as bend lines are very vulnerable areas of the flex design and placing components or vias in these locations can have a detrimental effect on performance and reliability. In addition, special attention must be given to component placement in relation to the stiffener location in the stackup.

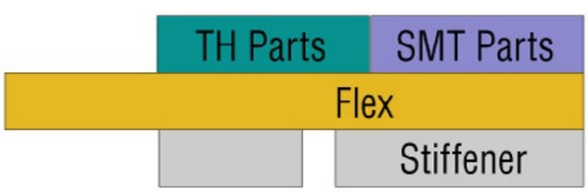
Through-Hole Parts

Stiffener should be incorporated on the ***same side*** of the board as the component to allow for soldering on the bottom side of the PCB. Soldering and the assembly process becomes more difficult if the stiffener is placed on the opposite side of the component, blocking access to the pads.

Surface Mount Parts

Stiffener should be incorporated on the ***opposite side*** of the board as the components. There are no aspects of the surface mount component that need to be accessed on the bottom side of the board for assembly or testing.





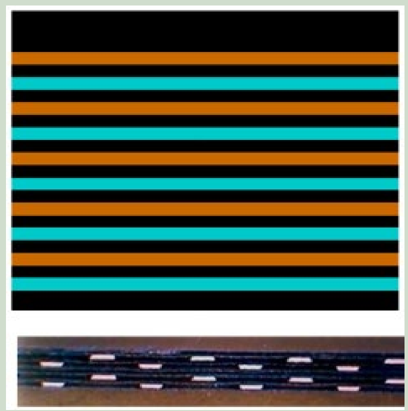
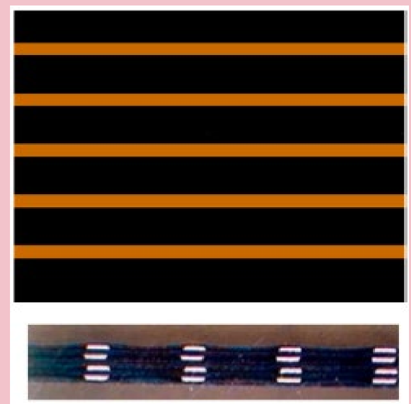
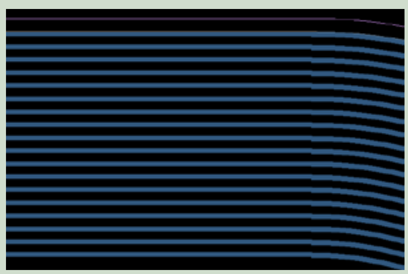
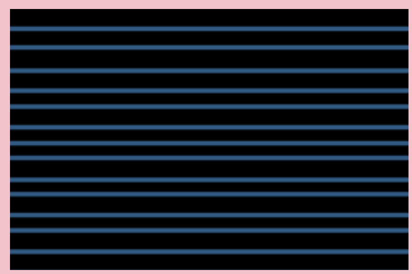
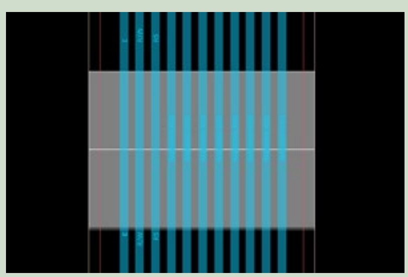
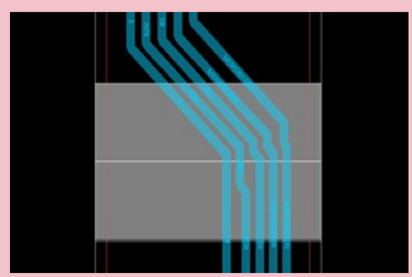
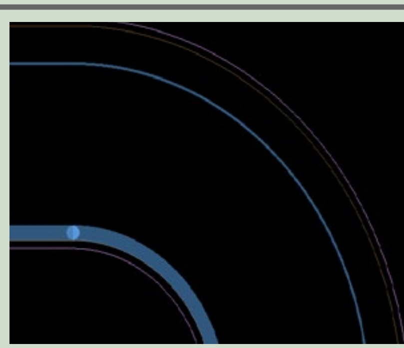
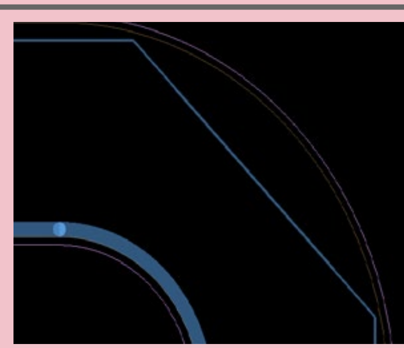
Both Through-Hole and Surface Mount Parts

Placement of the stiffener becomes more difficult when there is a combination of both surface mount and through-hole components. This scenario is possible but requires extra care in the stackup design and PCB documentation. To successfully place both through-hole and surface mount parts on flex, place both types of components on the opposite side of the board as the stiffener. Then configure the stiffener to have holes larger than the exposed pad of a through-hole component to allow for soldering on the bottom side of the board.

To increase the stability and mechanical support for components placed in flex designs, stiffener must be added to a location in the stackup that will reinforce components without interfere with assembly. Keep soldering access at top of mind when designing the stackup and placing components. Now that component placement has been completed, connections can be made through routing, vias, and copper pours.

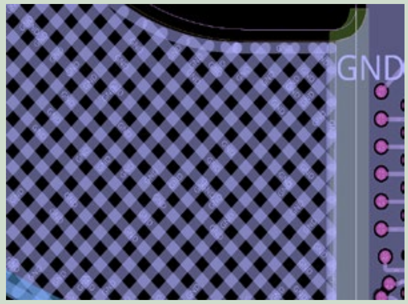
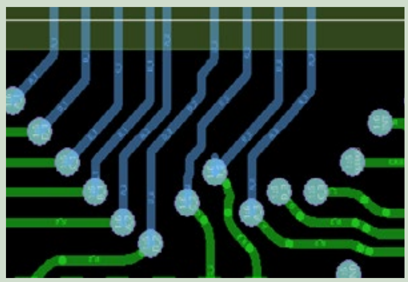
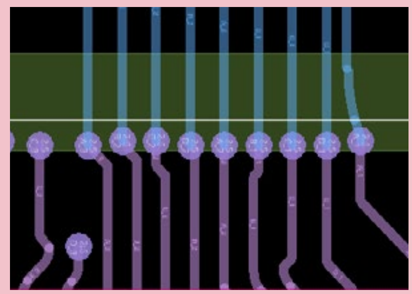
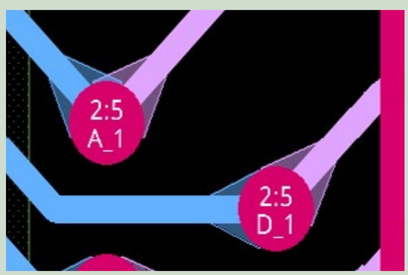
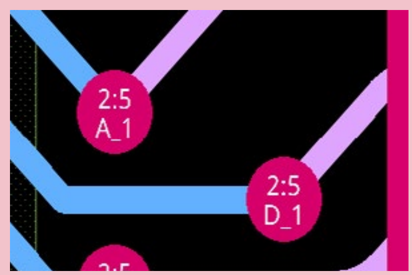
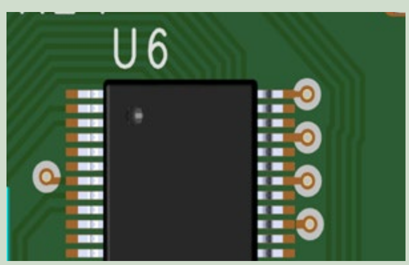
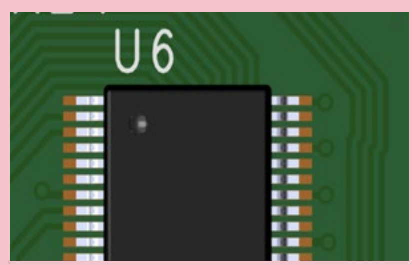
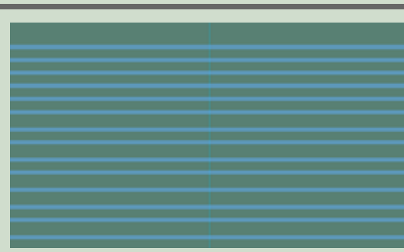
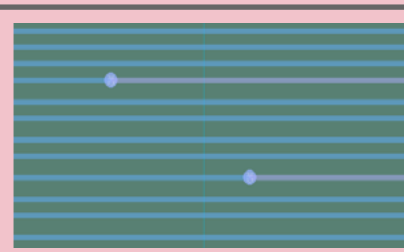
KEY CONSIDERATIONS FOR FLEX DESIGNS
ROUTING BEST PRACTICES

