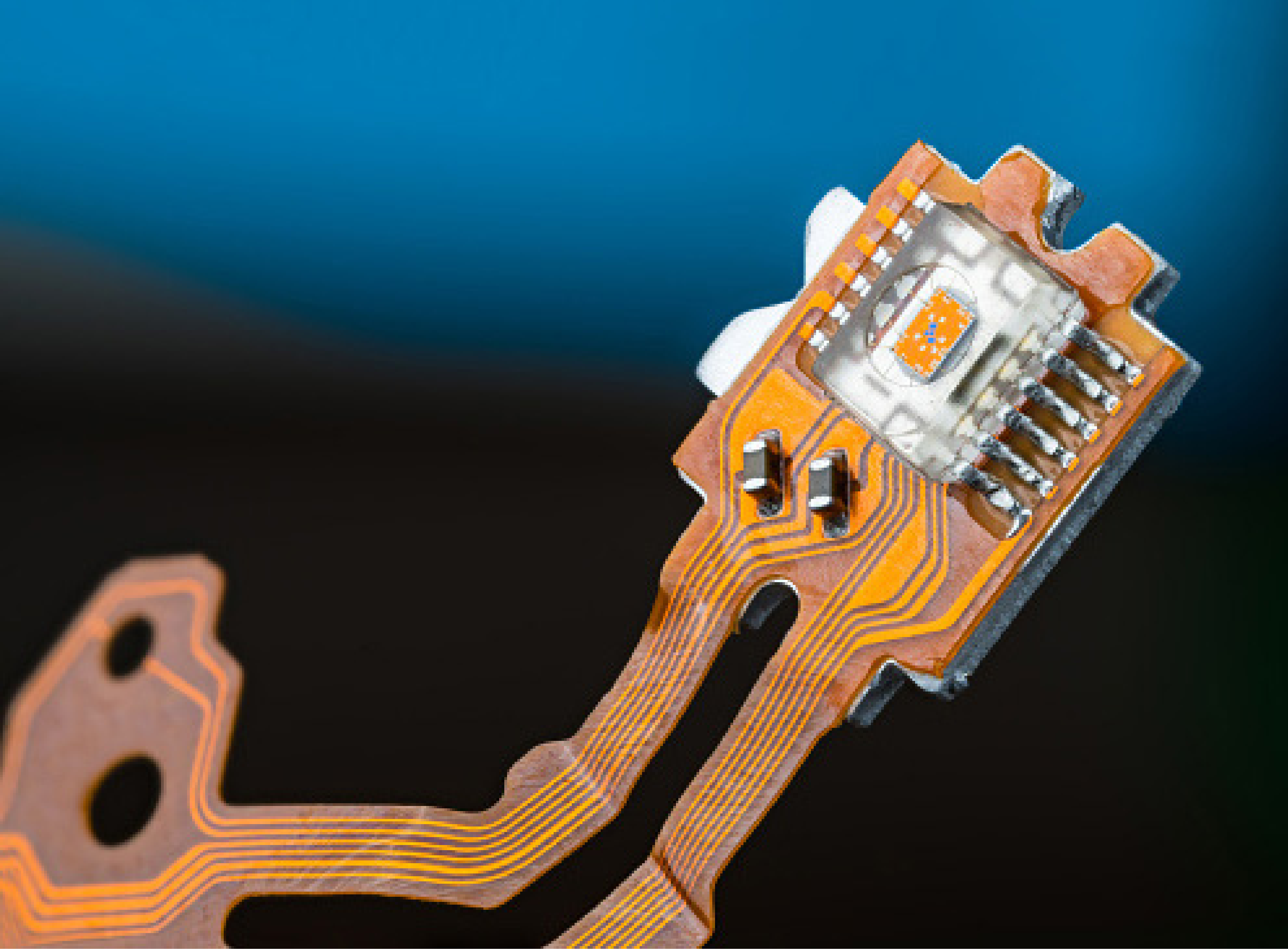


DIGITAL INDUSTRIES SOFTWARE

Rigid-flex PCB design guidelines

Providing a solid foundation for creating innovative and reliable rigid-flex PCB designs

siemens.com/software

SIEMENS

This e-book is designed to guide you through the intricacies of rigid-flex printed circuit board (PCB) design. In today's electronics industry, there has never been more demand for compact, efficient and versatile PCBs. Rigid-flex technology has emerged as a game-changer, offering engineers the flexibility to design boards that can bend and flex without sacrificing performance or reliability.

Given its unique challenges and requirements, understanding and mastering the rigid-flex PCB design process can be daunting. Whether you are a seasoned PCB designer looking to expand your skill set or a newcomer eager to learn, this e-book is tailored to provide you with a simplified and structured approach to rigid-flex design.

The content presented here is organized into a step-by-step design flow, which will walk you through each stage of the design process. From creating a master stackup to defining board outlines, assigning layers, establishing route borders and more – each step is explained in detail with clear instructions and illustrations.

Furthermore, the e-book covers essential aspects like creating bend areas, place and route procedures, 3D visualization options, design rule checking (DRC) and generating manufacturing outputs. Special emphasis is placed on adhering to flex-specific rules and leveraging output engines like ODB++ to effectively communicate multi-zone stackup details to fabricators.

Whether you are working on a flex-only or a flex-rigid design, the principles and guidelines outlined in this e-book are universally applicable, providing you with a solid foundation to create innovative and reliable rigid-flex PCBs.

Gregory Beers
Siemens Digital Industries Software

Quick overview

There are only a few differences between executing rigid and flex PCB designs in your PCB design tool (examples here are using Xpedition™ software, part of the Siemens Xcelerator business platform of software, hardware and services).

Regional stackups

- You can have a unique stackup in each region of the board
- Each "region" is defined by a separate board outline meaning that you can have multiple board outlines in a flex design. Each outline can have a type: "rigid" or "flex"
- Each board outline is associated with a stackup
- Two outlines must share a side of the outline
- There is always a master stackup that is the sum of all regional stackups. This drives the tool. Each board outline has a subset of this stackup

Flex specific layer types

- The special layers you use in flex designs such as flex core, cover layer, adhesive and stiffeners are all represented and understood in the Xpedition stackup

Named solder mask layers

- In traditional rigid designs there is only a top and bottom solder mask
- With multiple stackups, there is need for separate mask layers for each board region
- You can add any number of named solder mask layers

Flex elements in the pad stack

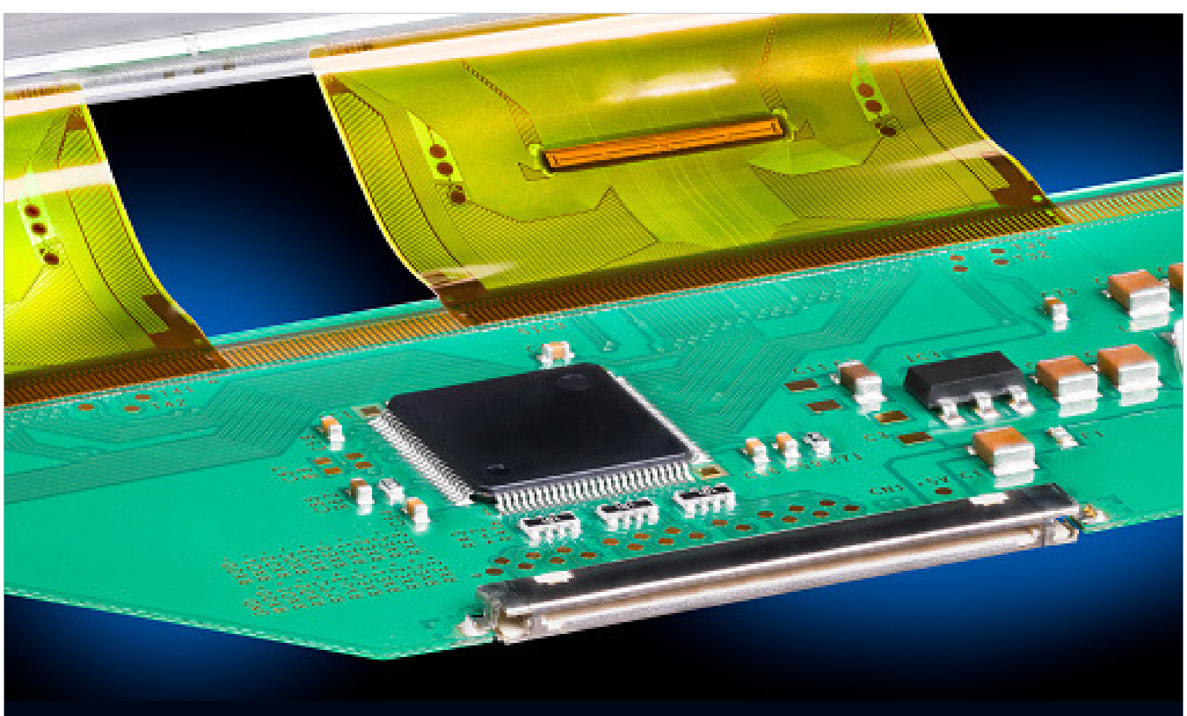
- The pad stack can hold cover layer openings for through hole pads and SMD pads
- It can also hold pads that define hole openings in stiffeners

