Flexible PCB Automation (FP9)

BPA Delivery 7 for V5R19 (V5.7)

User Guide

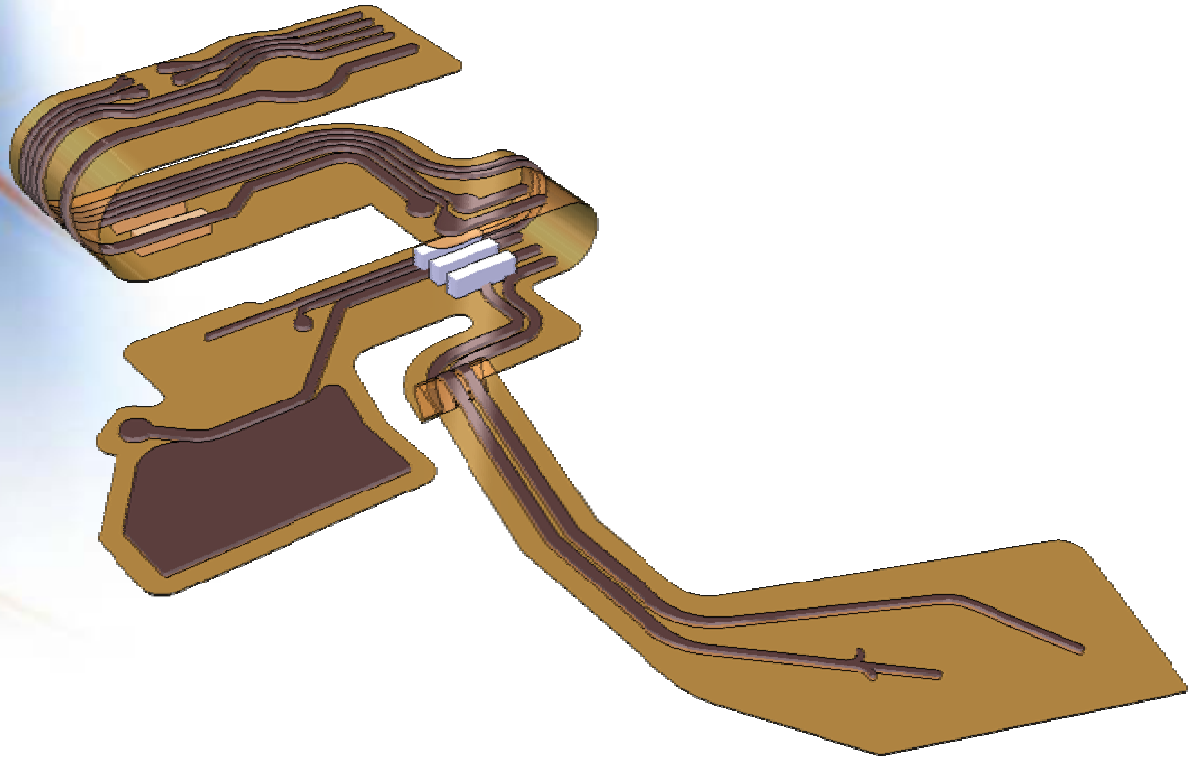


Table of Contents

Copyright Notice

This guide is delivered subject to the following conditions and restrictions:

About Printed Circuit Board (PCB) Design

Main Rules Reminder

Electrical design is mostly done in external ECAD system. This system needs to be integrated to Mechanical design, so that Mechanical design can use the space reservation of electrical data and Electrical designers can use circuit board shape that has been designed in CATIA.

For each set of data a maintaining system must be decided. This allows making changes to data and using that data as a reference in another system. In this design processes this should be divided as following:

	MCAD (CATIA V5)	ECAD
PCB outline	X	
PCB constraint areas	X	
Component shape		X
Component existence		X
Component position		X
Panel layout		X

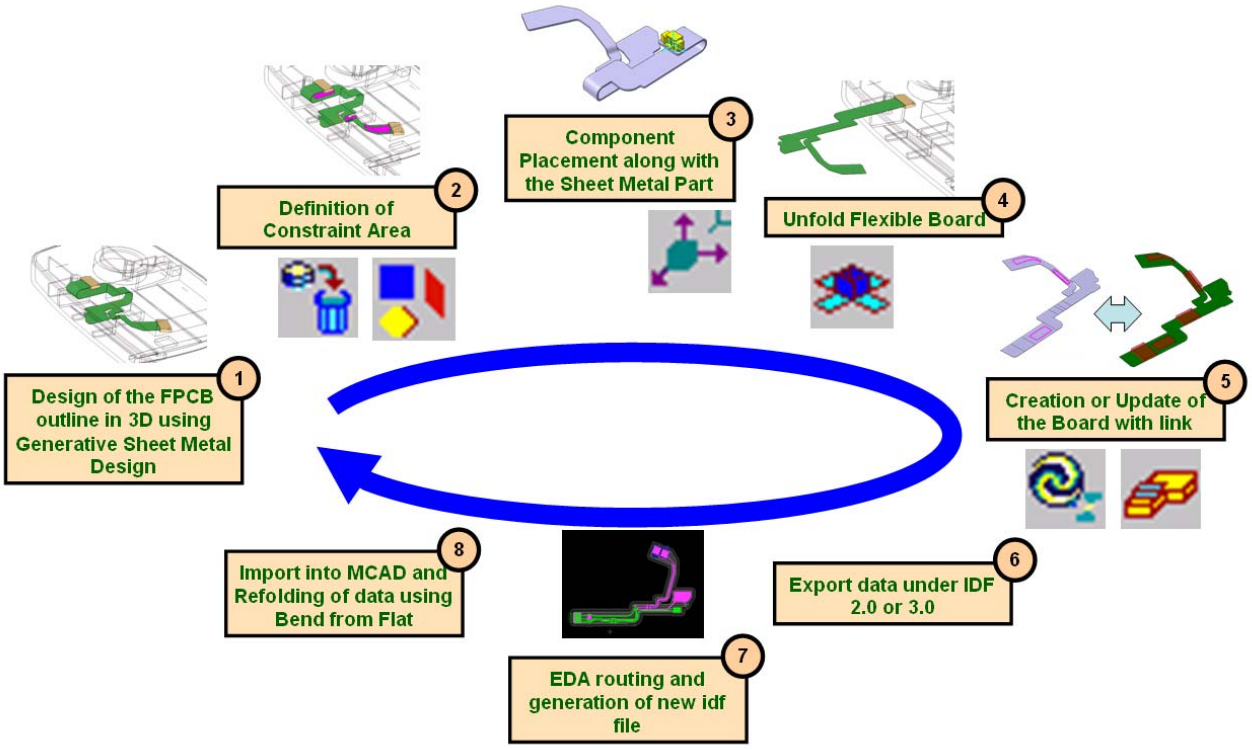
PCB board outline and Constraint Areas are created and updated in CATIA.
All updates are sent to ECAD. Because library files cannot be exported, all area and component space reservations must be done by Constraint Areas.

Components are created and updated in ECAD and all updates send to CATIA. Only some component modifications shall be made at CATIA and updated to ECAD.

WORKFLOW Reminder

1. Hardware assembly contains Flexible PCB board, Mechanical components divided into Top and Bottom side subassemblies and component subassembly from ECAD
2. Flexible PCB board outline and Constraint Areas are created by Mechanical Design with CATIA. This means that PCB board should always be created in CATIA and maintained there, it should never be imported from ECAD system.
3. Critical Components area placed in 3D together with constraint area
4. Flexible PCB board is unfolded in CATIA.
5. A model of the FPCB is created and structured. It is ready for export and drafting creation
6. Flexible PCB is translated to ECAD via IDF file.
7. All electrical components are created and placed in ECAD

8. All modifications are translated back to ECAD by IDF file.

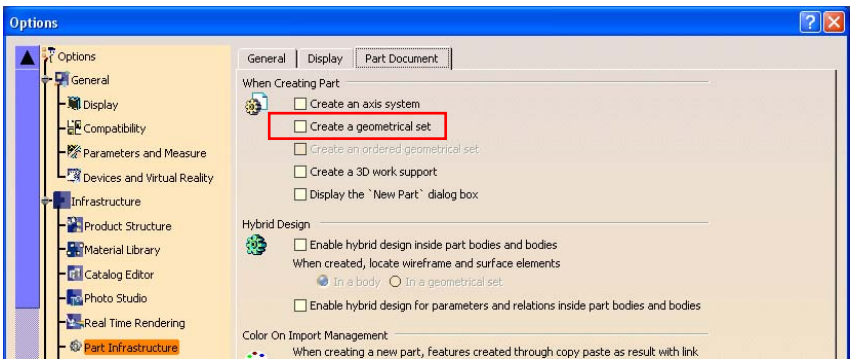


General Recommendation

The recommended methodology to create a flexible PCB part is to use Generative Sheet Metal Workbench. Once the flexible PCB shape is done, this model is to be sent to an Electrical Engineer.

Based on Generative Sheet Metal Design, Circuit Board Design workbenches and automation tools, this Flexible PCB workbench provides a way on how to populate a “Sheet Metal” part with electrical constraints and how to make it “ready for any EDA system via the IDF (Intermediate Data Format) file.

To be able to create a “sheet metal feature” in the Flexible PCB workbench, the activation of the option “create a geometrical set must be set to OFF.



Flexible PCB deliverables

The Flexible PCB deliverables are provided as an executable package including the following functions:

1. Constraint Area 

   Creation of 3D zones to help the electronics engineer for the FPCB Layout: Where are the zones for components? Where are the zones forbidden to components? Where are the bend (flexible) areas?

2. Fold/Unfold Board 

   Flatten and fold a master sheet metal part, attached components and constraint areas.

3. Component Placement 

   Attach a mechanical component to a flexible PCB. With this function, the attached components will move along with the sheet metal model's flattening and folding.

4. Generate CBD Part 

   Format the Sheet Metal model into a ready for export flattened board. The specification tree structure allows an easy drafting generation.

5. Update CBD Part 

   Update the master sheet metal model and the formatted model. Both are fully associative through the use of this function.

6. Export As IDF File 

   Generate an IDF files that can be read by any EDA tool.