Due to the flexible nature of these designs, special consideration needs to be given to routing including traces, copper pours, vias, and pads to ensure proper operation and increase the robustness of the PCB. The chart below discusses best practices for routing and provides examples of correct and incorrect implementation.

Routing Best Practice	Correct	Incorrect
Alternate Routing Paths Alternate routing paths when you are routing on more than one layer in the flex section of the design. Try to stagger the traces in channels rather than routing directly on top of the other traces. While this typically doesn't matter in a rigid design, it could create a weak point in the flex due to the thinness and flexibility. Staggering the traces on different layers of the flex design will create an even distribution and decrease the possibility of the flex breaking or cracking.		
Trace Spacing Traces should be equally spaced to make the best use of your flex while creating consistency. Providing adequate spacing will also improve signal integrity and prevent crosstalk.		
Perpendicular Routing Route traces that are crossing bend lines perpendicular to the bend line. Traces that are not perpendicular are more likely to crack when bent, disrupting the electrical connection.		
Curved Traces Instead of the typical 45° angles, curved traces should be used throughout the entire flex design to remove any sharp edges of copper. This is especially important along the curves of the flex. If possible, traces should contour the unique flex outline for best use of space and even distribution.		

KEY CONSIDERATIONS FOR FLEX DESIGNS

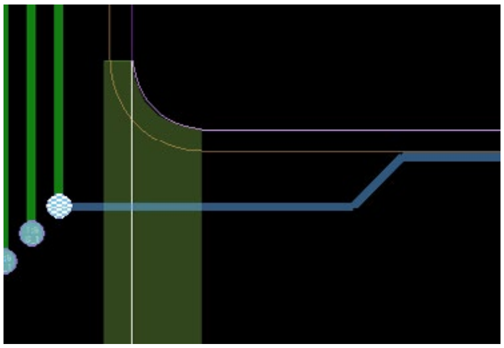
ROUTING BEST PRACTICES

Routing Best Practice	Correct	Incorrect
Hatched Ground Planes Hatched ground planes can be used to improve flexibility as using less copper will decrease thickness and increase flexibility. Hatched ground planes do disrupt the return plane by removing copper. If you are implementing this technique, hatching should be applied on a 45° angle to create a more consistent return path for signals.		
Via and Hole Placement Avoid placing vias, holes, or components within 50 mils (or the manufacturer specified value) of a rigid-flex transition. In this junction, the materials transition from rigid to the flex. Placing objects too close to the transition can decrease stability and result in breakage when bending the design.		
Teardrops and Fillets Incorporate teardrops and fillets on pads, vias, and traces for additional anchoring. Vias and traces should incorporate additional anchoring methods to prevent them from peeling off the flex portion of the design.		
Via Tenting Via tenting covers the via hole with solder mask to protect it from environmental factors. When possible, vias should be fully tented to protect them from peeling when the design is flexed.		
Bend Lines Avoid placing vias in flex areas of the design that do not include stiffener as much as possible to minimize breaking. Absolutely no vias should be placed in the bending area of the flex.		

Following these best practices will help keep the flexibility of your design while increasing reliability and durability. These routing best practices for flex designs can be ensured by determining and implementing rules during the PCB layout process.

KEY CONSIDERATIONS FOR FLEX DESIGNS
DESIGN RULES

Design rules will dictate critical elements in the PCB and are often driven by manufacturer's capabilities in conjunction with the electrical requirements. When defining the rules in which the PCB layout needs to follow, it is important to work with the flex fabrication house to determine:

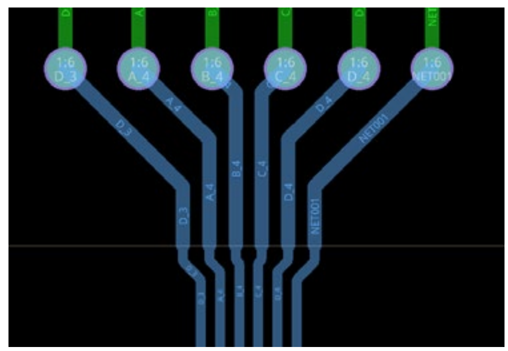


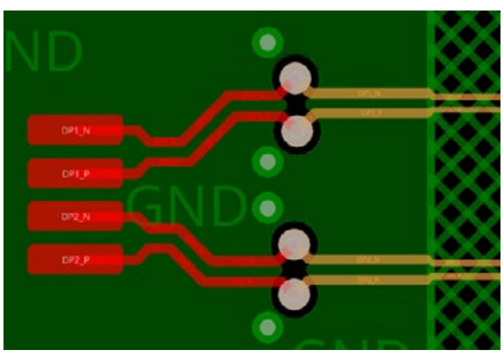
Board Edge Clearance

With flex designs, the board edge clearance can often be minimized on flex regions due to the fabrication method. During the fabrication process, a laser router can be used instead of the typical mechanical router to create the board edge. This will minimize the clearance required between the board edge and copper in the design and produce additional space in the flex area which can be used for routing.

Minimum Trace Width and Spacing

As always, even with rigid designs, the minimum trace width and spacing requirements will be driven by the copper weight required for proper electrical functionality and the capabilities of the fabrication house. These values will differ based on the manufacturer as well as the rigid and flex zones. Keep in mind that there will often be a price increase for high-precision configurations with tight spacing and thinner traces.





Controlled Impedance

Controlling impedance across flex is achievable but requires some additional specifications. To achieve controlled impedance, trace width and spacing requirements will change as you cross the rigid-flex boundary. Flex materials, layers, and hatched planes will impact impedance calculations and standard calculators may not be accurate. Work collaboratively with the fabrication house to determine a modified trace width and spacing across the flex area of the design which will meet your impedance requirements.