Bend areas

- Bend areas are areas that define where and how the flex circuit bends. DRC and 3D design use this information

Fabrication output

- We need to convey design intent and regional stackups to the board fabricator. Xpedition Layout ODB++ output fills this purpose and is the preferred output format for flex and flex/rigid



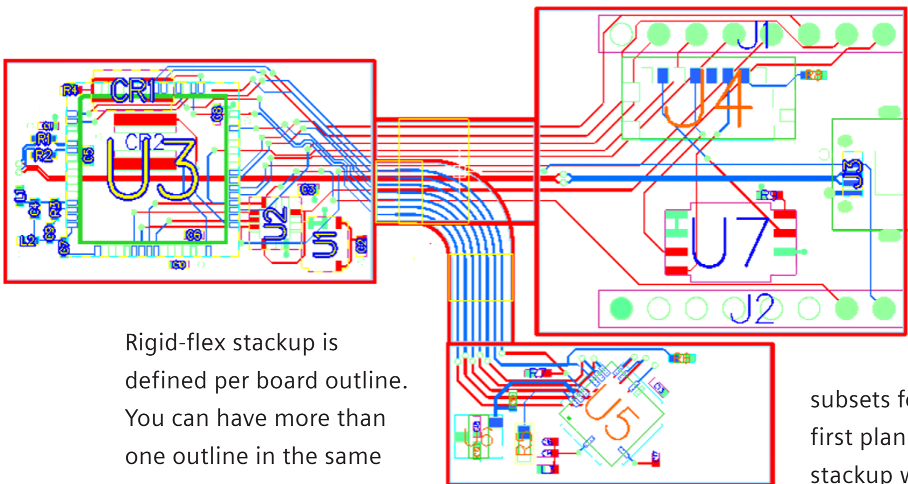
Simple 10-point rigid-flex design process

Given the peculiarities listed to the left, here is a simplified design flow for rigid-flex PCB designs.

1. **Set the "rigid-flex"-technology before starting your design.** This notifies layout that you are doing a flex/rigid-flex design.
2. **Create a master stackup** that has all layers from all regions of the board included.
3. **Define multiple board outlines.** Give each a name to identify each sub- board and give each a type: flex or rigid. Even if a single flex outline is used, the name and flex technology must be set.
4. **Assign subsets of layers** from the master stackup to each outline, thus defining the stackup for each region.
5. **Establish route borders.** If multiple board outlines have been created, the appropriate route borders must be created per board outline.
6. **Create bend areas.** Attributes on each bend area defines the bend radius, bend angle and direction of bend as well as a few settings controlling if vias and components are allowed in the bend zone.
7. **Place and route as usual**. All outer layers can be used for placement and components will automatically end up on the correct layer when placed. Thus, the same component can be placed in any of the board outlines without cell or pad stack alterations.
8. **Display in 3D if desired.** 3D can show flat or folded assembly and use the bend areas to model the fold.
9. **Run appropriate DRC checks.** Several flex specific rules are checked in real time with "correct by design" and some rules are checked in batch DRC.
10. **Create outputs.** All output engines can export manufacturing data for a flex or flex/rigid design. ODB++ is especially well suited and can correctly convey the multi-zone stackup to the fabricator.

STEP 1

Regional stackups and multiple board outlines



Rigid-flex stackup is defined per board outline. You can have more than one outline in the same design and outlines may overlap partly or completely.

This approach is chosen to offer maximum design flexibility with less manual work and less risk when doing complex flex systems design – especially when boards overlap.

Here is a simple but realistic design example that includes rigid and flexible regions as well as a combination of solder mask and flex cover layer and a stiffener.

The first step is always to review the layers you need in all different regions of the design and then create a master stackup that includes all the layers that you will need.

It's only after this step that we can assign stackup subsets to different board outlines.

Preparing to set up the layer stackups

Important: To set up the master stackup and then create stackup subsets for each board outline, we need to first plan our design and consider what stackup we need in each board outline and then merge all subsets into a master stackup that we can enter into the Xpedition layout.

You can do the planning in the stackup editor and thus sequentially build the master stackup. The stackup for each board in the sample design is shown here to help understand the master stackup:

In this example we have not defined adhesive layers separately. Some flex materials have integrated adhesive and then you must decide if you want the adhesive in the stackup or just increase thickness of the flex layer.

All three rigid boards

Soldermask_Top
SIGNAL_1
FlexCore_1
PLANE_2
Dielectric_2
PLANE_3
FlexCore_4
SIGNAL_4
Soldermask_Top