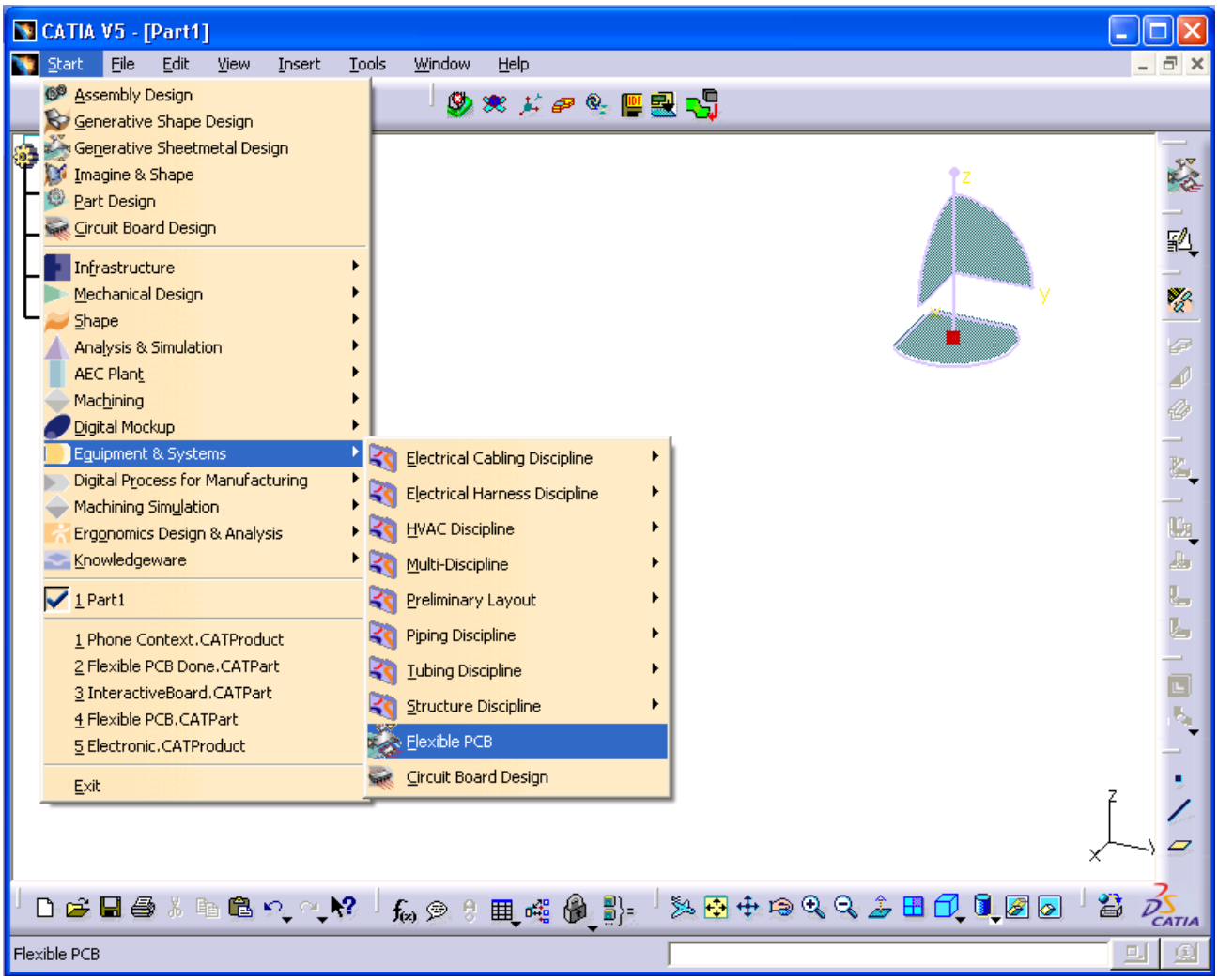
7. Import In Context 

   Import an IDF file and place it at the same location as its flexible PCB master model.

8. Refold Flexible Board 

   Fold the constraint areas and components of the imported IDF file according to the master flexible PCB part.

These capabilities can be access through the use of a CATIA V5 workbench, available and accessible from top menu Start/“Equipment and Systems”:



Generation of Constraint Area

Constraint Area means the communication channel of the specifications between the electronics realm and the mechanical realm. Through them, the mechanical engineering indicates where the areas which are available or forbidden to electronic component placement are located.

Figure 1 is a picture of our starting point. We are working in “part” context, but assembly context is also supported.

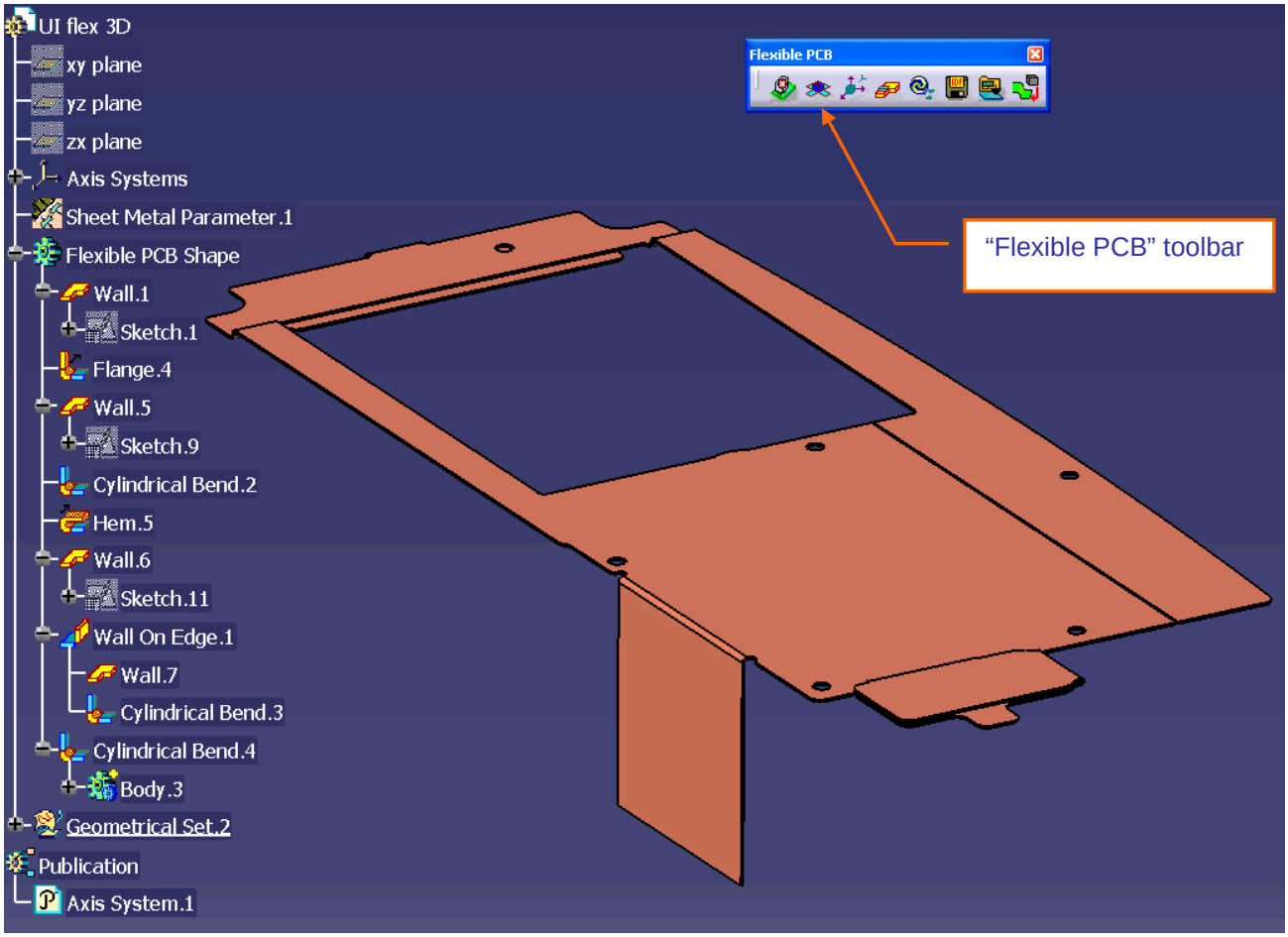


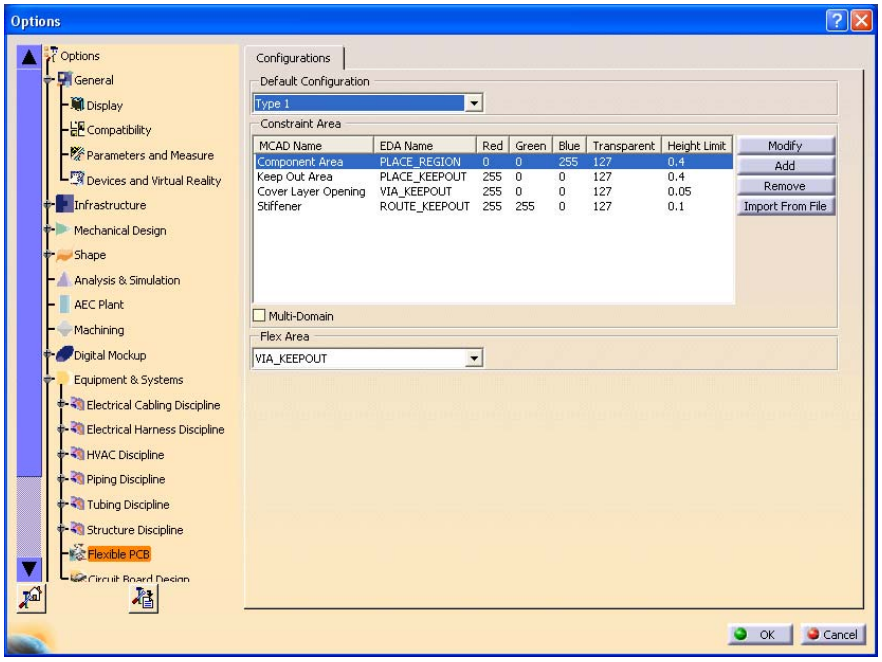
Figure 1

The constraint area will be created and gathered into body “Constraint Area”. The function related to constraint area translation identifies the body called “Constraint Area”, automatically. This body is created on the fly the first time “constraint area function is launched.

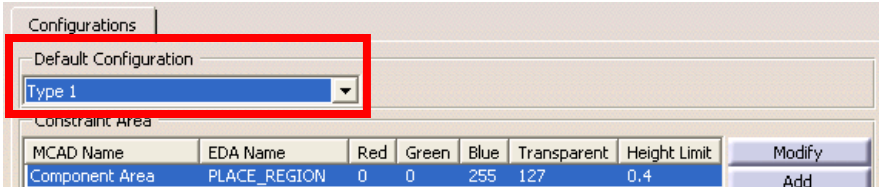
Sketch in body “Constraint Area” or in any other Solid body or Geometrical Set, constraint area profiles in 3D context.

Constraint Areas Default Configurations

Constraint area default configurations are accessible through Tools/Options/Equipment & Systems/Flexible PCB dialog.