These rules can be incorporated as constraints in your CAD software to ensure that manufacturing rules are adhered; however, it is essential to obtain this information early in the design process to streamline the design process, avoid late-stage design changes and minimize errors or delays when the design is sent to manufacturing.

KEY CONSIDERATIONS FOR FLEX DESIGNS
FABRICATION: EARLY INVOLVEMENT

Early fabricator involvement is critical in the flex and rigid-flex fabrication process; not only instituting 1-on-1 communication with the manufacturer but fostering multi-disciplinary involvement to ensure the electrical, mechanical, and management teams are all aligned. Establishing early fabricator involvement will help to:

Drive Design Decisions

Improve Collaboration

Find Creative Solutions

For flex designs early fabricator involvement is especially important as there are specific criteria that fabrication houses need to provide in terms of capabilities, materials, thicknesses, bend radii, recommendations for via technologies, and more to confidently make design decisions. Each stakeholder in the design process will have different areas of concern that will produce critical trade off decisions:

Management

- Lead Time
- Fabrication Cost
- Design Cost

Electrical

- Material Requirements
- Copper Thickness
- Minimum Trace Width
- Minimum Trace Spacing
- Controlled Impedance

Mechanical

- Board Thickness
- Bend Radius
- Fitment with Housing

Producing a flex or rigid-flex board is very different from a rigid board in terms of materials, fabrication capabilities, mechanical restrictions, cost, and lead time. Establishing early fabricator involvement will help designers fine creative solutions for flex design challenges that save time and money while producing a manufacturable PCB; however, to ensure accurate communication of the increased flex design complexities, additional data and information must be included in the fabrication documentation and manufacturing deliverables.

KEY CONSIDERATIONS FOR FLEX DESIGNS

FABRICATION: MANUFACTURING DELIVERABLES

Manufacturing deliverables must be provided to communicate your design intent and accurately fabricate and assemble your PCB design. The required manufacturing package will vary per fabrication house but typically includes:

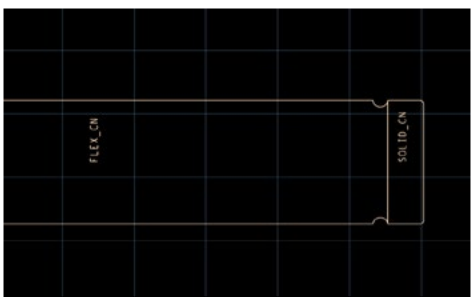
- Gerber Files
- Neutral Database File (IPC-2581 or ODB ++)
- Fabrication Drawing
- Assembly Drawing
- Pick and Place File
- BOM
- Schematic

For flex and rigid-flex designs, the Gerber files, fabrication, and assembly documentation will need to be modified with additional information.

1

Gerber and Neutral Database Files

Gerber files are standardized file formats used in PCB manufacturing to define the different layers of a board design. These files contain detailed instructions for PCB fabrication, including copper traces, solder mask, silkscreen, and drill holes. Neutral database files, such as IPC-2581 or ODB++, expand upon this information providing intelligent design data to improve the manufacturing, assembly, and testing processes. For flex and rigid-flex designs, additional files should be included in these packages to ensure proper fabrication.

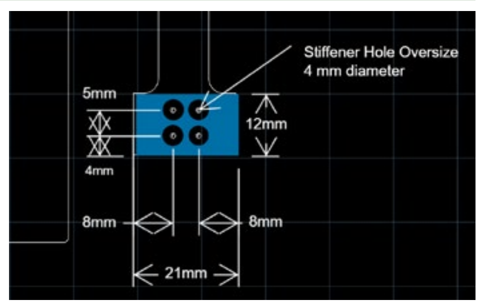


Outlines of Each Unique Stackup or Zone

A Gerber file containing outlines of each unique stackup or zone is required to ensure fabrication knows where the boundaries lie between regions and stackups such as flex, rigid, and stiffener.

Stiffener Outlines Defining Hole Sizes

Typically, the hole sizes for stiffener are oversized to help with alignment and exposure of the copper. With differing hole sizes between the flex and stiffener layers, a Gerber file detailing the stiffener outlines and defining the hole sizes should be provided to manufacturers.



Top and Bottom Coverlay

Make sure to include the Gerber files for the coverlay for both the top and bottom of the flex areas. This is especially important in rigid-flex and combinational designs where both soldermask and coverlay are required.

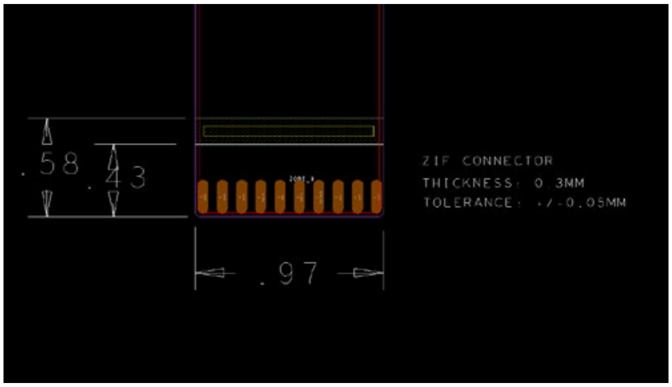
KEY CONSIDERATIONS FOR FLEX DESIGNS

FABRICATION: MANUFACTURING DELIVERABLES

2

Fabrication and Assembly Documentation

Fabrication and assembly documentation refers to the detailed set of drawings, specifications, and files provided to manufacturers for fabricating the bare PCB and assembling components onto it. These documents ensure that the manufacturer accurately builds the PCB according to the design requirements. For flex and rigid-flex designs, the following additions to the fabrication and assembly documentation may be required:

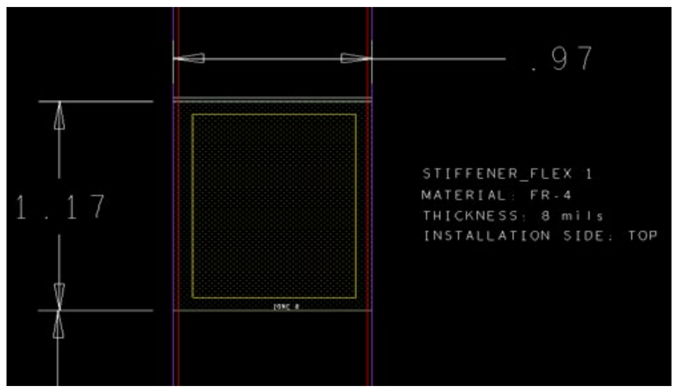


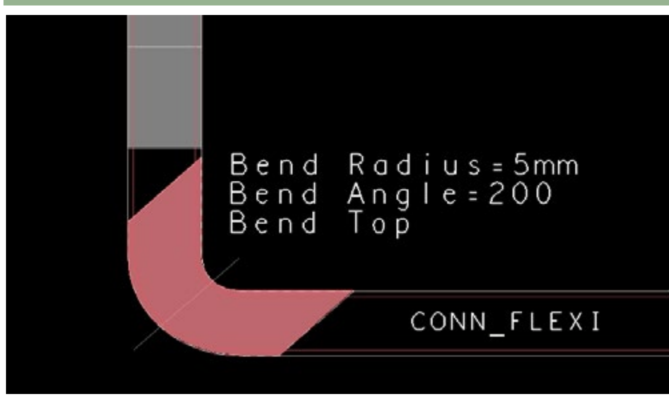
ZIF Connector Details

This is especially important for ZIF connectors which demand exact requirements that are critical for the component to install into the connector correctly such as the width of the flex, minimum length of the stiffener, and thickness required for installation.