Flex_connector

CoverLayer_1
SIGNAL_1
FlexCore_1
PLANE_2
CoverLayer_2

Flex_connector 2

CoverLayer_2
PLANE_3
FlexCore_4
SIGNAL_4
CoverLayer_4

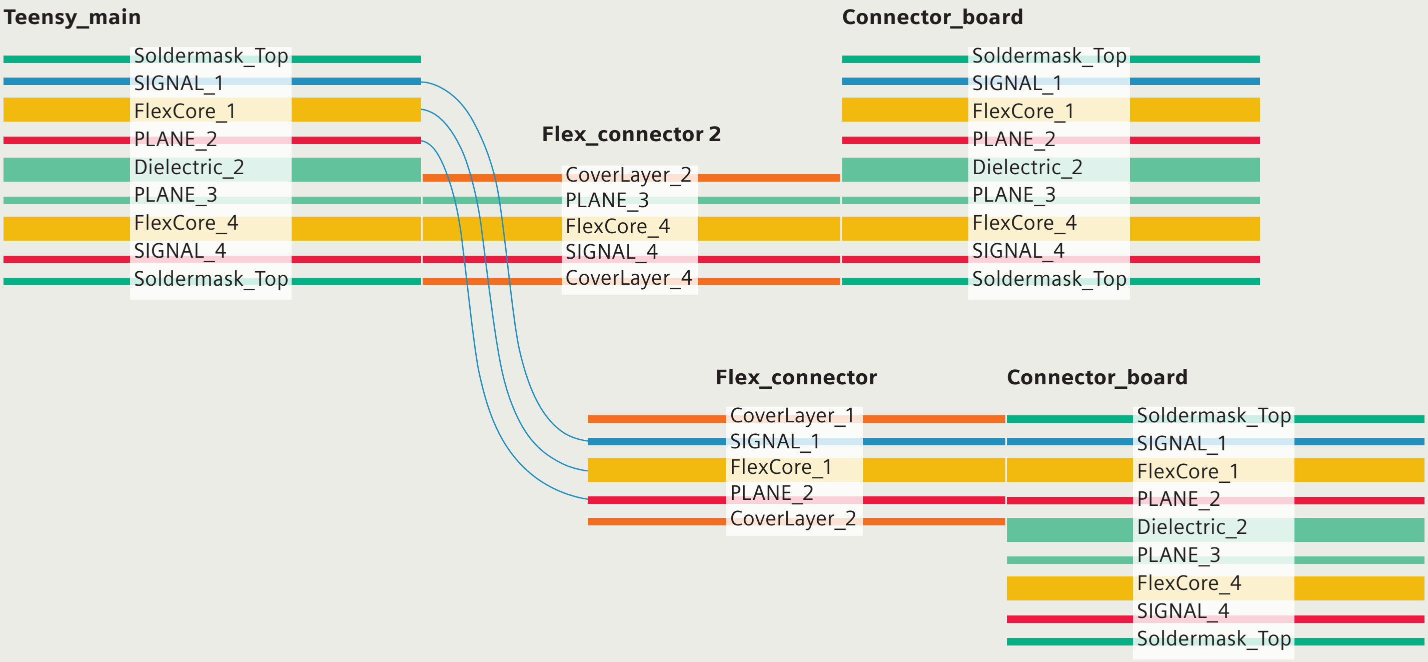
STEP 2

Create a master stackup

The master stackup

All these stackups must now be added into the master stackup where all layers are defined and are placed in correct order. For some designs this may be a massive stackup as it has to hold a merge of every layer from every outline.

There is no command or function to do this merge. You manually enter each layer in the stackup editor. The boards are built together this way:



STEP 3

Define multiple board outlines

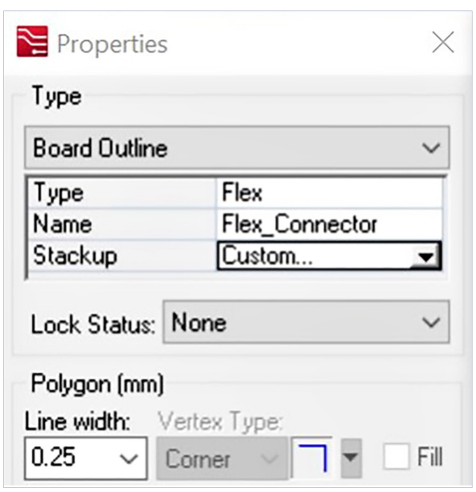
Creating multiple board outlines for rigid-flex
Before we can assign stackup to the different outlines, we have to create them.

This is simply a matter of drawing board outlines or importing using DXF or other methods. With flex technology enabled, Xpedition Layout changes to allow any number of board outlines in a design. Each board outline must be assigned a **Name** and a **Type**.

Type is either rigid or flex.

The **Name** is just an identifying name such as **Connector Board** or **FlexCable2**.

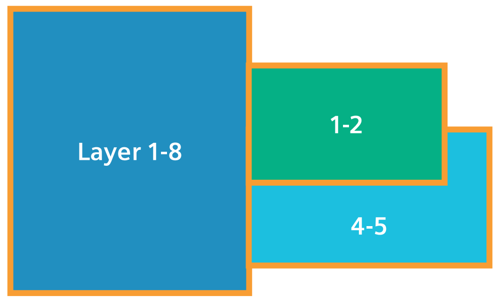
In a design with multiple board outlines, each outline must have a coincident side with at least one other outline (so the multiple board pieces connect together).



Multiple board outlines may completely overlap or partially overlap if desired. However, overlapping outlines cannot have any layers from the stackup in common (that could not be fabricated).

This example is legal as the overlapping outlines don't have any common layers:

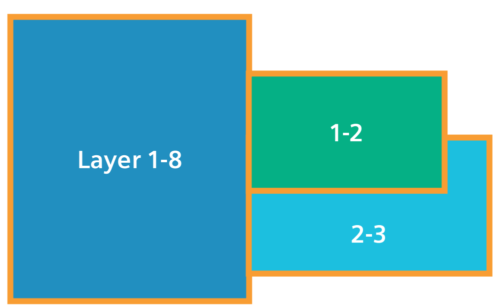
OK



Ensure that route borders don't unintentionally overlap
It can be tricky to have two board outlines exactly coincide along one edge. Instead, you can let one outline slightly overlap the other and use **Draw > Trim Board Outlines** to trim off the overlap.

Floating board outlines (not sharing a side with another outline) and incorrectly overlapping board outlines are found in the online DRC section of the **Hazards Explorer**. (**Analysis > Hazard Explorer...**).

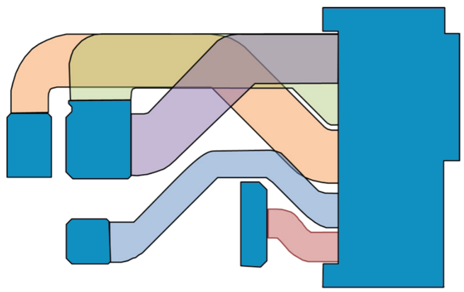
Not OK!



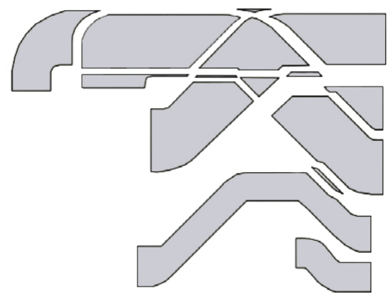
Creating board outlines – a couple schools of thought

If you had to work with just one board outline you would not be able to independently manage each zone in case of overlapping layers. This design would have over eight zones with each zone being impacted if the most minor board outline change occurred.

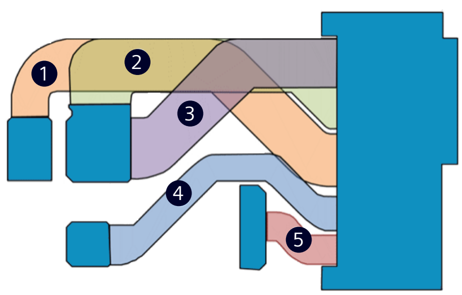
Multiple outlines



Multiple zones



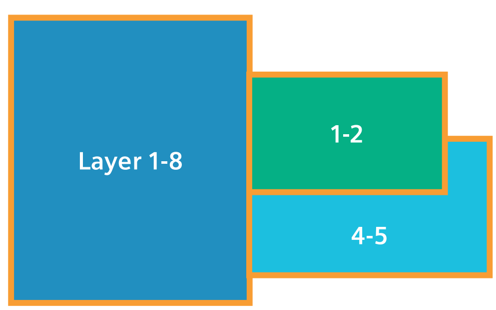
Now consider this design where separate board outlines are used.



By using individual board outlines to define the rigid flex design, we assign stackups per board outline to make managing complex flex structures (as in overlapping designs) easy.

Board outlines must have at least one side in common with at least one other board outline. Overlapping outlines may not have layers in common.

OK



STEP 4

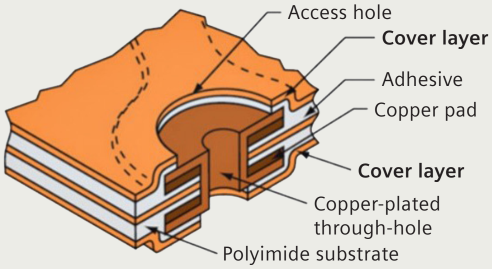
Assign subsets of layers

Details on special flex layers

Flex design includes many special layer types. Before we continue the design, we need to review these and how they are managed/defined using Xpedition Layout:

- Cover layer (sometimes called cover sheet or cover coat depending on what material system is used)
- Adhesive
- Stiffener

We dedicate layers in the stackup for these layers, but additional data can be put on these layers:



Cover layer

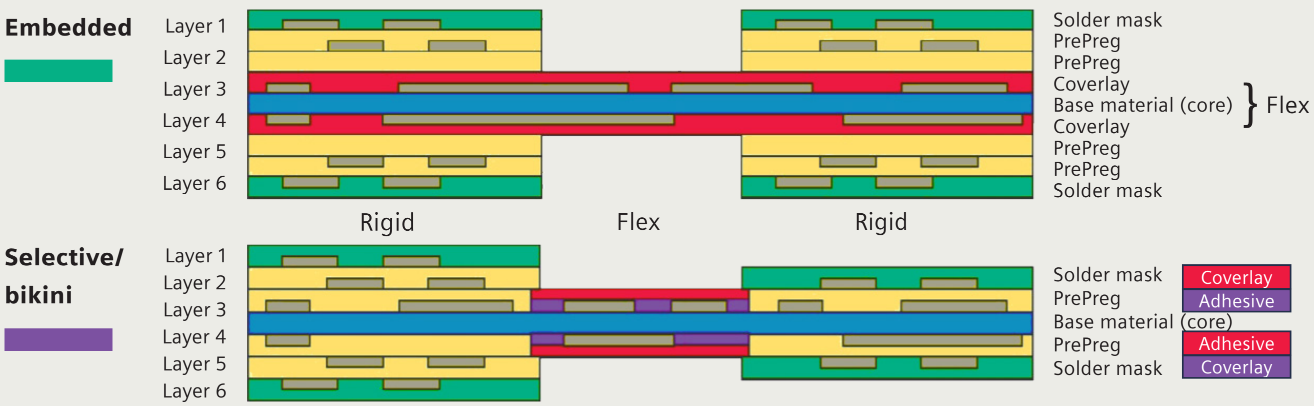
Cover layer is a sheet covering the entire outline. Openings are required in the cover layer to access solder pads for assembly/soldering. Component pad and via openings similar to solder mask are defined in the pad stack.

Custom cover layer openings can be put in the design and in a component cell.

You create a "partial cover layer" also known as "bikini cover layer" by only including the cover layer in the flexible board stackup and put a corresponding rigid prepreg style material in the corresponding rigid board.

Selective/partial cover layer

Cover layers can extend completely through the entire circuit (embedded) or just over the flex portion (selective/bikini). To define a partial style cover layer, you simply include cover layer in the board outline where you want it and exclude the cover layer in the board outlines where you don't want it.

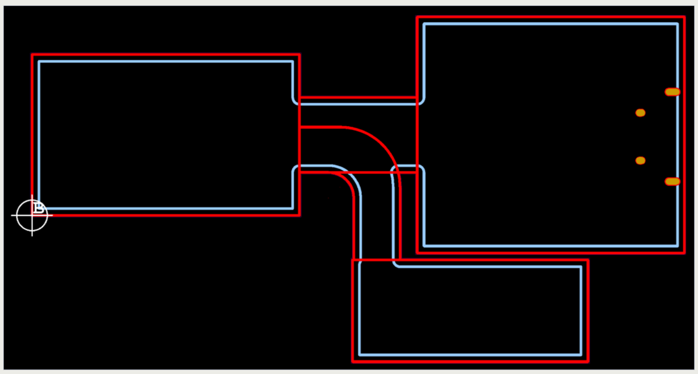


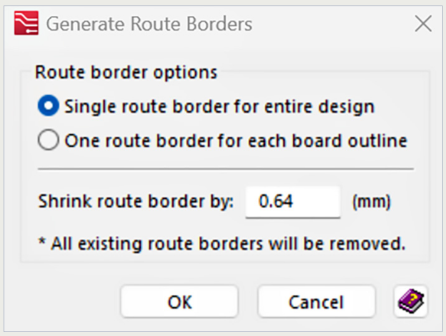
STEP 5

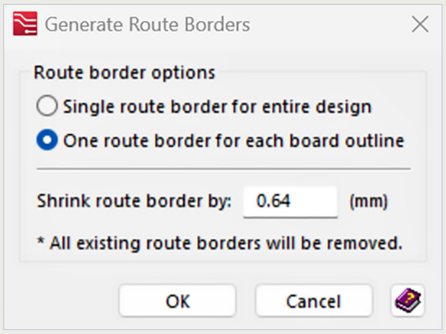
Establish route borders

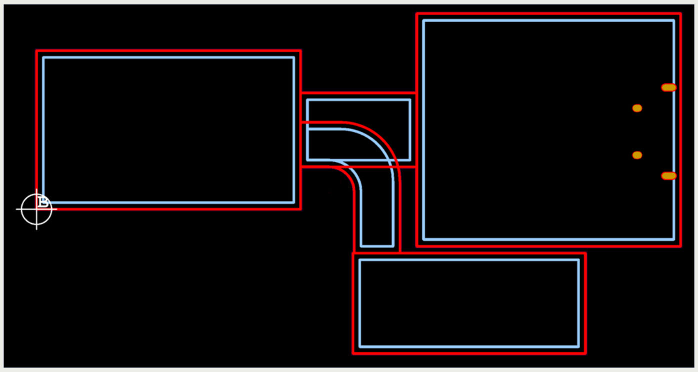
If multiple board outlines have been created, the appropriate route borders must be created per board outline.

The command **Draw > Generate Route Borders** can create the route borders automatically from the board outlines:









In both graphics you can see the red is the board outline and the blue is the route border.

This enables you to choose the option that works best for your situation.

STEP 6

Create bend areas

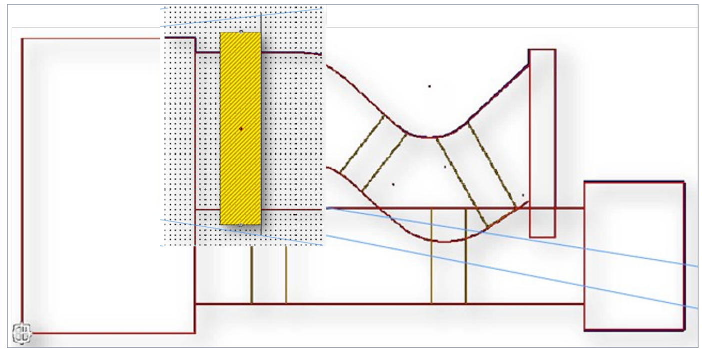
Defining where and how it bends

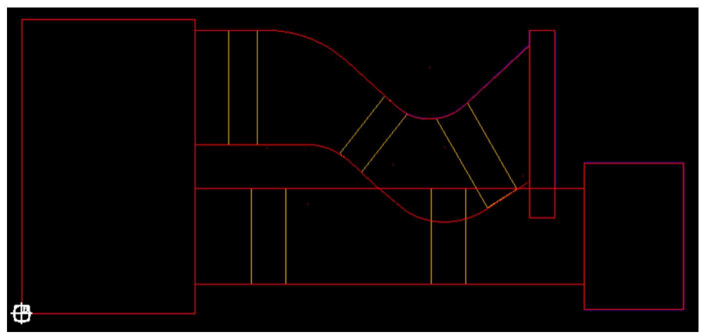
A new design object: Bend area, has been created to let the designer define where the design bends and how it bends. Bend area properties allow you to define what is legal to do inside the region impacted by the bend in terms of traces, vias, parts and plane fill. Bend area properties can be any of the following:

- Radius
- Angle
- Allow corners (trace bends)
- Allow width change (neck up/down)
- Allow non-perp (traces not at right angle of bend)
- Allow vias
- Allow solid fills (requires cross hatch fill if disabled)
- Allow parts
- Bend order (folding sequence)
- Left/right slide distance (dynamic bending animation and DRC
- Binding length (extra length for book binder flex)
- Adhesive
- Stiffener

The bend area is a draw object placed as a line. The bend will happen along this line.

By default, the bend happens along the center of the bend area. The bend area property bend origin = center means bending along the bend area center. Other settings are left or right.





Parameters on the bend area draw object define how the board bends. In the picture to the right, we have placed four bend areas shown in yellow.

The width of the shape indicates the region that is impacted by the bend. It is thus a function of bend radius and bend angle.

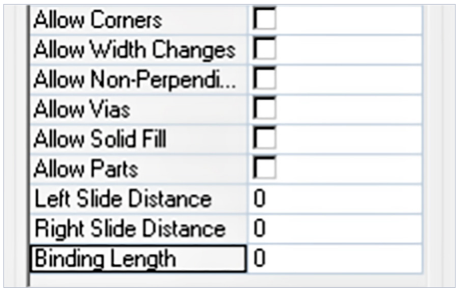
For a full 360-degree bend, the width of the bend area is π*2*Radius (length of circle circumference formula).