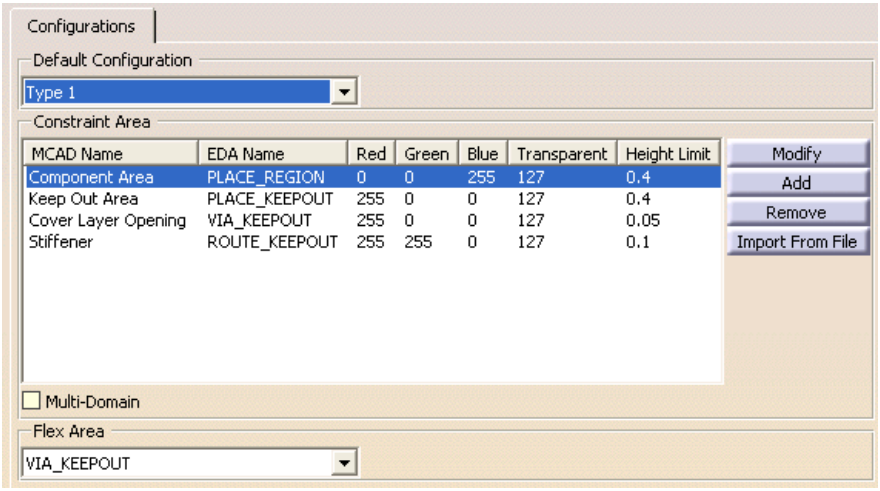


Default Configuration:

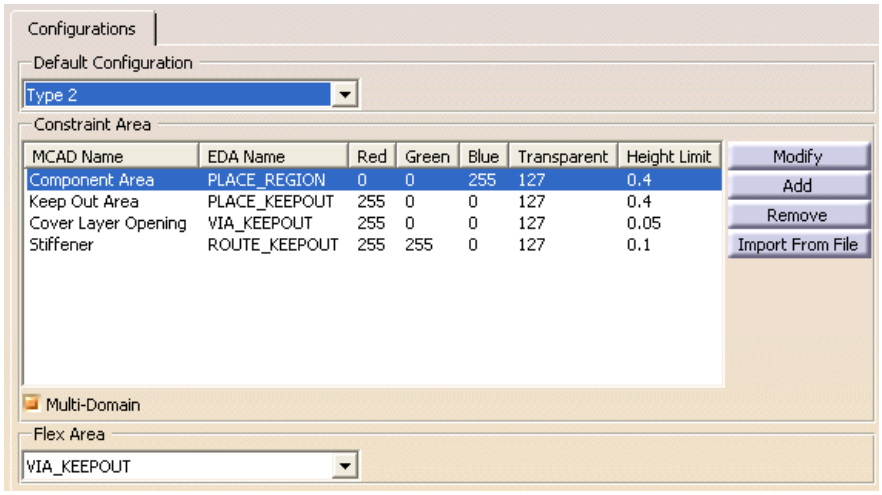


By default, there are 3 different types of constraint area configurations available:

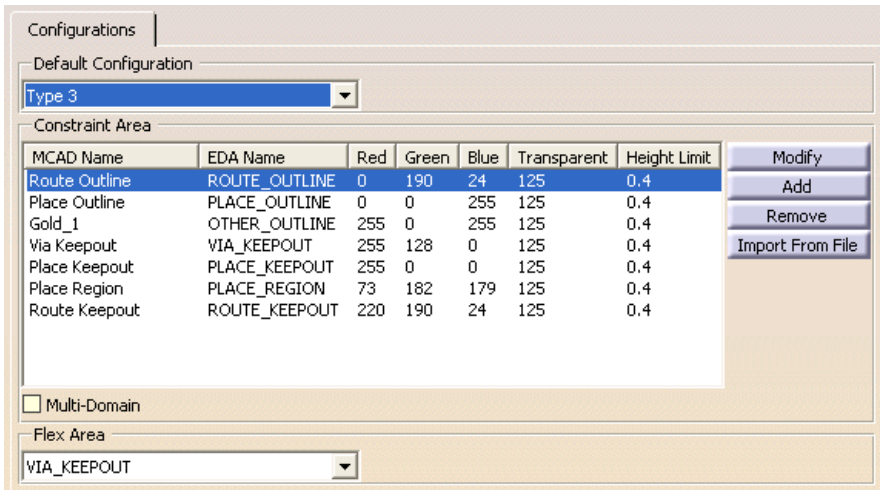
Type 1:



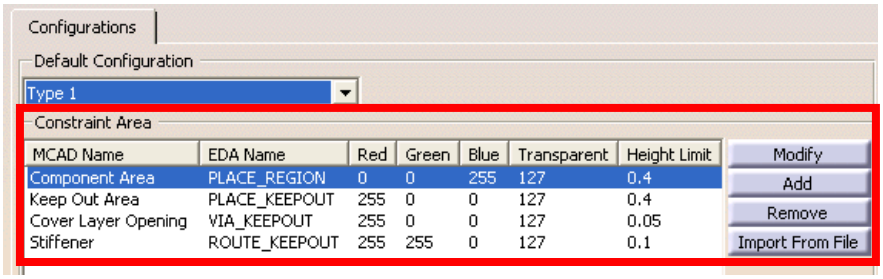
Type 2:



Type 3:

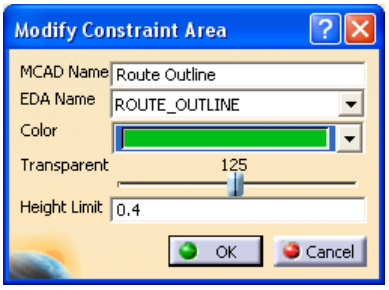


Constraint Area:



- List of available constraint areas

- Modify:

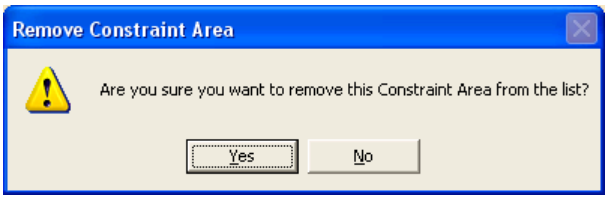


  Modify a selected constraint area type

- Add:

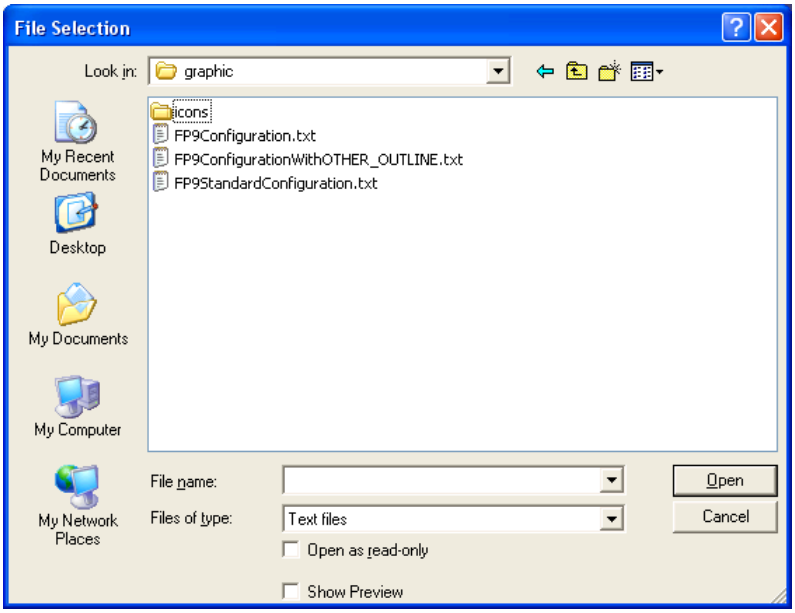
  Add a new constraint area type

- Remove:



  Remove the selected constraint area type

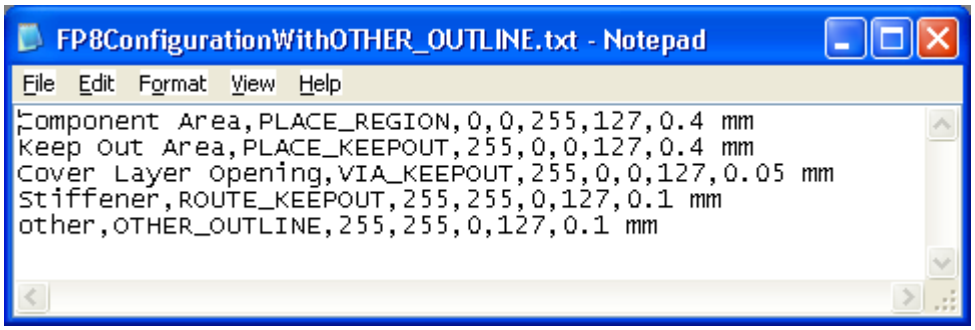
- Import from file:



  Import constraint areas definition from an existing file

Specific configurations files defined prior to Delivery 7 can be imported.

Here is a sample of the format of the text file to be used:



Multi Domain: 

Option of allowing multi-domains sketches for constraint area

Flex Area: 

There are 7 types of constraint area that could be used for flexible areas when generating CBD board:

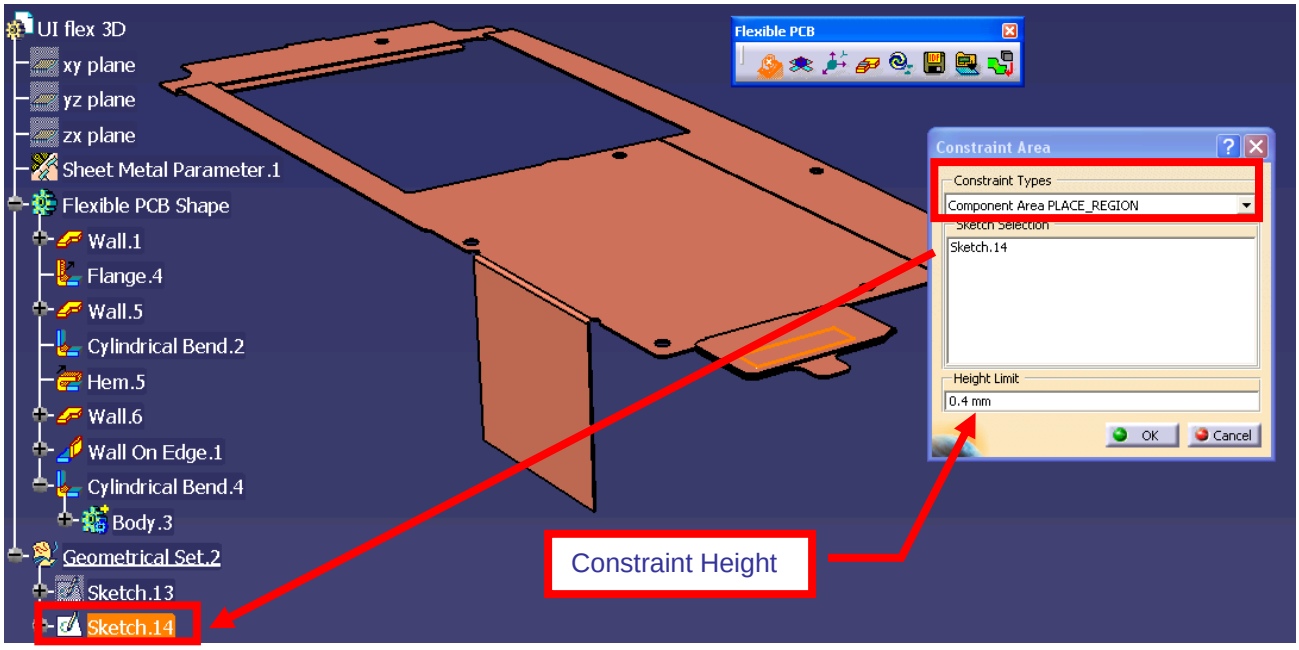
- VIA_KEEPOUT
- PLACE_KEEPOUT
- PLACE_REGION
- ROUTE_KEEPOUT
- ROUTE_OUTLINE
- PLACE_OUTLINE
- OTHER_OUTLINE

Component Area Generation

The creation of any of them follows the same procedure. Here is an explanation for Component Area.