Stiffener Installation Side

Call out the installation side for the stiffener. While this should be included in the stackup details, it is important that it is referenced very clearly within your documentation.





Bend Lines and Final Assembly

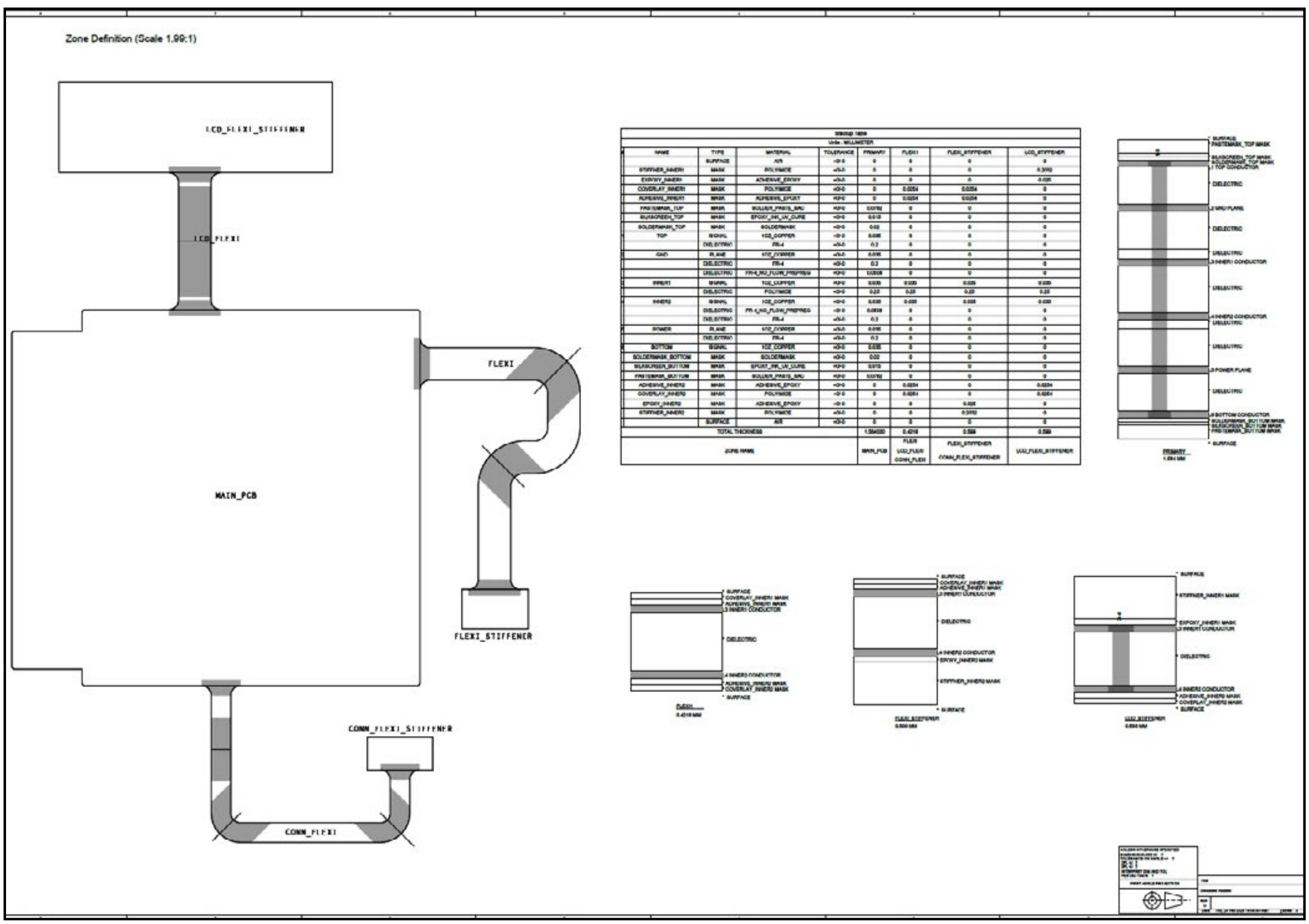
Defining bend lines and providing a 3D model of the final bent assembly will call out the final assembly details and can streamline the testing and verification of the manufacturing process.

KEY CONSIDERATIONS FOR FLEX DESIGNS

FABRICATION: MANUFACTURING DELIVERABLES

3 Fabrication Drawing

Within the fabrication drawing, ensure the stackups used in the design are documented clearly and accurately. This can be achieved by creating a stackup table which includes the layers, materials, and thicknesses for the multiple stackups that are included in the PCB. For designs with three or more stackups or zones, it is critical to produce a corresponding drawing detailing the location of each zone. This is important to clearly communicate the design intent to the fabricator to eliminate errors with stiffener placement, stiffener thickness, and more to produce the flex or rigid-flex design accurately.



KEY CONSIDERATIONS FOR FLEX DESIGNS

FABRICATION: MANUFACTURING DELIVERABLES

4

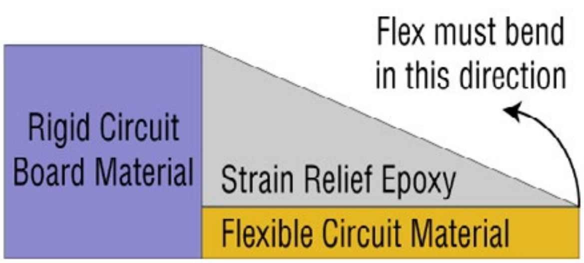
Fabrication Notes

Fabrication notes are detailed instructions and specifications provided to the PCB manufacturer to ensure the board is produced accurately and to the required standards. These notes clarify critical information that cannot be conveyed through just the drawing. Fabrication notes for flex and rigid-flex designs will need to be modified as there are different standards and fabrication methods which must be adhered to:

IPC Standards

References to IPC standards should be adjusted to reflect any changes between rigid and flex designs. This includes referencing IPC-6013 (flex) instead of or in addition to the standard IPC-6012 (rigid) for qualification and performance metrics. Soldermask and coverlay material callouts will also need to refer to different IPC specifications.

- IPC-4202: Flexible Base Dielectrics for Use in Flexible Printed Boards
- IPC-4203: Cover and Bonding Materials for Flexible Printed Circuitry
- IPC-4204: Flexible Metal-Clad Dielectrics for Use in Fabrication of Flexible Printed Circuitry



Strain Relief

Epoxy might be used by manufacturers at rigid-flex transitions for strain relief. Strain relief strengthens the transition between flex and rigid materials, which is a very fragile junction that can be easily ripped or damaged, at the cost of lowering the bend radius. If strain relief is included in the design, make sure to include relevant specifications in the documentation.