A user doesn't do this calculation. You simply enter radius and bend angle, and Xpedition creates the appropriate bend region for you!

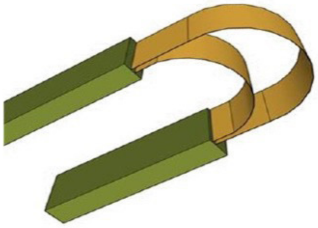
Bend area properties explained

We will now review the **Bend Parameters Allow Corners to Binding Length**.

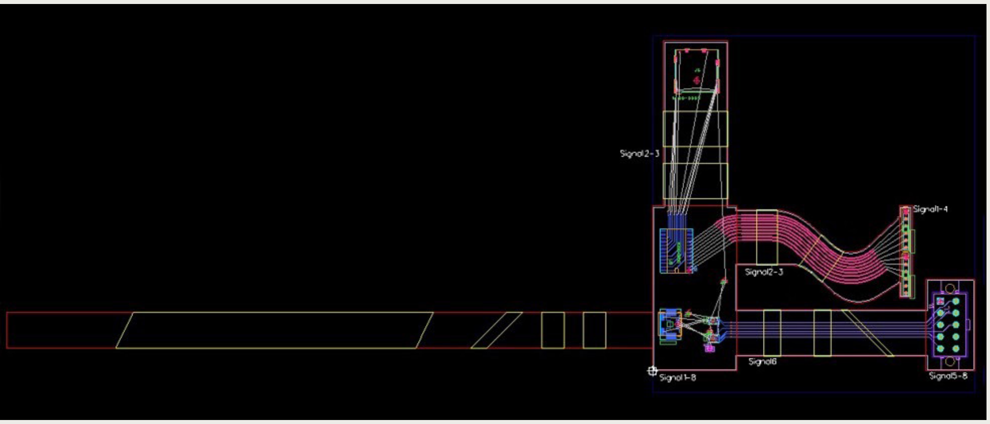
- **Allow corners:** Defines if a trace may have a vertex within the bend area
- **Allow width changes:** Defines if a trace may change width within the bend area
- **Allow non-perpendicular:** Defines if traces may cross the bend line in other than exact 90-degree angles
- **Allow vias:** Defines if vias are legal inside the bend area
- **Allow solid fill:** Defines if area fill may be filled solid (as opposed to hatch/cross-hatched) inside the bend area
- **Allow parts:** Defines if parts may be placed inside the bend area
- **Left slide distance:** Future use; to define dynamic flex for DRC and 3D

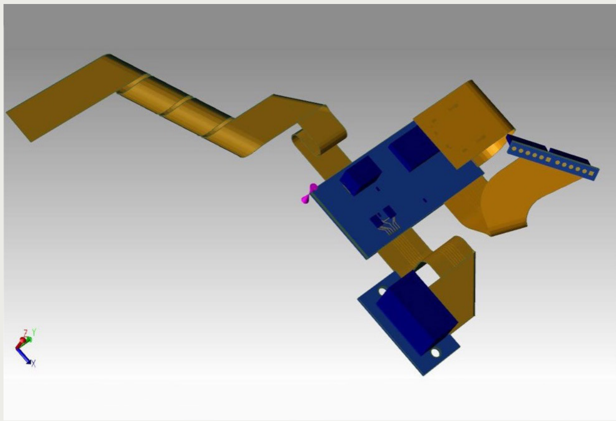


- **Right slide distance:** Future use to define dynamic flex for DRC and 3D
- **Binding length:** Future use for book binder flex extension length. Binding length is this extra added length to the center of the bend area



Here is an example of a design with multiple flex regions with various bend areas shown in 2D and in 3D bent state:



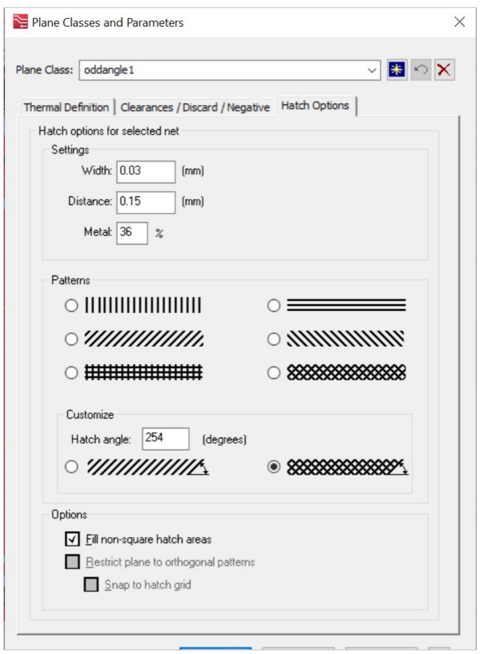


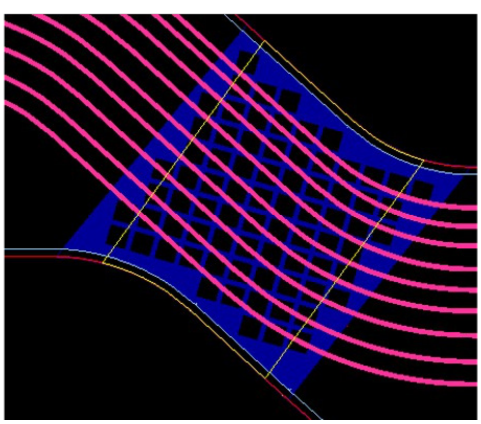
Plane fill for flex

You typically want to avoid solid plane fill in bends as solid fill would increase metal strain.

The solution is to use a cross-hatched pattern. However, to provide the best reliability in the bend, you want the cross-hatch to be rotated to be exactly 45 degrees off to the bend line at least in the region impacted by the bend.

To find the correct angle, you have to measure the angle of the bend line and add/subtract 45 degrees to it and enter it in the plane classes and parameters dialogue.





Adding odd angle cross-hatch plane fill

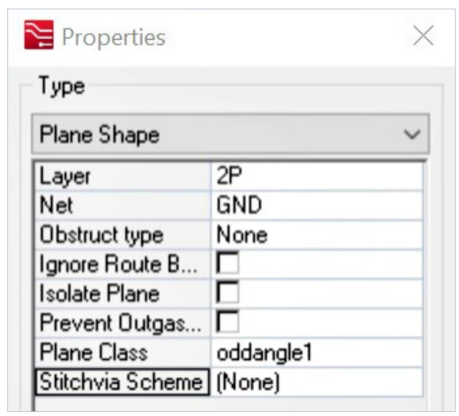
Draw the outer boundary of your plane fill.

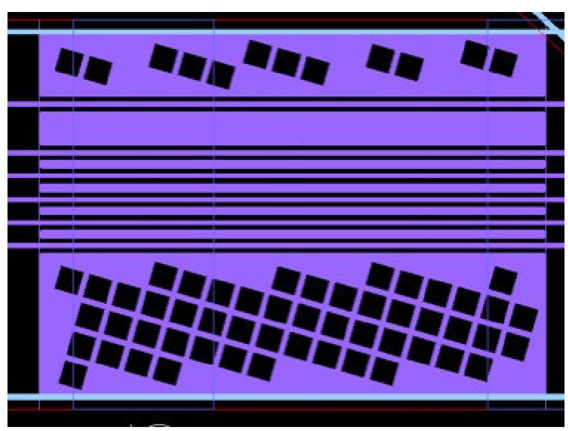
The **Net** that is associated with the plane fill is set as a property on the plane shape object. This is also how you can define which **layer** it should be on.

You assign a **plane class** to the plane shape.

You create and edit this plane class in **Planes > Plane Classes and Parameters**... dialog. This is where you define the plane to be cross-hatched and which cross-hatch angle to use.

All plane classes defined in this dialogue are available in the plane shape properties dialog.