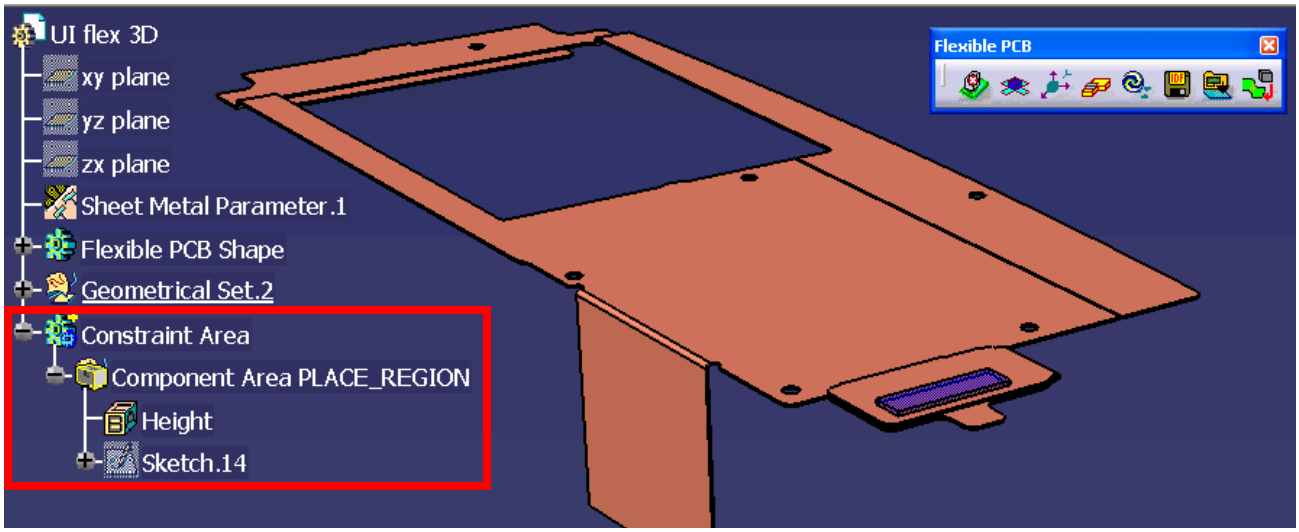
1. Select Function “Component Area” in toolbar “Flexible PCB“

2. Select inputs required to create this area



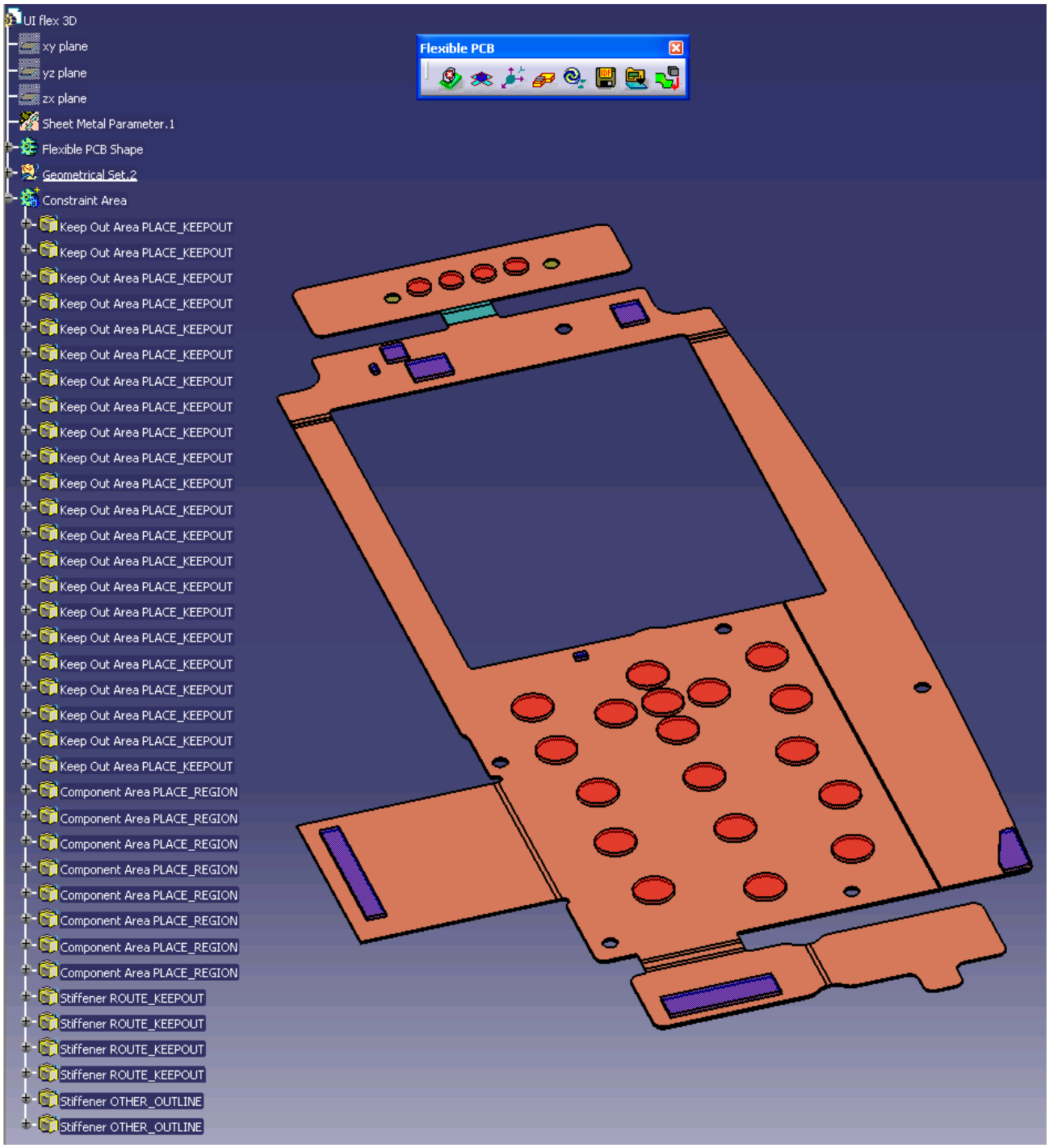
3. If necessary, set another area’s limit and press OK

A body for the Constraint Area’s 3D shape in the folded view and unfolded configuration is created.



All the constraint areas are gathered automatically in the body “Constraint Area”.

After all constraint areas creation, the Constraint Aera body will gather all constraint areas definition.



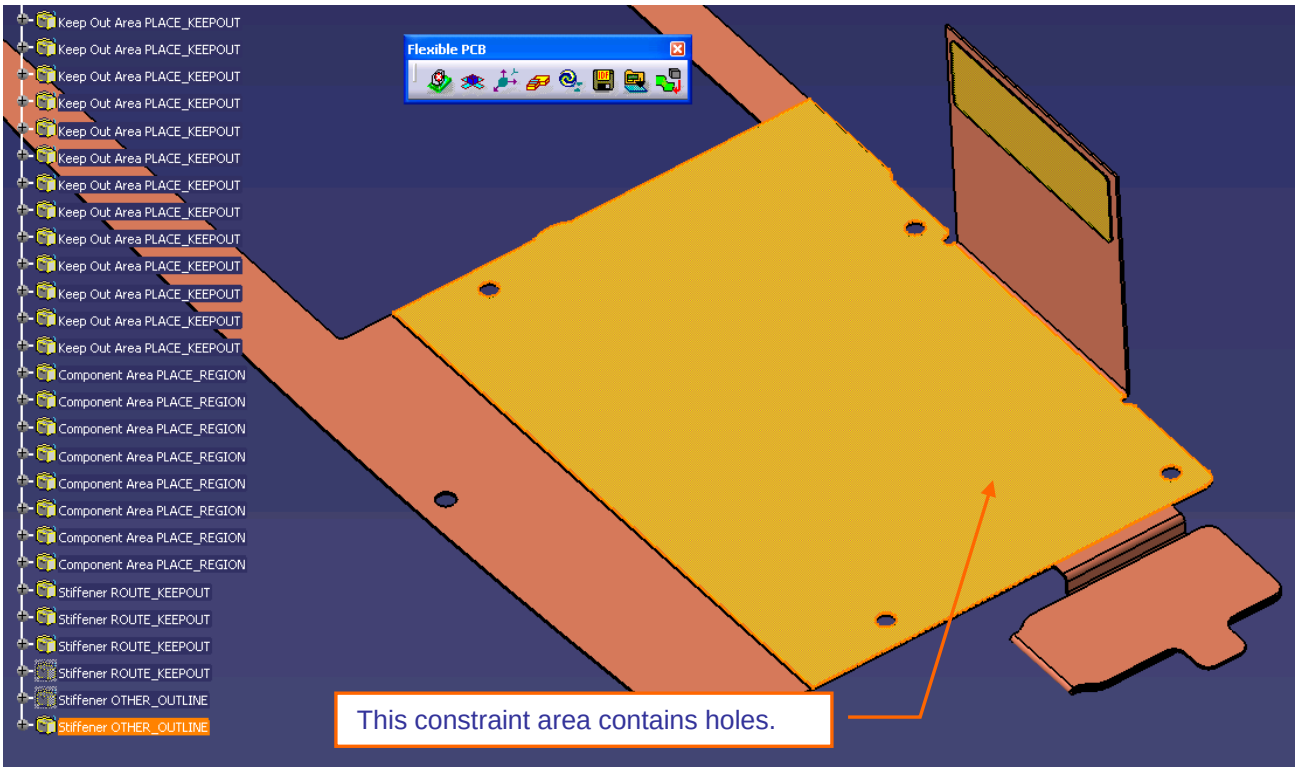
The selected sketch must lie within the flexible PCB’s outline

Selection of isolated sketches is forbidden

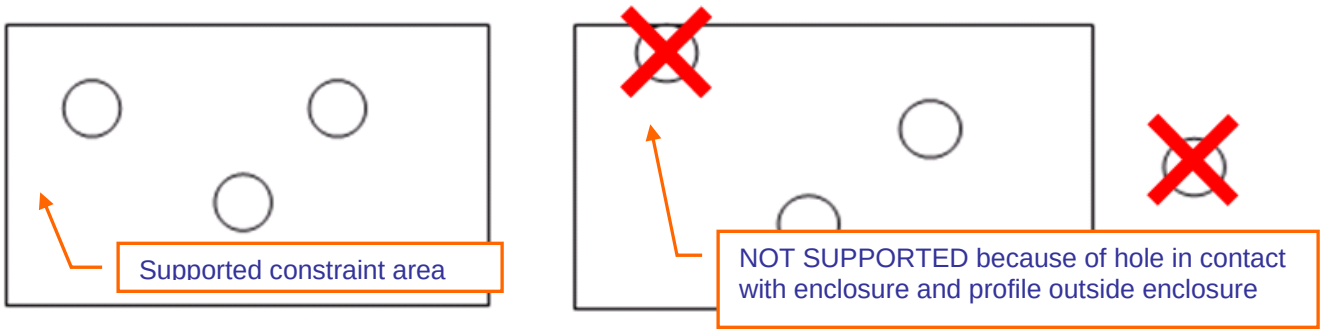
Selection of closed profile sketches with holes allowed if the multi-domai option is set in the configuration:

Multi Domain: 

By default, only configuration Type 2 does come with this option selected.