Including this information in your manufacturing deliverables will ensure your flex design intent is communicated accurately and ensure a smooth production process. With all these different requirements and design techniques required to design flex boards, it can be a real challenge to keep it all straight- this is where your CAD tools come into play.

NAVIGATING FLEX DESIGN CHALLENGES
HOW CAD SOFTWARE SIMPLIFIES THE PROCESS

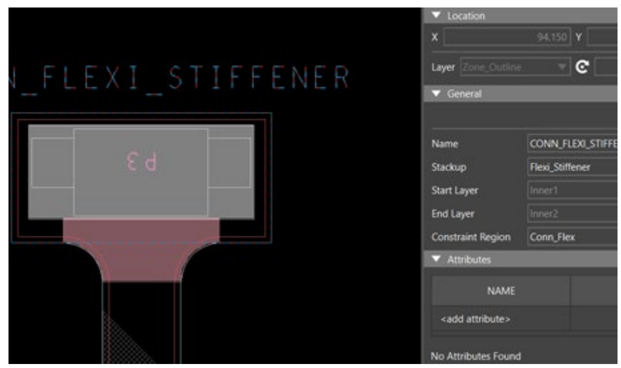
CAD software can be used to help manage the additional complexity found in flex and rigid flex designs. To successfully produce a rigid-flex PCB, employ a software solution that leverages the following features:

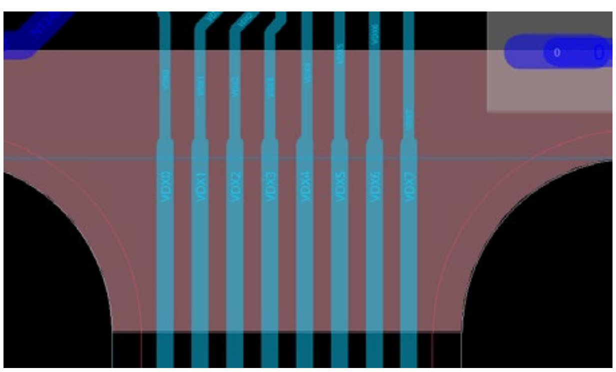
Multiple Stackup Support

The numerous stackups included in rigid-flex designs must be properly managed. Your software solution should include the ability to define multiple stackups with easy selection of the flex and rigid materials required for the PCB. Both graphical and tabular visualization is beneficial to ensure cross-section accuracy.

Zone Definition and Management

Each section of the PCB (rigid, flex, flex with stiffener) requires a different stackup and rules. These sections, or zones, should be defined and managed efficiently. CAD software should allow you to define a zone as well as assign the corresponding stackup and rules for that zone. The zone definition should be clearly defined and visible in any manufacturing documentation.



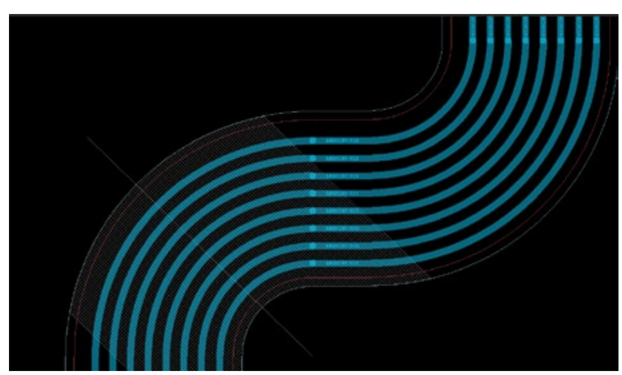


Region-Based Rules

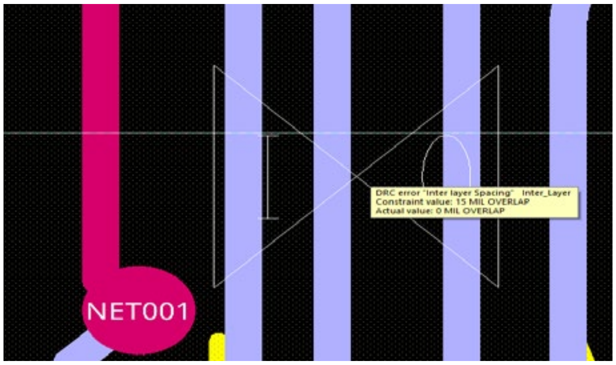
Flex and rigid zones on the PCB will have different rule requirements due to the varying materials and fabrication capabilities. To accommodate this, your CAD solution should allow rules for trace width, trace spacing, clearances, vias, and more based on the region of the board. This will ensure design rule checks are performed throughout layout, errors are identified early when changes can be easily implemented, and improve manufacturability.

Arc and Contour Routing

Traces in flex zones must conform to the unique board outline and geometry, often curved. At a minimum, your CAD software should include the ability to route traces in an arc instead of 45° or 90° degrees. As a bonus, contour routing allows designers to automatically conform to unique geometry by contouring to a selected object (keepins, outline, trace, etc.) to quickly route traces.



NAVIGATING FLEX DESIGN CHALLENGES
HOW CAD SOFTWARE SIMPLIFIES THE PROCESS

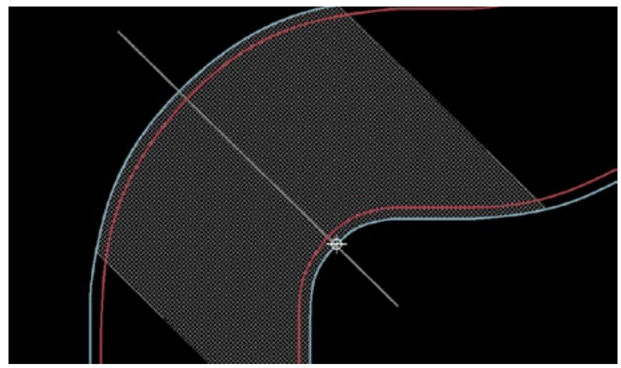


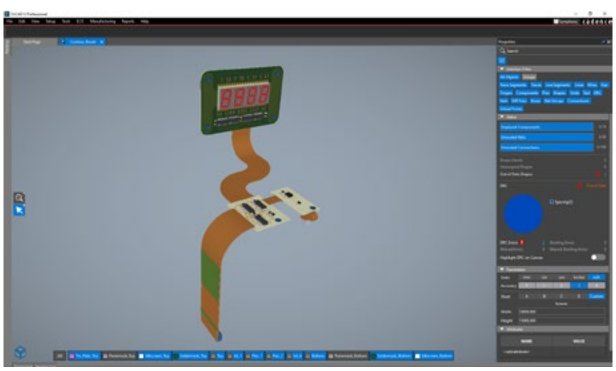
Inter-Layer Checks

With rigid-flex designs, there are many scenarios in which rules need to be checked in between layers such as transition zones, overlapping layers, and oversized holes. Errors can be easily overlooked when taking a manual approach to rule checking in these situations. Your CAD tool should provide automated checking between layers in the PCB design allowing you to define the rule requirements and which layers to analyze.