STEP 7

Place and route as usual

Routing

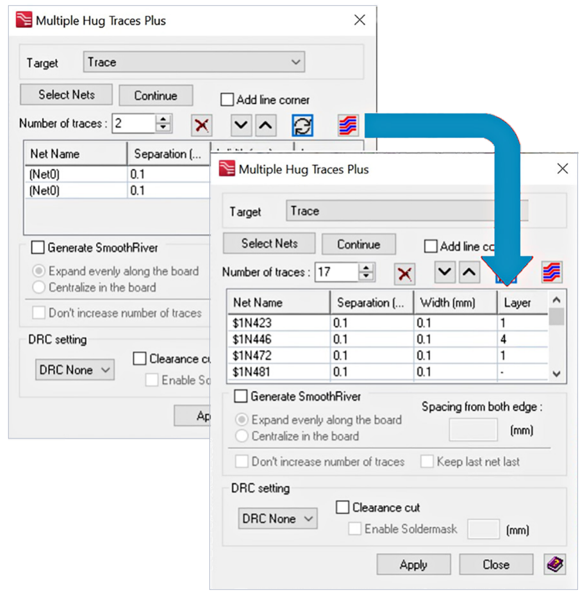
Routing on the rigid panel is no different from routing an ordinary PCB.

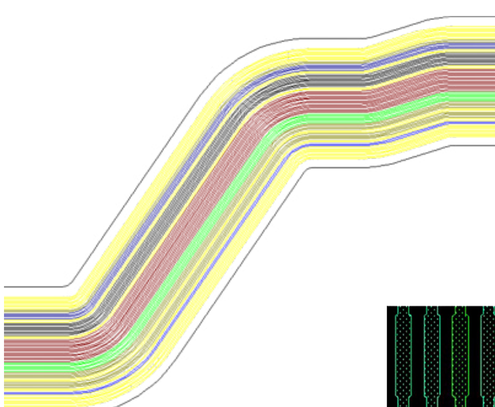
The challenge in routing the flexible board sections lies in the often complex board shapes that restrict routing channels and often require routing of busses on flex cables that follows the shape of the curving flex cable and then smoothly breaks up the bus as it continues into rigid sections or flex sections with part placement. In addition, there are recommendations and rules for how to route in the bend zone for reliability reasons.

Smart Utilities includes several features that add efficiency and controllability to routing of a flex PCB.

Note: Smart Utilities (SU) is not installed by default so please check your Xpedition Install and install the Smart Utilities if not present on your system.

We will review two of the Smart utilities for flex but please see the Smart Utilities documentation for a complete reference as there are many useful utilities.



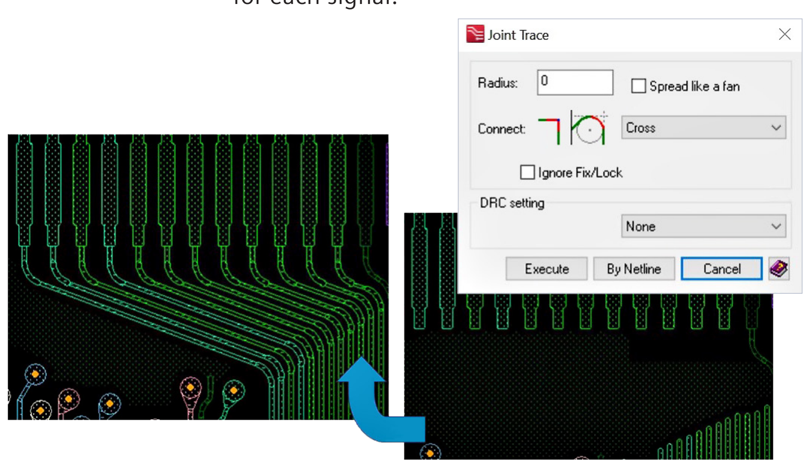


Routing busses along curving flex cables

Use the **Multiple Hug Trace Plus** utility from Smart Utilities to select, set up and implement such routes. This utility can even prevent I-beaming on multilayer flex.

Routing from a flex cable bus to a connector or component

Use the **Joint Trace** utility from Smart Utilities to quickly connect a pre routed bus to a connector or similar. Use the **Spread like a fan** option to have concentric curves with incrementally larger radius for each signal.



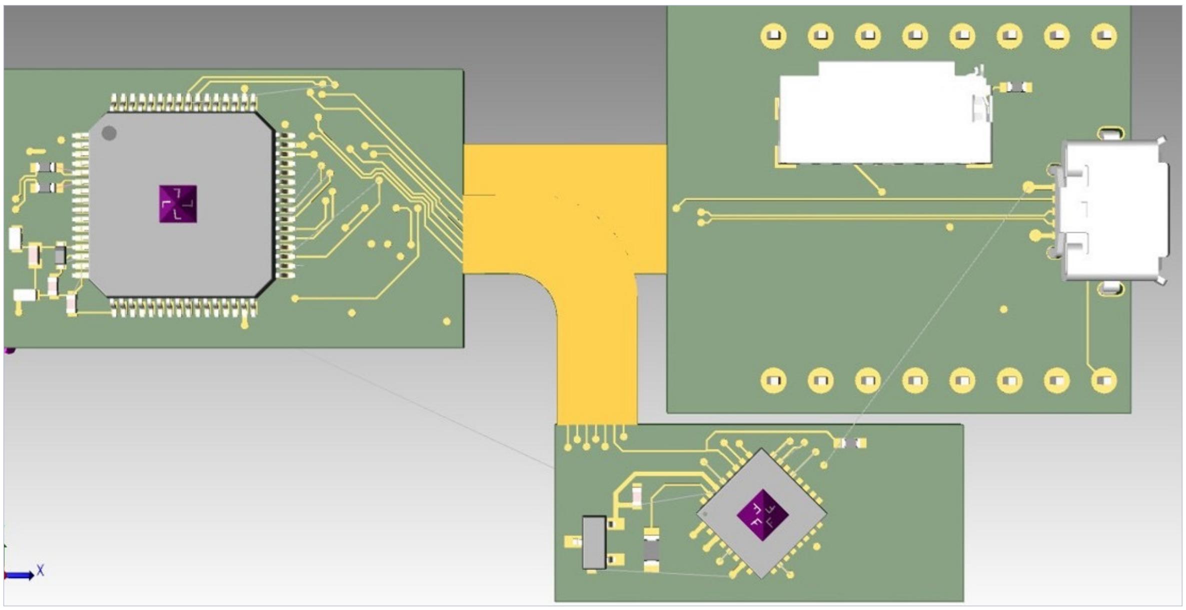
STEP 8

Display in 3D if desired

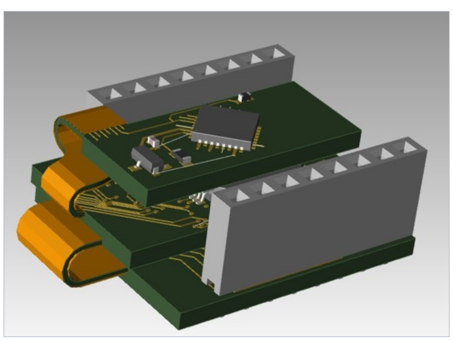
3D design

In PCB 3D view of the design, you have the choice of viewing the design in flat mode or in folded state. The bend areas with their parameters define how the folding is done.

This picture shows the flat view of our sample design:



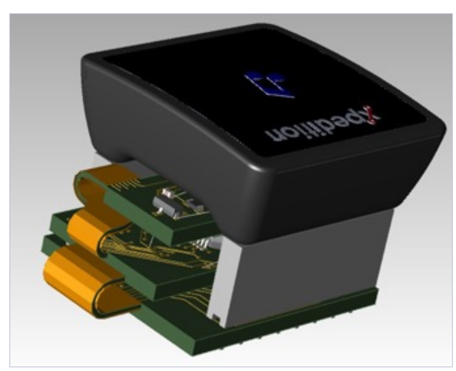
Selecting 3D Flex Bend gives this:



You can also set color and display for cover layers, stiffeners etc. Z-axis zoom is not available in bent state for obvious reasons. Furthermore, we can import mechanical models for the mechanical display and for the enclosure.