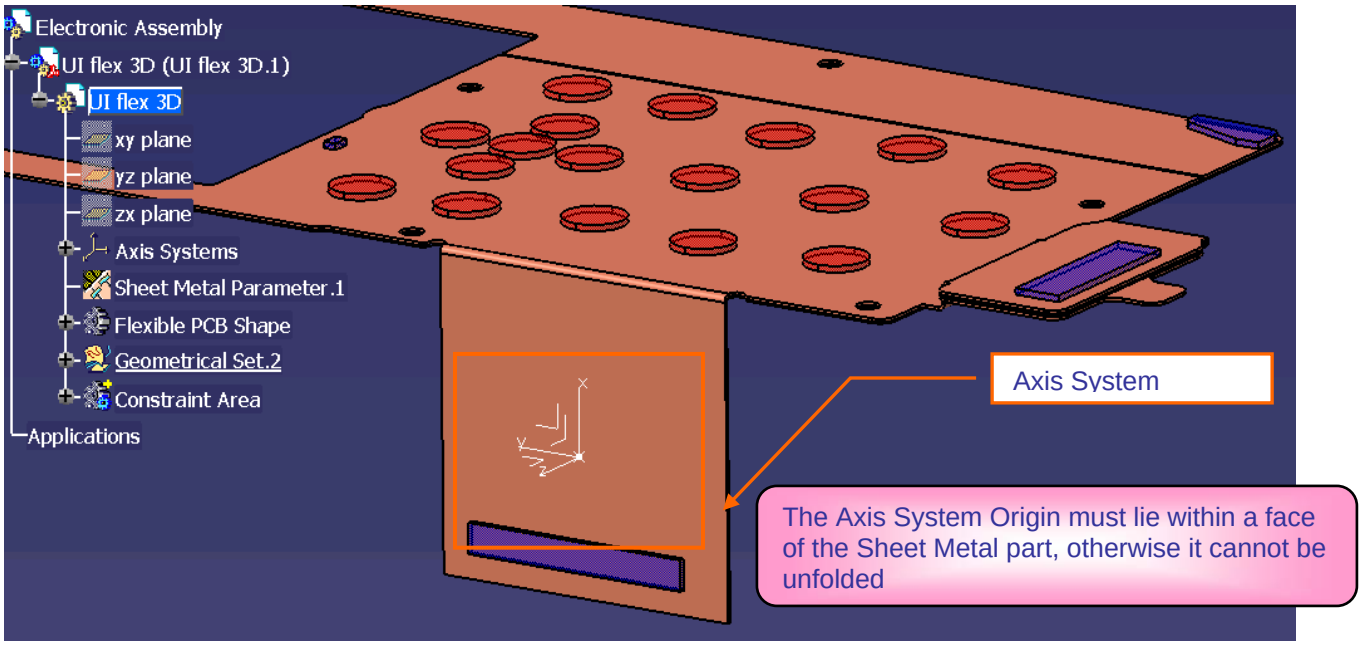
For all types of constraint, IDF format supports closed enclosure with holes. Those internal contours must not have a contact with the outer outline.



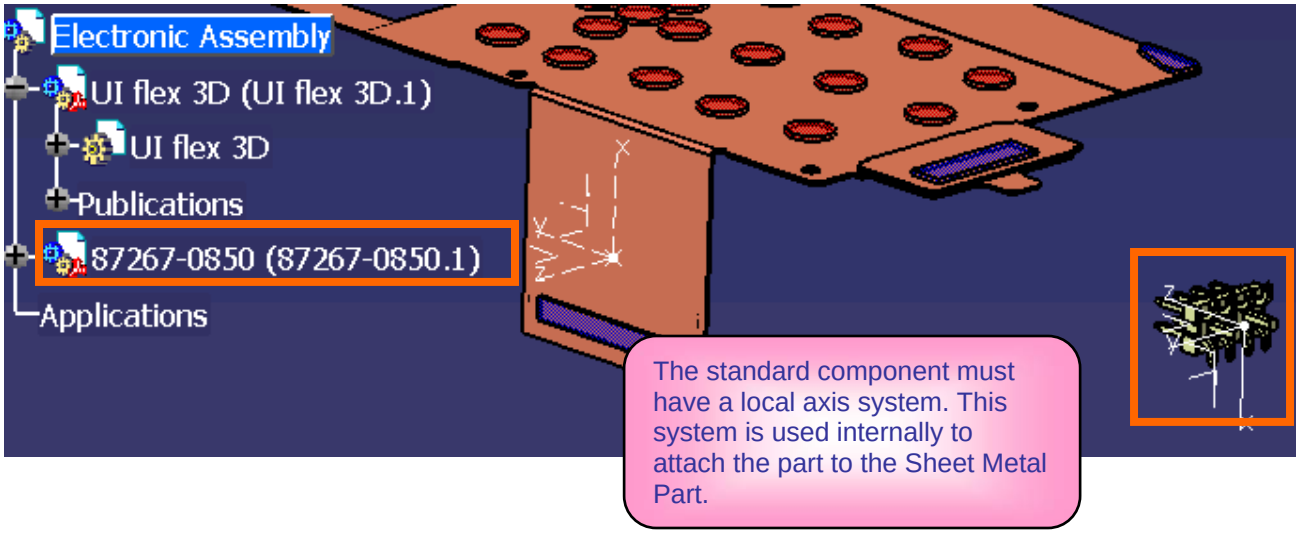
Placing Standard Component with Axis Systems

1. Create an Axis System in the Sheet Metal master part

This axis system corresponds to the location of the standard part to be instantiated later. XY plane is the base plane and Z axis define the vertical direction.

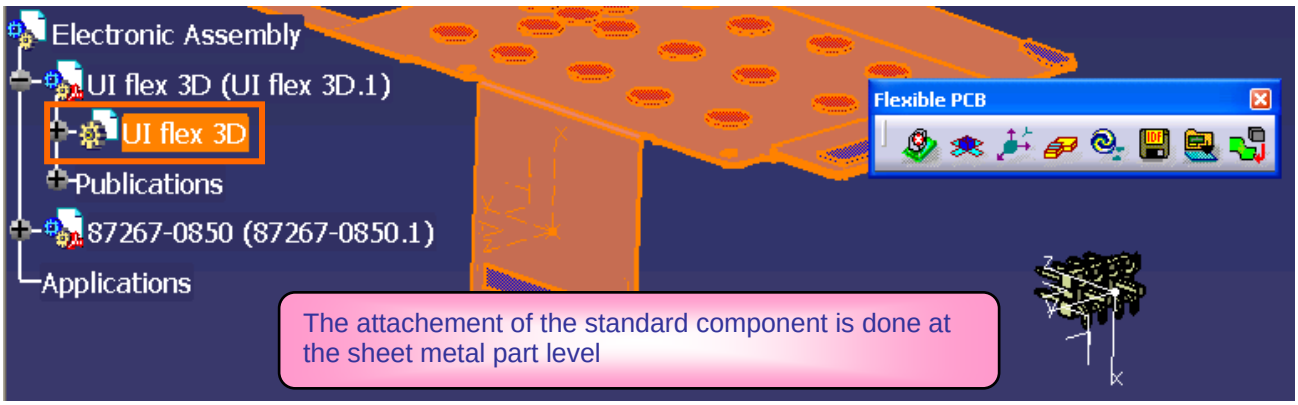


2. Instantiate the standard component at the assembly level or into a sub assembly



Everything is ready for component placement on the Sheet Metal Master part

3. Activate the Sheet Metal Master Part. If necessary, switch into folded view using Function “Fold/Unfold Board”

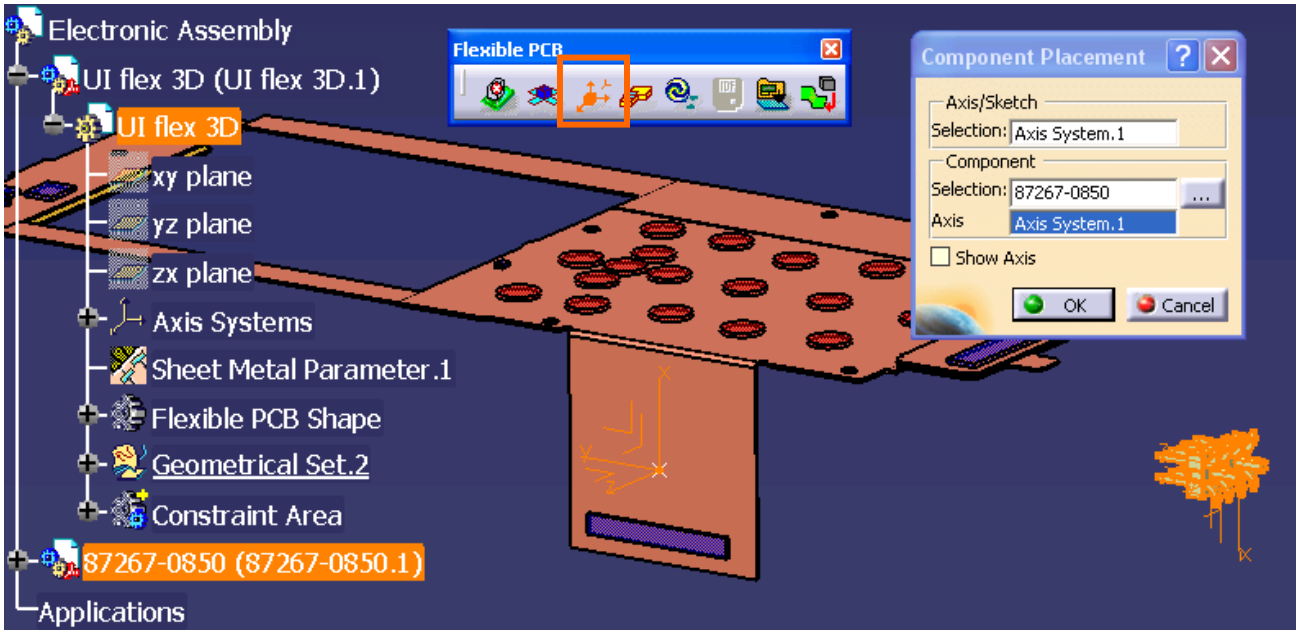


4. Select Function “Component Placement” in toolbar “Flexible PCB”

5. Select the Axis system defined in the Sheet Metal master Part. This Axis system is the location of the Standard Component in the flexible PCB part (folded view), and XY plane of Flex PCB axis system must lie within a SMD face.