Bend Definition and Management

For flex PCBs, it is important to incorporate the bend lines upfront in the design process as these bend lines will affect part placement and routing. Your CAD software should include the ability to define the bend location, bend radius, and bend angle. Defining these bend zones will also allow designers to visualize an accurate representation of the final assembly if 3D modeling is deployed.



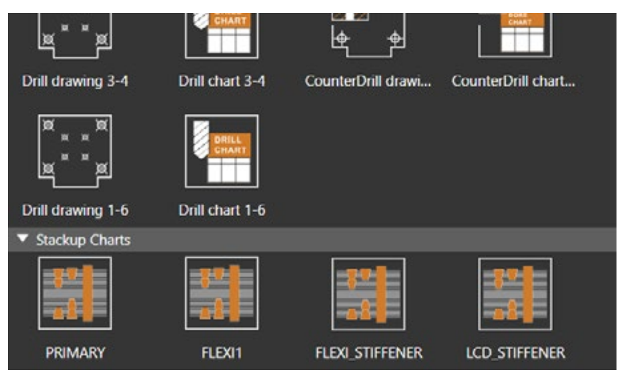


Advanced 3D Modeling

Creating and analyzing a 3D model of the final PCB assembly is critical in rigid-flex designs to verify mechanical feasibility before the board is fabricated. When creating the 3D model, it is important to accurately configure the thickness for each region as well as bend to create an accurate representation of the PCB. This can enable clearance checking between separate regions of the board when the model is bent to guarantee a functional assembly.

Efficient Manufacturing Documentation

Attention to detail must be given to manufacturing documentation for flex and rigid flex designs to clearly communicate the design intent. Your CAD software solution should provide efficient methods for creating the required artwork, fabrication, and assembly drawings-including zone definitions and support for multiple stackups through pictorials and charts. Additional supportive features include a real-time sync between the PCB and documentation to ensure accuracy as well as automatic generation and packaging of files.



It is important to select a CAD tool that includes these features and support the additional complexities of flex and rigid-flex designs. OrCAD X contains these features and more to help create accurate and reliable flex and rigid-flex PCBs.

OVERCOMING FLEX DESIGN CHALLENGES
WITH ORCAD X

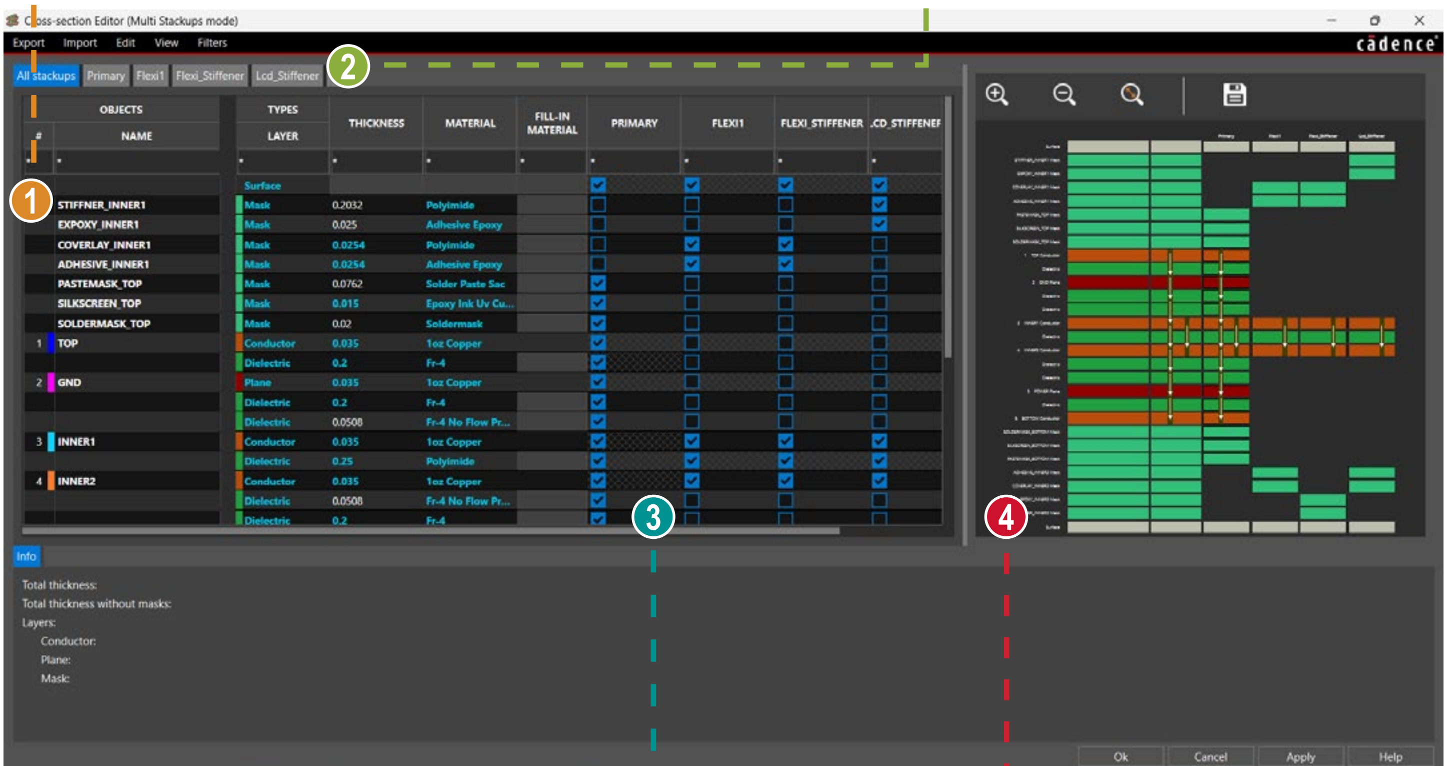
OrCAD X includes the features and capabilities needed to efficiently and accurately design your flex and rigid-flex PCBs. The streamlined layout environment, realistic 3D modeling, advanced routing, and industry-leading constraint management tools or OrCAD X enable engineers to confidently manage the increased design complexities associated with these versatile and cutting-edge designs. To efficiently manage the vertical design intent for differing sections of rigid flex designs, OrCAD X provides:

1 Design Setup

Easily manage the complexities of flex and rigid-flex stackups with efficient methods for defining flex materials, configuring cross-sections, and managing multiple stackups in OrCAD X.