The mechanical computer-aided design (CAD) files for display and enclosure are included in the sample design for you to import.

Design rule checking (DRC) and measure in the folded 3D view as well as move components – same as in the flat view.



STEP 9

Run appropriate DRC checks

Design rule checking for flex and rigid-flex

The Xpedition design flow provides DRC in three tiers:

- Correct by design –or online preventive DRC
- Batch DRC
- Online section of the hazards explorer

Correct by design DRC

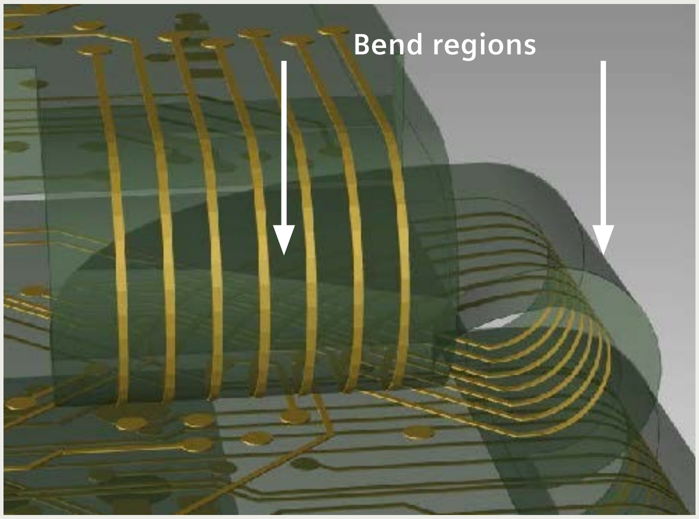
Correct by design includes:

Parts too close to a bend area. Applicable to bend area property: Allow parts option

- Bend area to:
  - – Placement outlines
  - – Part pads through hole and SMD

Illegal holes in Bend Area. Applicable to Bend Area Property: Allow Vias option

- Bend area to:
  - – Via holes
  - – Part holes
  - – Mounting holes

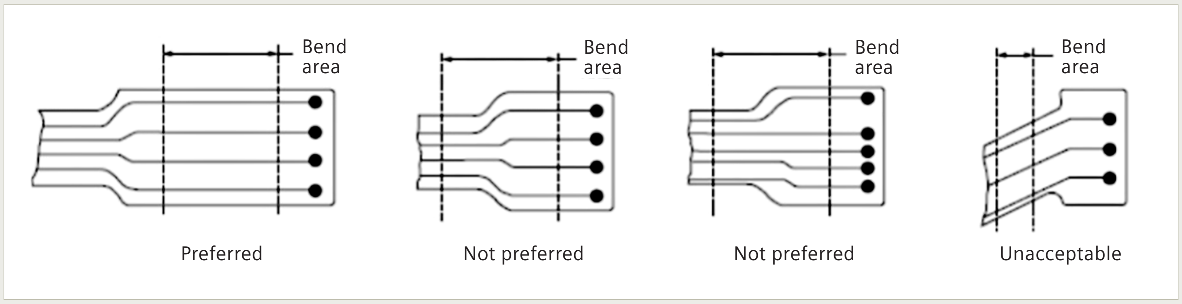


Illegal traces:

- Traces not perpendicular to bend line
- Trace bends in the bend area
- Trace width changes in the bend area

Plane violations:

- Solid fill when in conflict with bend area setting
- Cross-hatch angle is not 45 degree of bend line

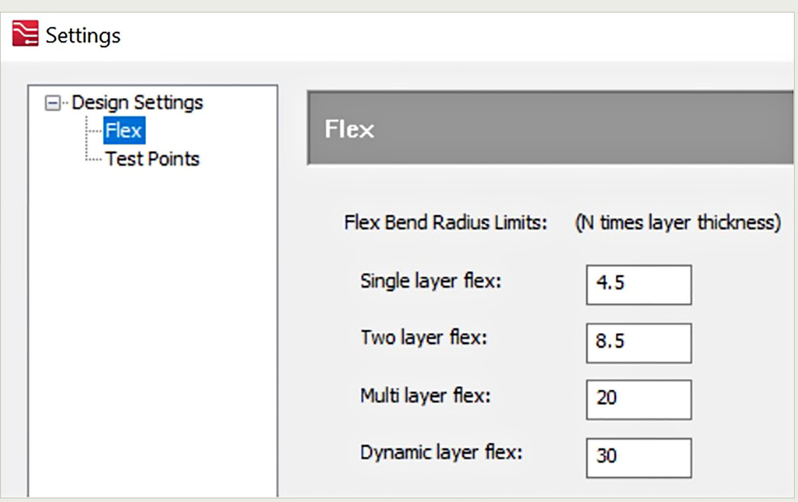


Source: IPC-2223

Online section of hazards explorer DRC

Online section of hazards explorer includes:

- Traces not perpendicular to the bend line. Applicable to bend area property: allow non-perpendicular
- Trace bends in a bend area. Applicable to bend area property: allow corners option
- Trace width changes in bend area. Applicable to bend area property: allow width change option
- Board outlines must share at least one coincident edge with at least one other board outline
- Bend radius DRC: The minimum rules for bend radius as defined in **Setup > Settings**... is reported here. In **Setup > Settings**... under "Flex," you define the acceptable minimum radius as multiples of layer thickness of the bending flex. Different multiples are defined for different types of flex (single layer, two layer, multi layer and dynamic flex)

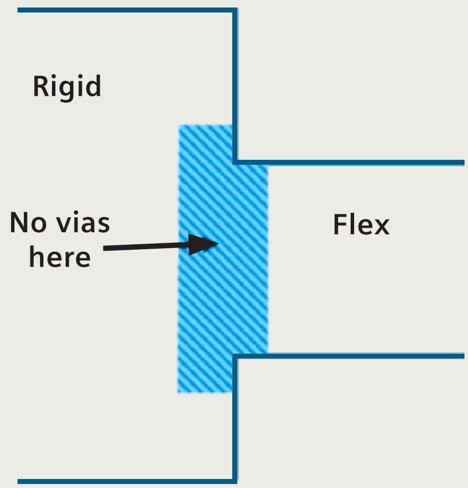


DRC of vias and parts placed too close to beginning of flex

Especially with partial/bikini style cover layer, you should not allow vias or other holes close to where the flex or cover layer starts as shown in this sketch:

The rules and shape of this "off limit" region can be complex.

For maximum flexibility, this off limit zone is created by the designer using a **via obstruct** and **placement obstruct**. This will then be checked in correct by design preventive DRC as well as in batch DRC.



STEP 10

Create outputs

Fabrication output ODB++

Flex items are using the flex specific layer types in ODB++.

However, holes in stiffeners are output as stiffener pads on a documentation layer.