6. Select the standard Component. (If no component is selected, then creation of the moving axis without automatic assembly constraint)

7. If there is a published axis system in component, it will be display in the command panel or use have to select a axis system from component part

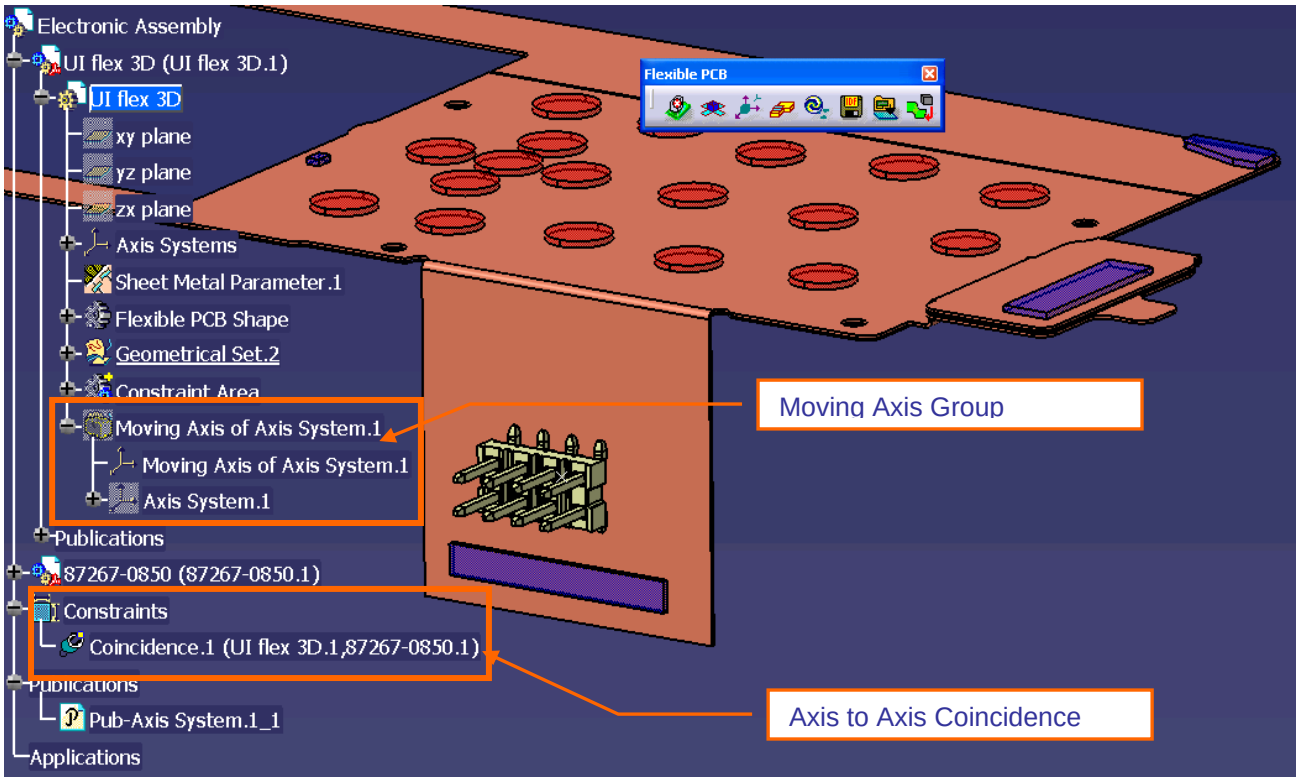


8. Select OK

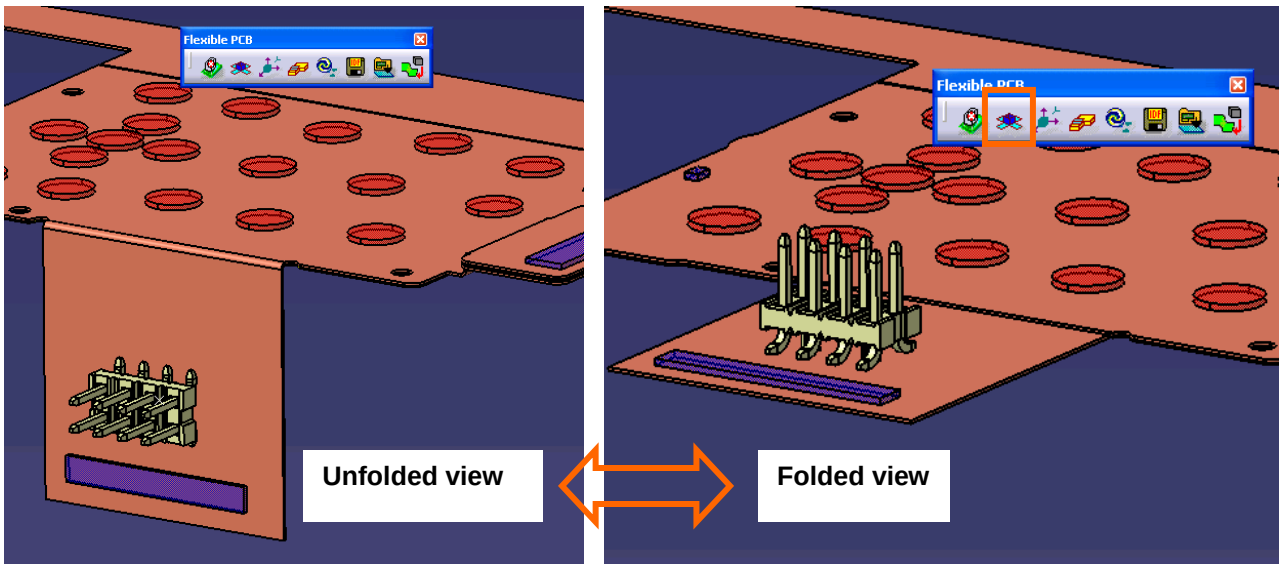
A group is created in the sheet metal master part. This group is linked to the axis system.1.

This group behaves like a “moving axis”, it follows the fold or unfold of the sheet metal part.

An assembly constraint, Coincidence “Axis to Axis” is created between the Standard Part Axis system and the Sheet Metal part “moving axis”. This coincidence can be modified manually with any other assembly constraints such as offsets.

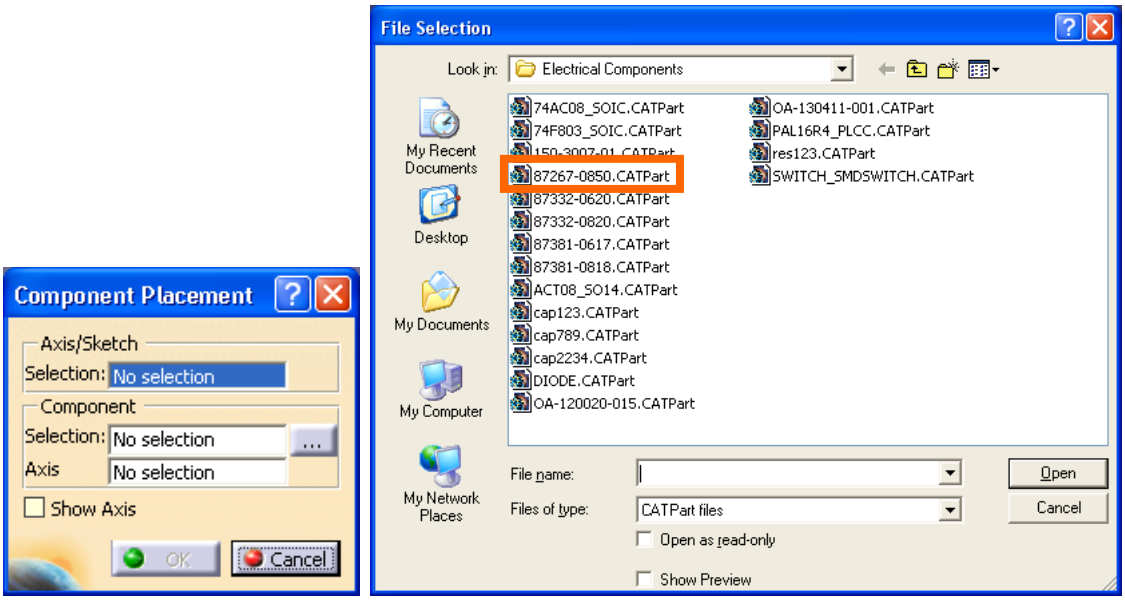


Therefore the standard component can follow the switch folded or unfolded when done with the function “Fold/Unfold board”



Placing Standard Component with import from external part file

1. Activate Flexible PCB Sheet Metal Part. If necessary, switch into folded view using Function “Fold/Unfold Board”
2. Select “Component Placement” function
3. Choose Axis system created in the Sheet Metal Part. This Axis system is the location of the Standard Component in the flexible PCB part (folded view), and XY plane of Flex PCB axis system must lie within a SMD face.
4. In order to get a component from an external part file, select the ... button in Component Placement window

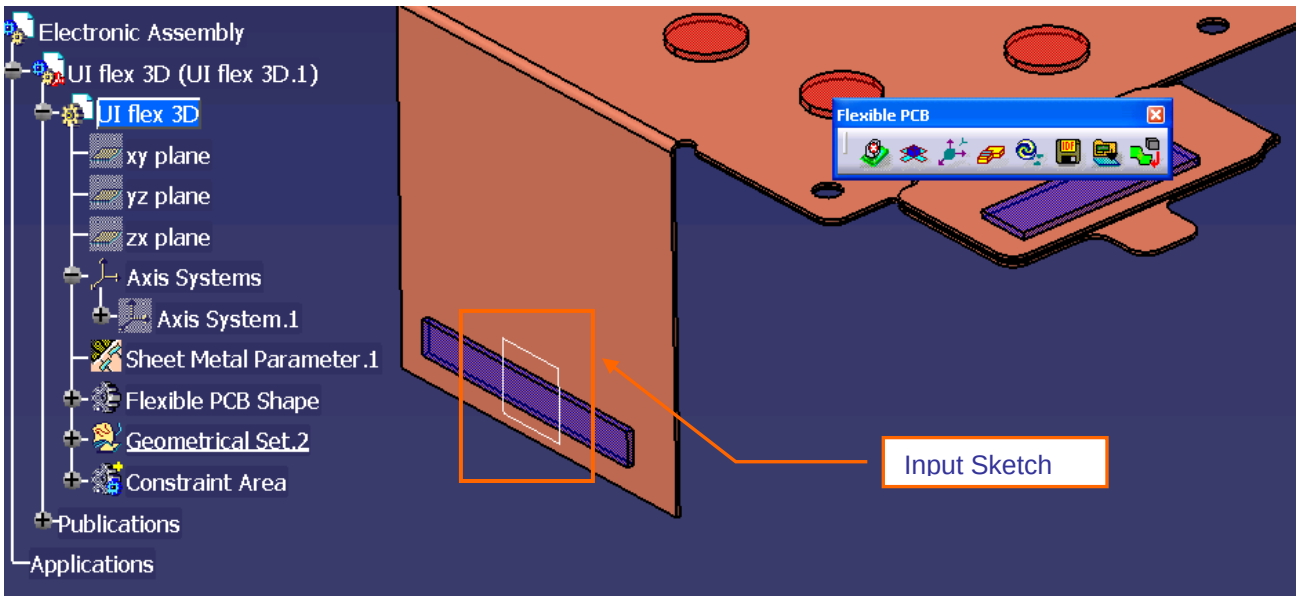


5. Choose and select the component to be placed
6. Display the published axis system of the component in the panel.
7. Press OK button
8. The component is inserted in the top product and placed on the sheet metal

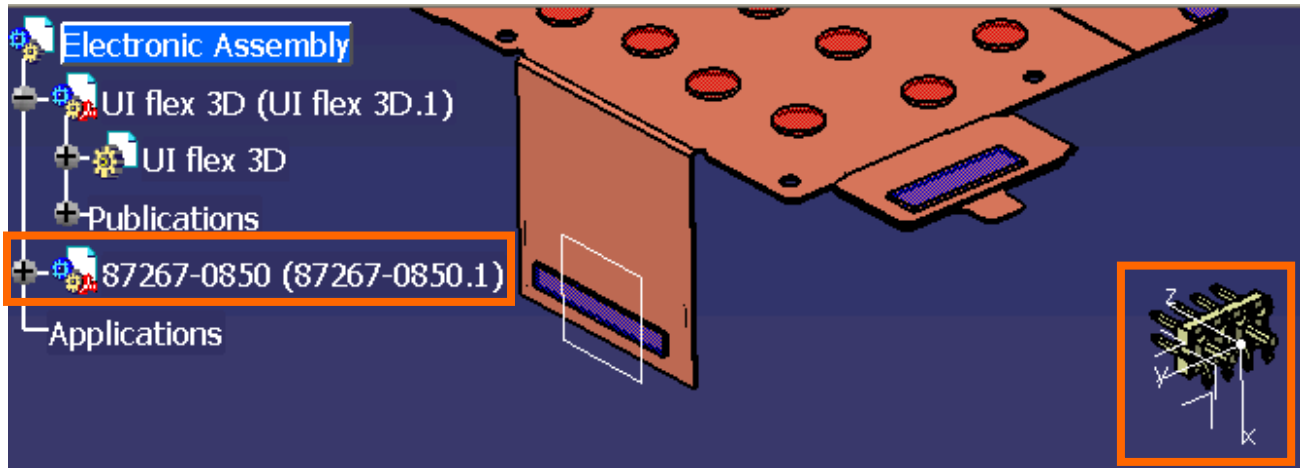
Placing Standard Component with Input sketch