1 Materials
Flex and custom material definitions.

2 Multiple Stackups
Support for multiple stackup definitions within a single board.



3 Layer Selection
Easy selection and modification of board layers in each stackup zone.

4 Visual Representation
Visual cross-section representation of every stackup in the design.

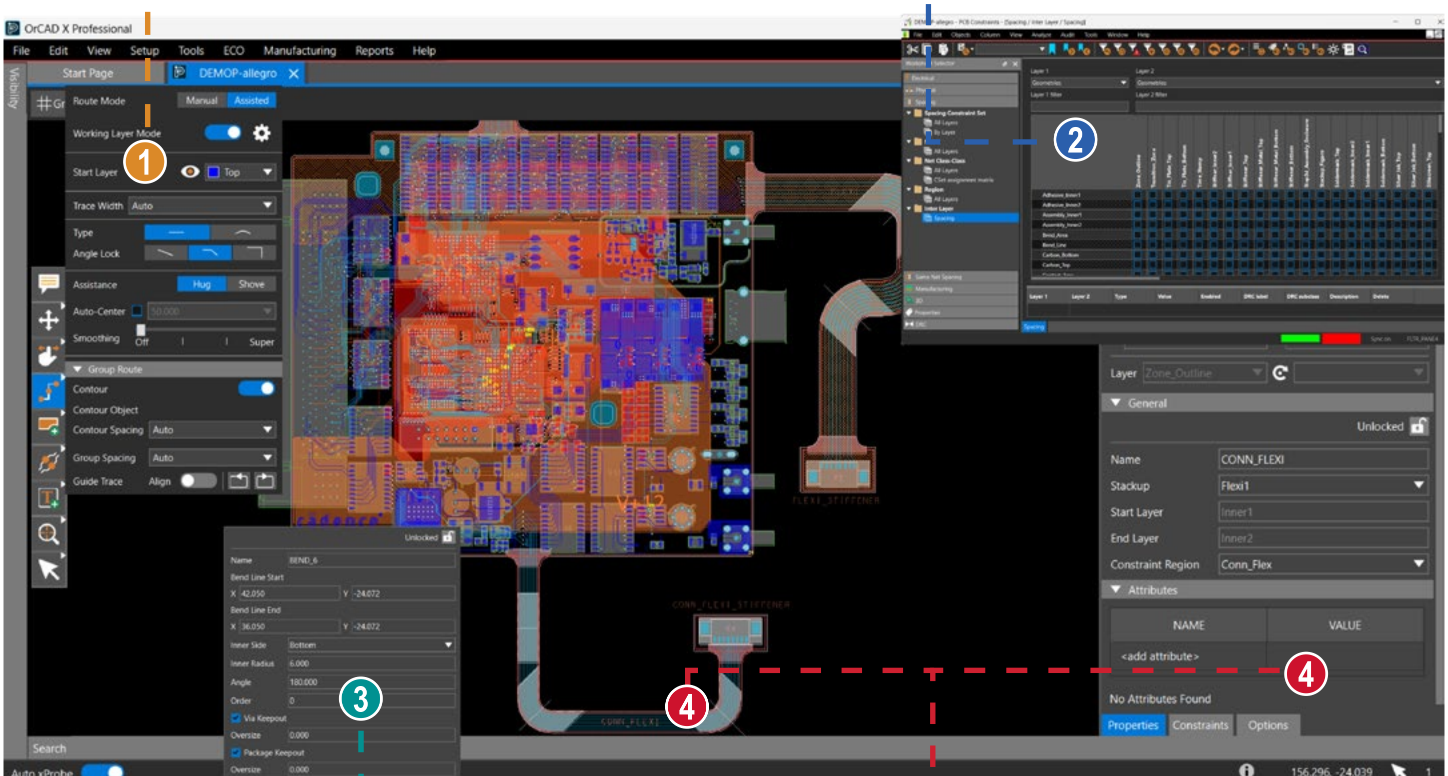
OVERCOMING FLEX DESIGN CHALLENGES
WITH ORCAD X

2 Efficient Design and Layout

OrCAD X contains a streamline design environment with a powerful layout and routing engine. By including the design tools necessary to layout and route flex and rigid-flex PCBs as well as efficient rule definition and checking, engineers can ensure electrical and manufacturing requirements are met throughout the PCB layout.

1 Arc and Contour Routing
Arc and contour routing to quickly adhere to complex flex geometries.

2 Inter-Layer Checks
Easy to configure design rules and inter-layer checks.



3 Bend Management
Efficient bend creation, definition, and management.

4 Zone Management
Zone creation and management with easy definition of corresponding stackups and constraint regions.

OVERCOMING FLEX DESIGN CHALLENGES
WITH ORCAD X

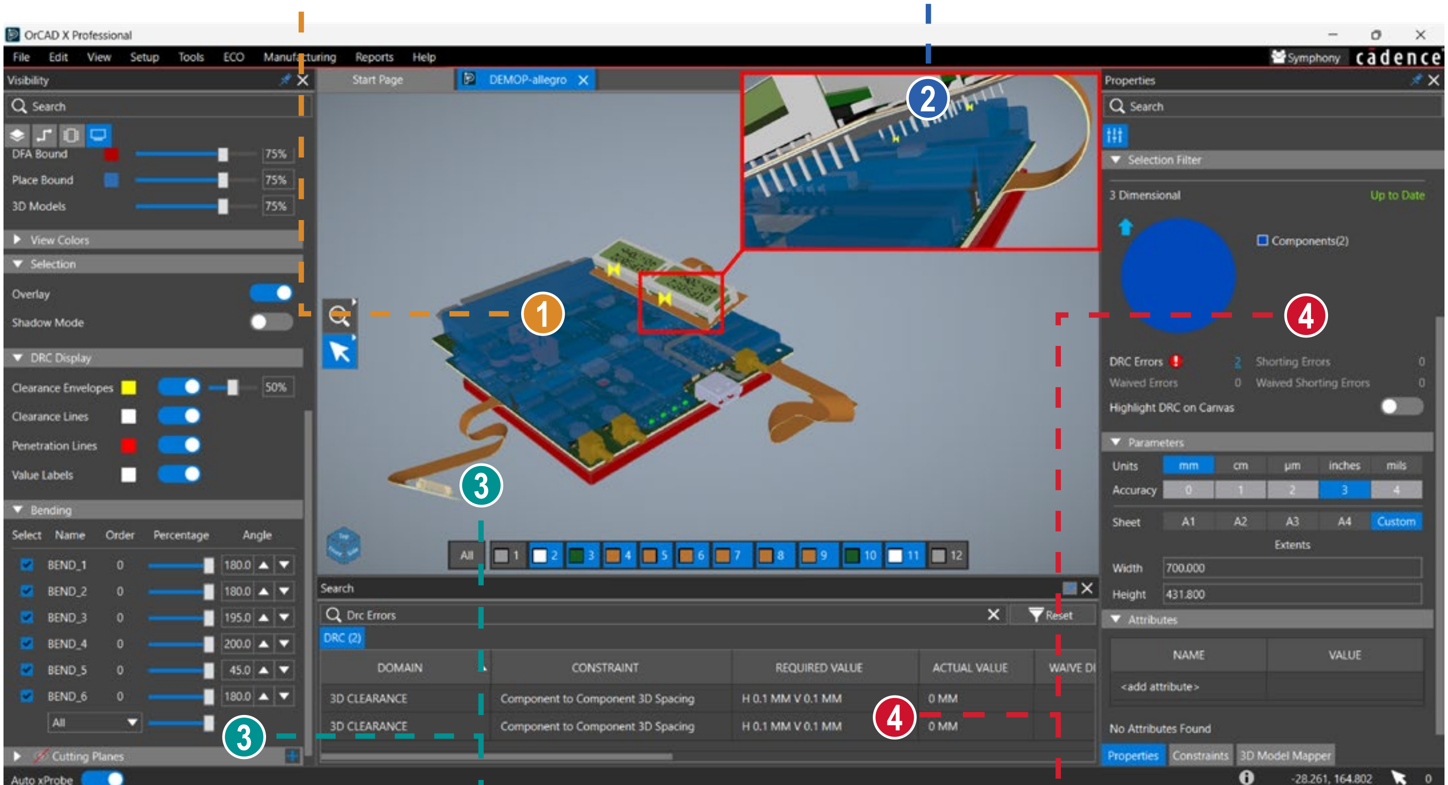
3

3D Verification

For flex and rigid-flex designs, it is imperative to verify the final assembly in 3D to visualize bends and interactions. OrCAD X contains a powerful 3D engine, allowing engineers to quickly transition between 2D and 3D views, bend flexible sections of the PCB in real-time, and identify and analyze collisions.