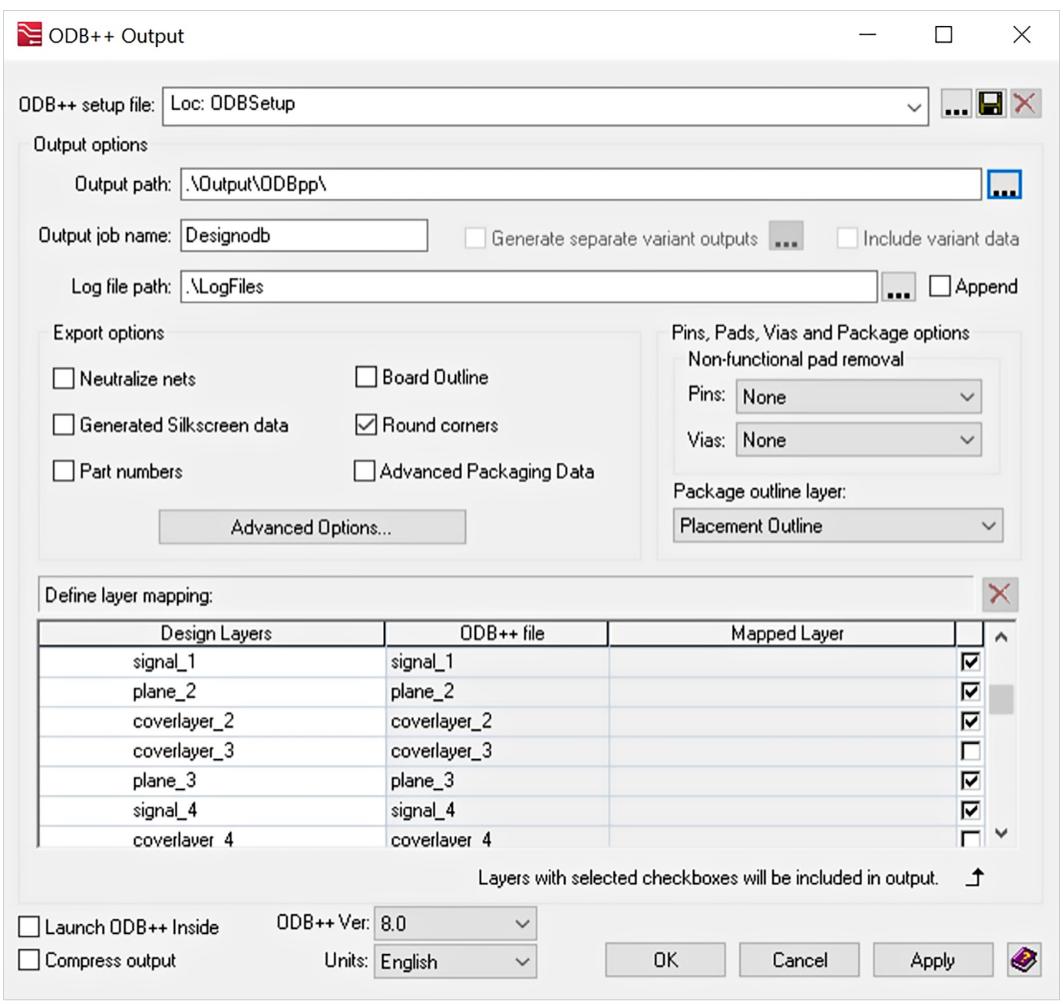
Pad diameter represents stiffener hole diameter and the board fabricator needs to extract this data for the stiffener drill operation.

ODB++ generation

When you are ready to generate ODB++, use Output > ODB++... to open the ODB++ generation dialog.

Select OCB++ version 8. This sets the output format to version 8 that has the required flex design data types.

Also review the layer mapping section and make sure the desired flex layers are mapped. This includes cover layers, adhesive layers, stiffeners, etc. Click **OK** to generate output.

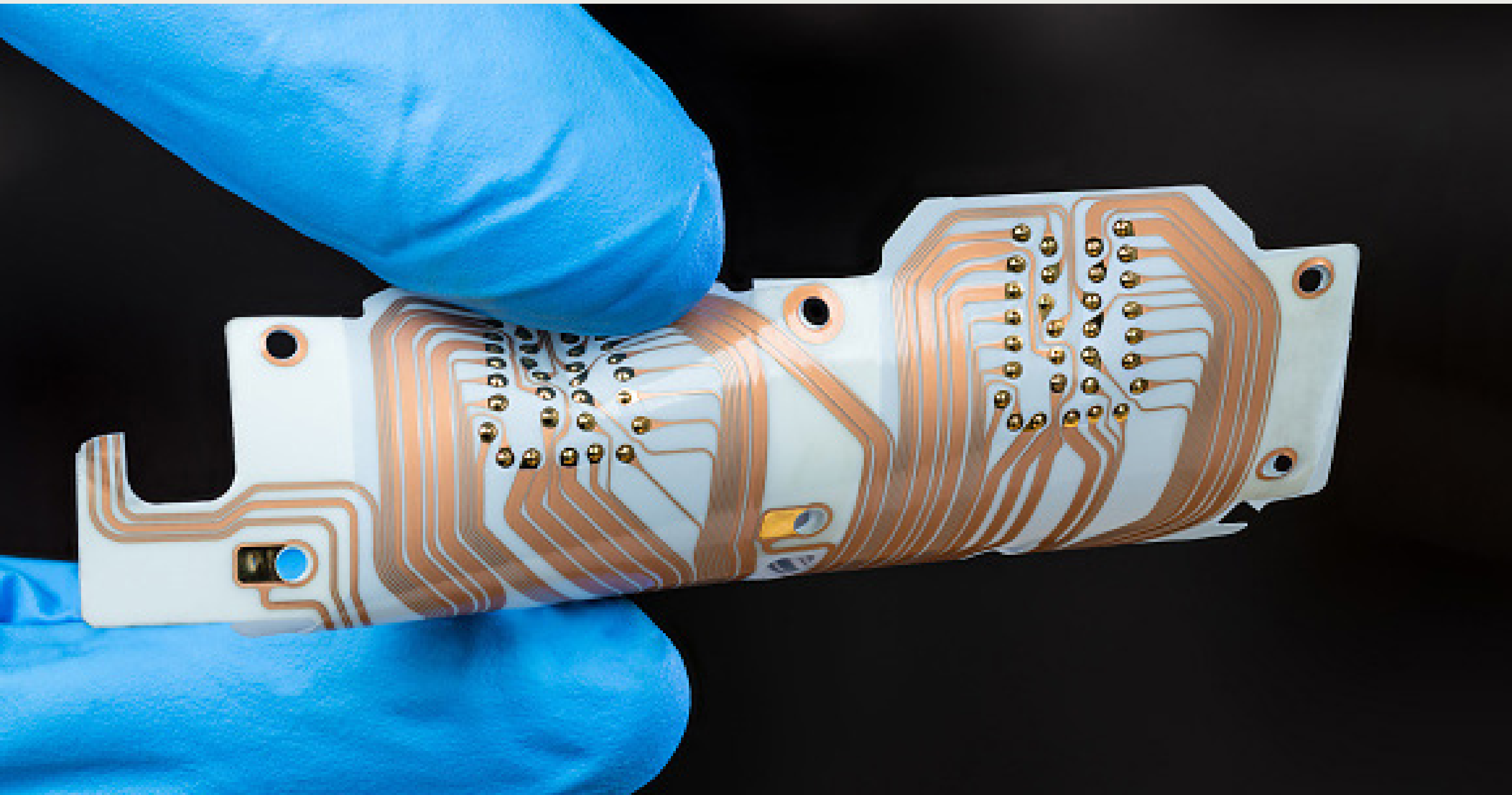


Conclusion

At this point in the process, we have the fabrication data for our rigid- flex design, and this last step concludes our start-to-finish, step-by-step guide.

The simple 10-point design process presented in this e-book serves as a structured roadmap, guiding you through each critical phase of rigid-flex PCB development. Whether you are dealing with flex-only designs or complex flex-rigid configurations, the principles and guidelines outlined here are designed to equip you with the knowledge and skills needed to navigate the challenges and intricacies of rigid-flex PCB design with confidence.

Remember, mastering rigid-flex design is a continual learning process. As you apply these principles and techniques in your projects, you'll undoubtedly encounter unique scenarios, complexities and opportunities for innovation. Embrace these challenges, seek continual learning and leverage the insights gained from this e-book to push the boundaries of what's possible in rigid-flex PCB design.



Siemens Digital Industries Software helps organizations of all sizes digitally transform using software, hardware and services from the Siemens Xcelerator business platform. Siemens' software and the comprehensive digital twin enable companies to optimize their design, engineering and manufacturing processes to turn today's ideas into the sustainable products of the future. From chips to entire systems, from product to process, across all industries, Siemens Digital Industries Software – Accelerating transformation.

Americas: 1 800 498 5351

EMEA: 00 800 70002222

Asia-Pacific: 001 800 03061910

For additional numbers, click here.

85955-D2 4/24 C