1. Create a sketch in the Sheet Metal master part or use the profile of a constraint area

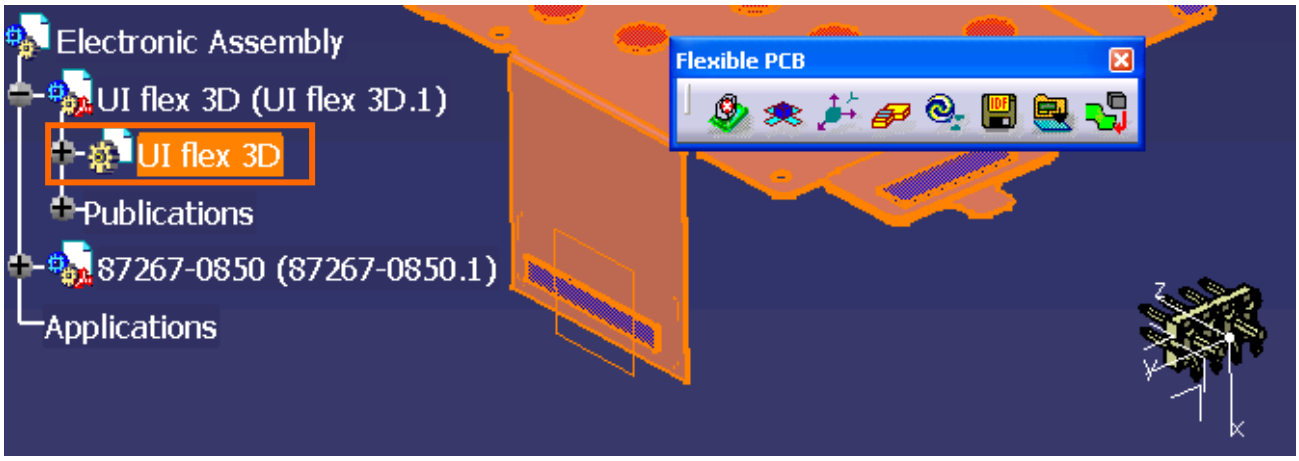
This sketch corresponds to the location of the standard part to be instantiated. It can be a profile already used for a Component placement constraint area. The standard part local axis system will be centered on the barycenter of the sketch with Z axis orthogonal to the sheet metal face.



2. Instantiate the standard component at the assembly level



3. Activate the Sheet Metal Master Part



4. Select Function « Component Placement » in toolbar “Flexible PCB”
5. Select the Axis system defined in the Sheet Metal master Part
6. Select the standard Component

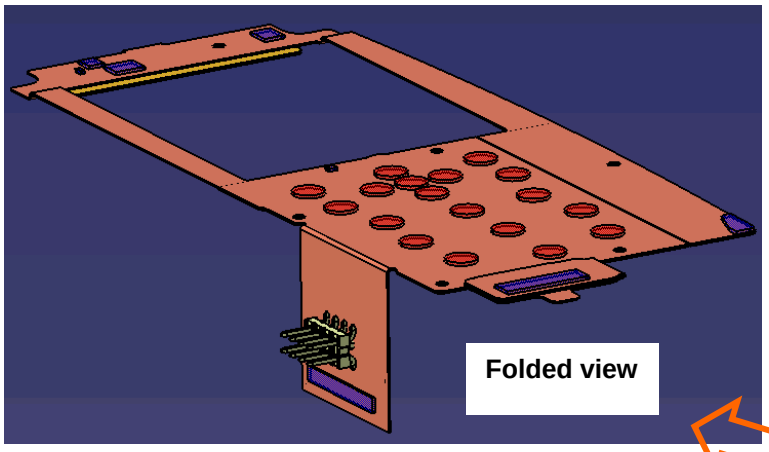
7. Select OK

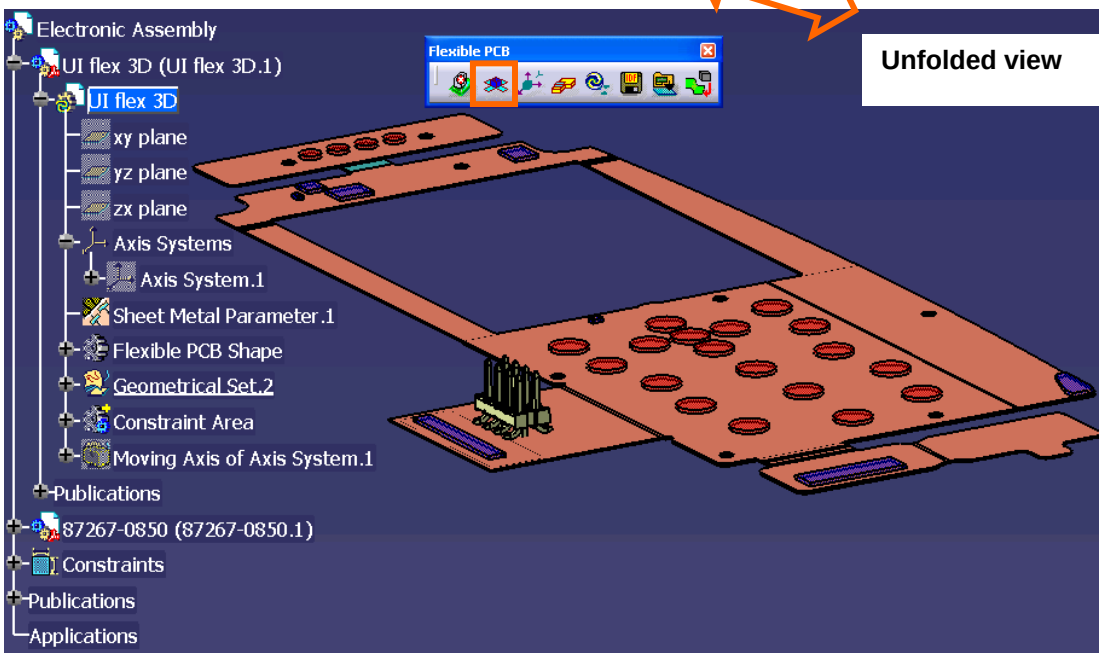
A group is created in the sheet metal master part. This group is linked to the input sketch.

Switch to Unfold View

1. Select function “Fold/Unfold” in “Flexible PCB” toolbar to flatten the flexible PCB part.

It will reposition the attached electrical component along with the sheet metal master part as well





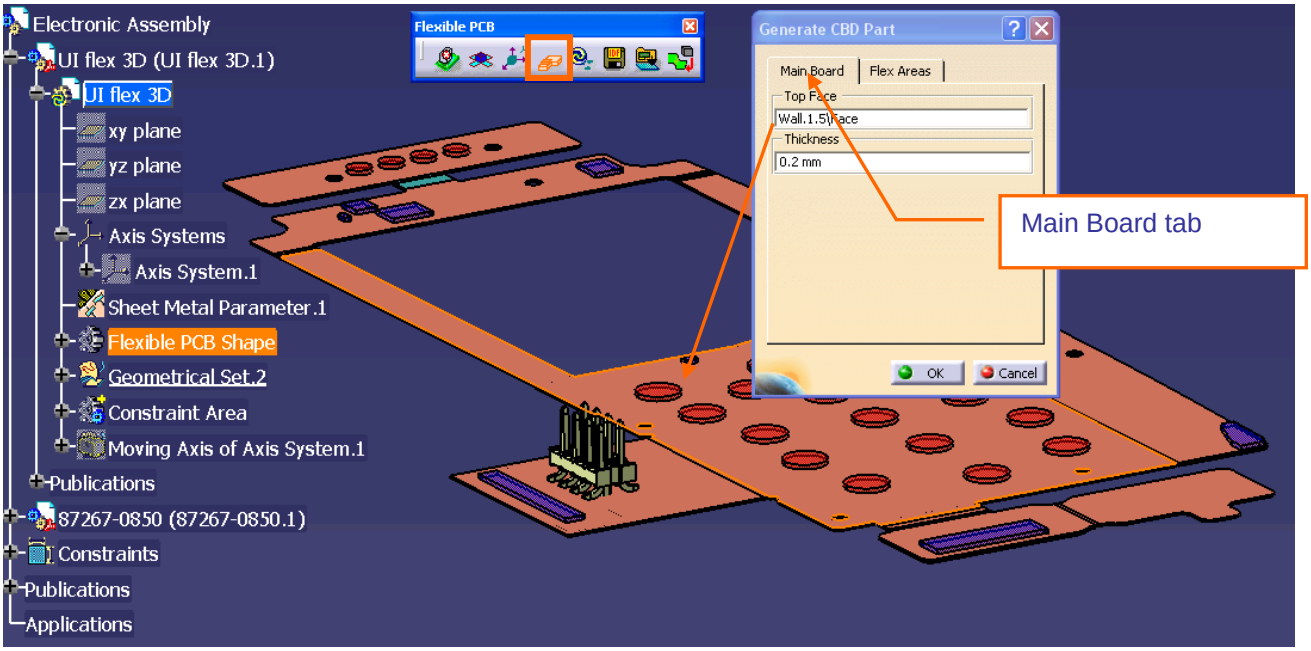
We can start the procedure to have this flexible PCB Part ready for export to any EDA System through IDF format. A new part is generated to that purpose: “InteractiveBoard.CATPart”. This part is associative with the sheet metal based master Flexible PCB CATPart.

Generation of the Flexible PCB Part

The creation of the flexible PCB Part must be done in the unfolded view. Make sure you area in this view otherwise a warning panel will remind you.