1 3D Modeling
Realistic modeling of the PCB, components, and mechanical elements.

2 Collision Detection
Collision detection and analysis for design verification.



3 Bend Visualization
Real-time bending for a realistic visualization of the final assembly.

4 3D Design Rule Checks
Detailed analysis of design rule errors and collisions.

OVERCOMING FLEX DESIGN CHALLENGES
WITH ORCAD X

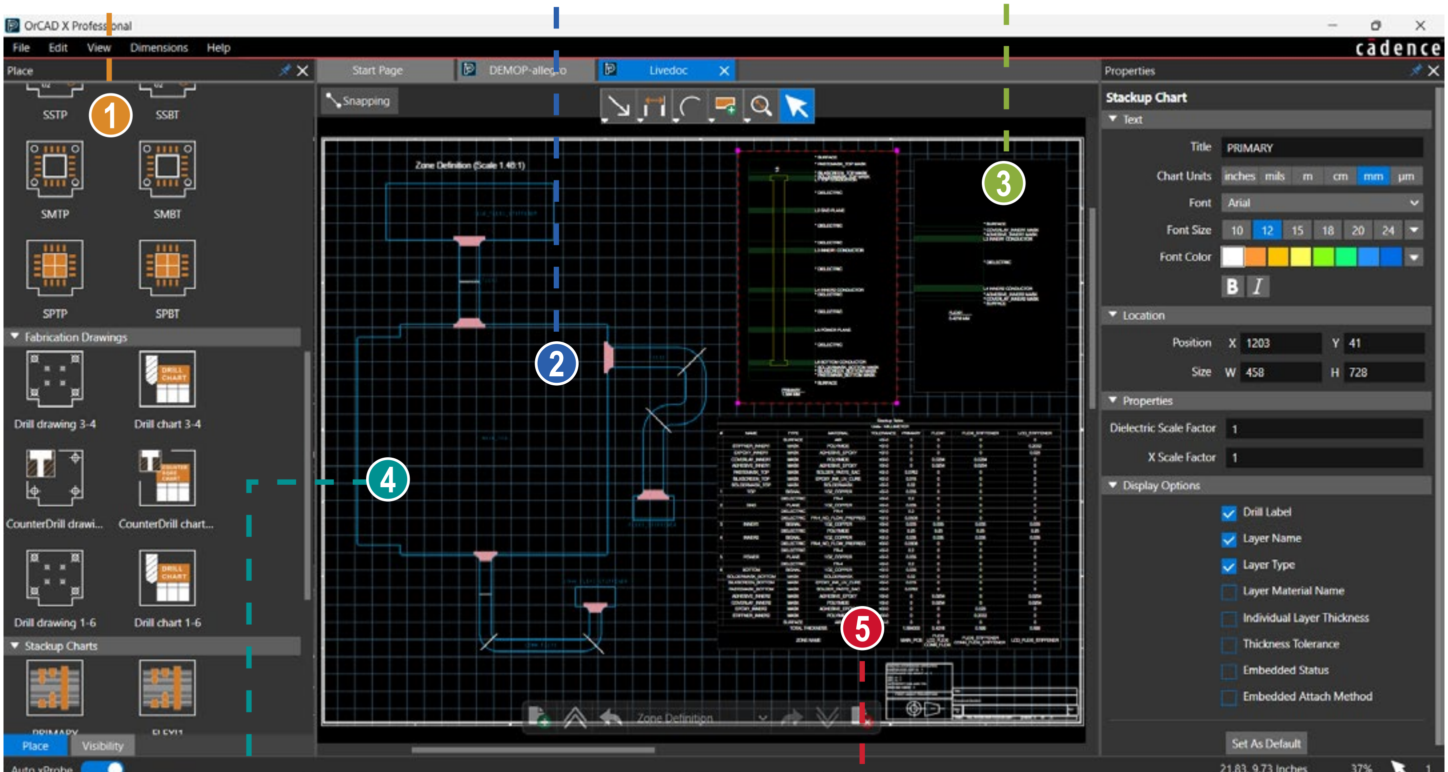
4 Manufacturing Documentation

Once the PCB is complete, it is critical to accurately convey the design intent to manufacturers through artwork definition, fabrication drawings and assembly drawings. OrCAD X includes a real-time PCB documentation authoring tool, LiveDoc, which provides a templated approach to documentation creation, a real-time sync with the PCB canvas, and multi-stackup support for complex cross-sections.

1 Template-Based
Drawing templates based on the design stackup.

2 Customizable Views
Customizable views to show zones, bend definitions, etc.

3 Customizable Charts
Individual stackup charts with customizable options.



4 Real-Time Sync
Real-time sync with the PCB canvas and automatic updates to documentation to ensure accuracy.

5 Stackup Tables
Multiple stackup support for stackup tables including zone definition.

OrCAD X can improve your rigid-flex design process, helping achieve functional, manufacturable flex and rigid-flex boards on the first pass.

To learn more go to:

OrCAD X

Info

REFERENCES

EMA Design Automation and Lauren Waslick, Newgrange Design. Flex in Focus: Designing the Perfect Stackup for Flex. 2025. https://www.ema-eda.com/ema-resources/on-demand-webinars/flex-in-focus-designing-the-perfect-stackup-for-flex/

EMA Design Automation and Lauren Waslick, Newgrange Design. Flex in Focus: Designer's Guide to Flex PCB. 2025. https://www.ema-eda.com/ema-resources/on-demand-webinars/flex-in-focus-designers-guide-to-flex-pcbs/

EPEC Engineered Technologies. ITAR/Military Flex and Rigid-Flex PCBs. https://www.epectec.com/flex/industries-market-focus/flex-and-rigid-flex-pcbs-for-itar-applications.html

Glenair Flexible Circuit Board Assemblies. Flex Circuit Assemblies: Turnkey, Connectorized PCB Solutions. https://www.glenair.com/flex-circuit-assemblies/index.htm

GT Smart Technology Co. Automotive Flexible PCB Board With Blue Solder Mask And White Silkscreen. https://www.gtpcb.com/sale-40891901-automotive-flexible-pcb-board-with-blue-solder-mask-and-white-silkscreen.html

Molex (2016). 0.5 PITCH NON-ZIF CONN. SMT ST ASSY Product Drawing. https://www.molex.com/content/dam/molex/molex-dot-com/products/automated/en-us/salesdrawingpdf/781/78127/781272210_sd.pdf?inline

Multi-cb Printed Circuit Boards. Flexible Circuit Boards. https://www.multi-circuit-boards.eu/en/products/printed-circuit-boards/flexible-pcb.html

Newgrange Design. https://www.newgrangedesign.com/