Generation of the flexible PCB outline (Main Board)

1. Select Generate CBD Part function in toolbar “Flexible PCB”
2. Select Tab Main Board



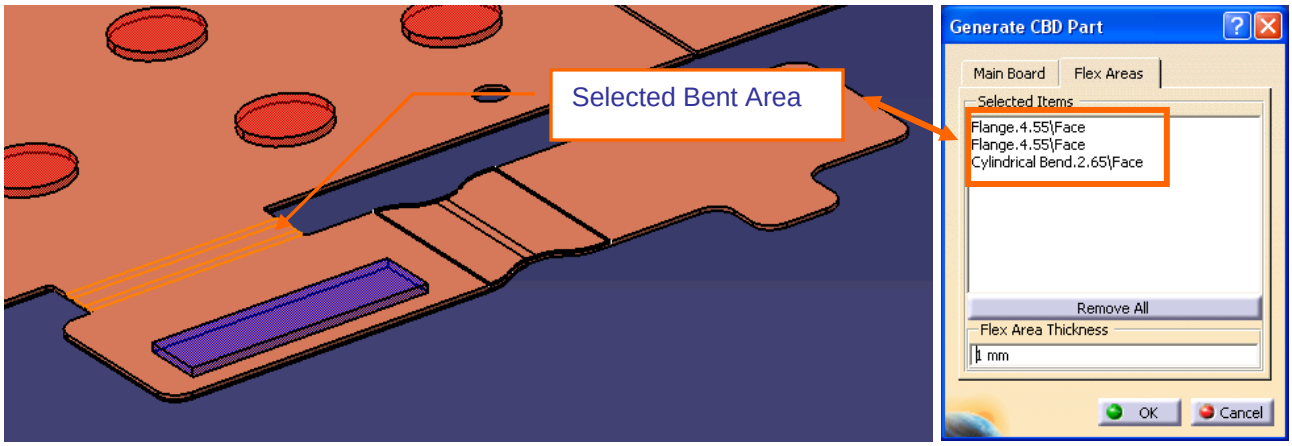
3. Choose the Top of the Board by selecting any face on it

By default, the thickness of the Flexible PCB in the panel is equal to the Sheet Metal Part’s thickness. It can be modified.

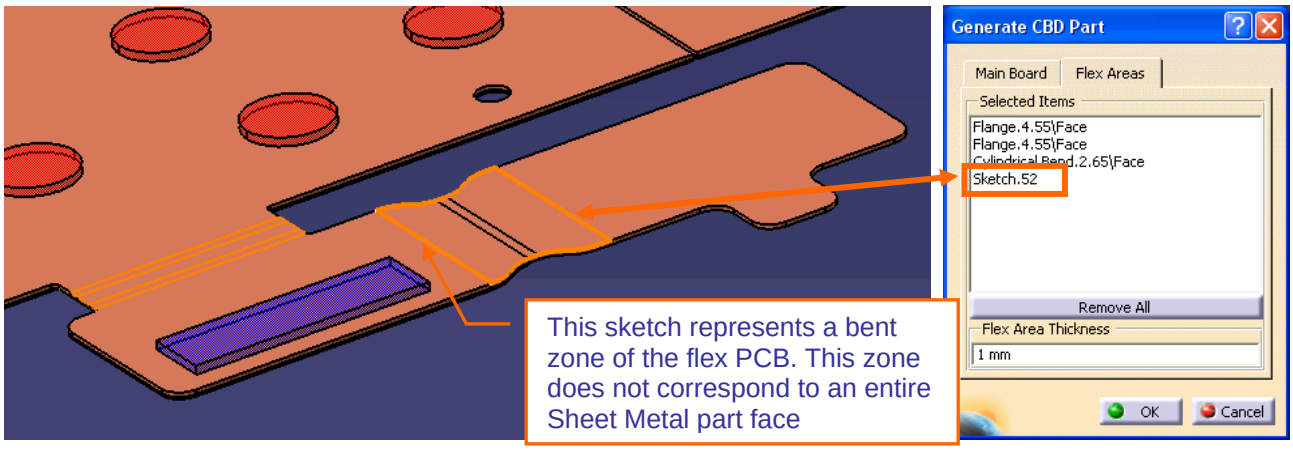
Selection of the flexible areas

4. Select Tab Flex Area

5. Multi Select the faces of the area that are bent in the unfolded view of the master CATPart

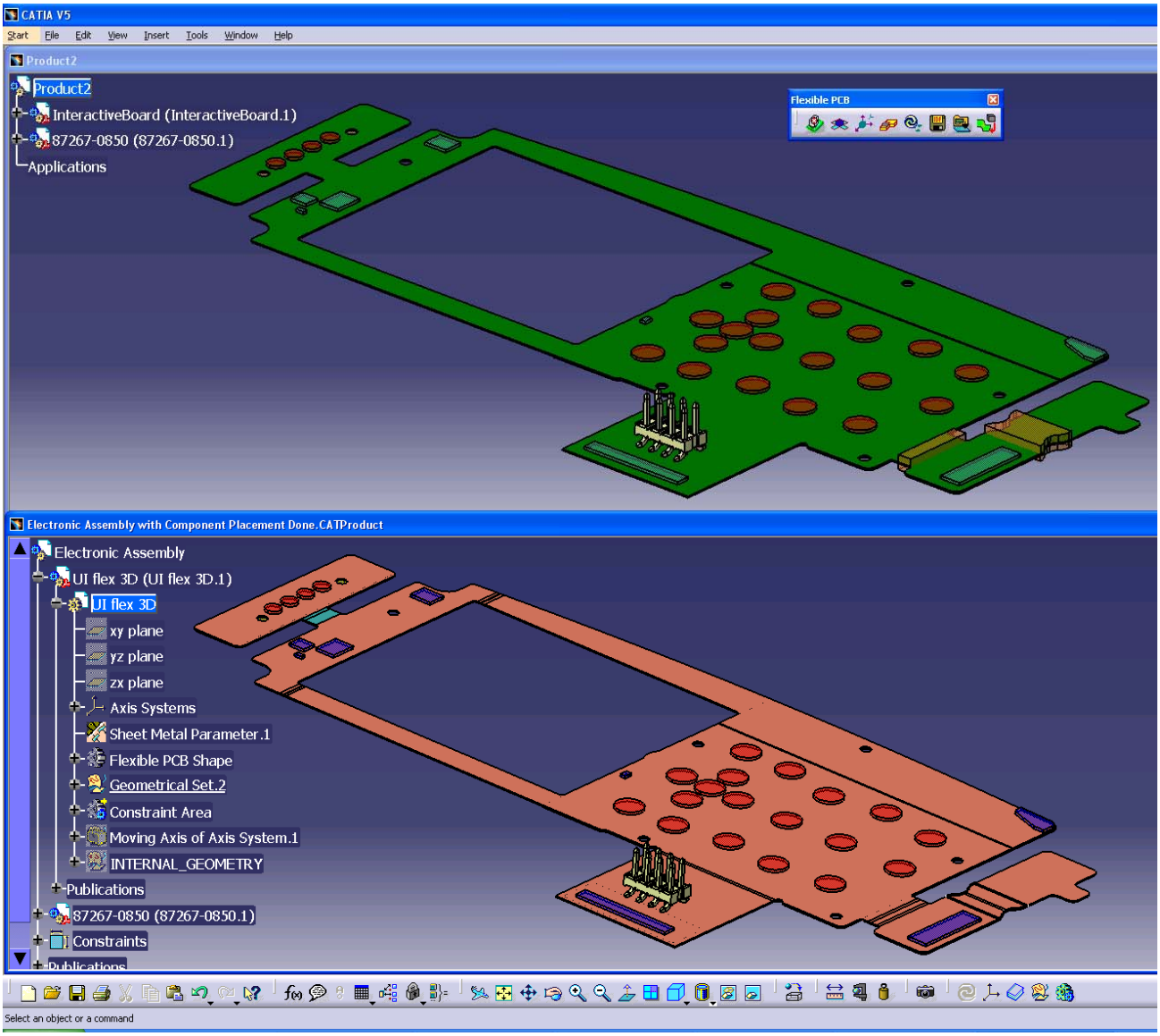


6. For a bent area which does not correspond to an entire face of the sheet metal model, it can be done through a sketch created in the flat view. This sketch can then be selected in the same Flex Area Tab.



7. The default thickness is set to 1mm and can be modified

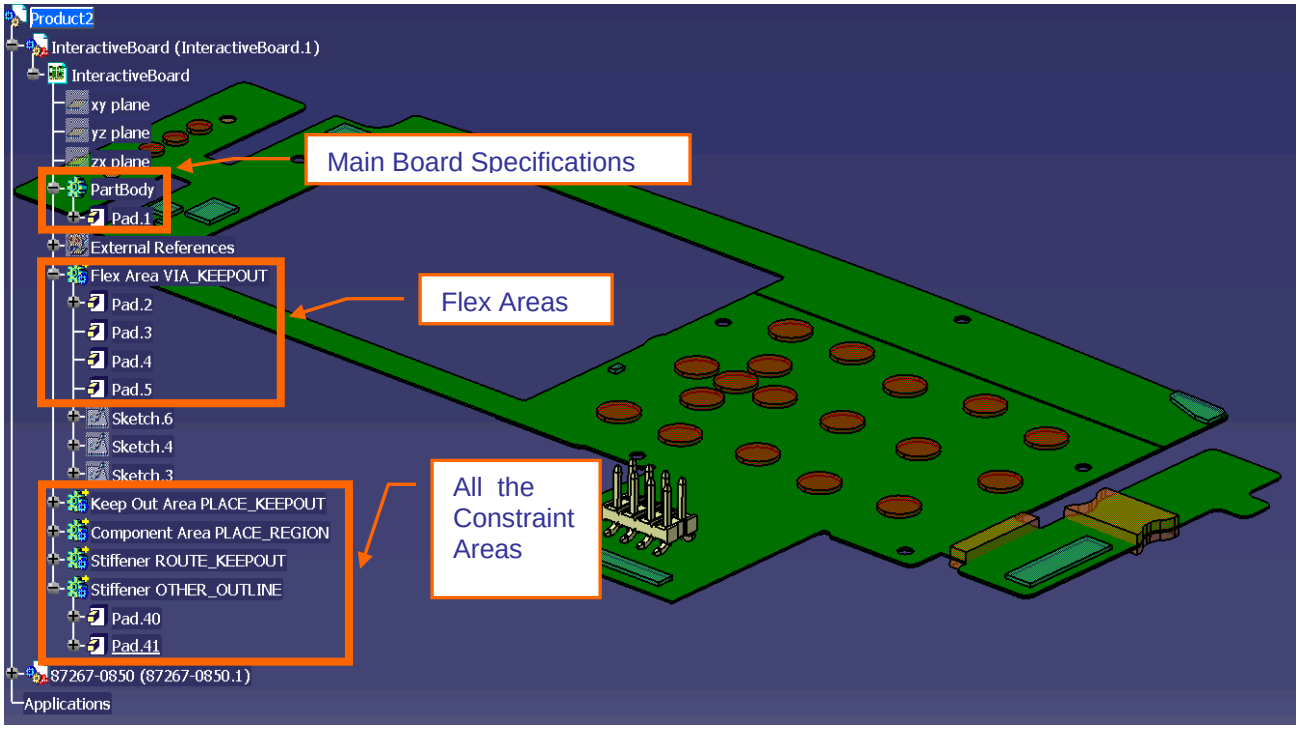
8. Select OK to validate selection



A new product has been created, with a CATIA Circuit Board Design (CBD) part (the Flexible PCB part). This part is made of the flatten shape of the Flexible board, the Constraint and Flexible areas defined. The product does contain also the electrical components that have been positioned earlier.

Generation of the Flexible PCB Part

The Flexible PCB part is generated. It contains a Geometrical set External references" with the planes corresponding to the flexible PCB's top and bottom faces of the flexible PCB and the link to all the sheet metal par features relevant to create a link. This part is linked to the master CATPart.



Bent areas are created in InteractiveBoard.CATPart as VIA_KEEPOUT in a body called "Flexible Area VIA_KEEPOUT".

This can be changed by modifying the constraint area configuration.

Flex Area:

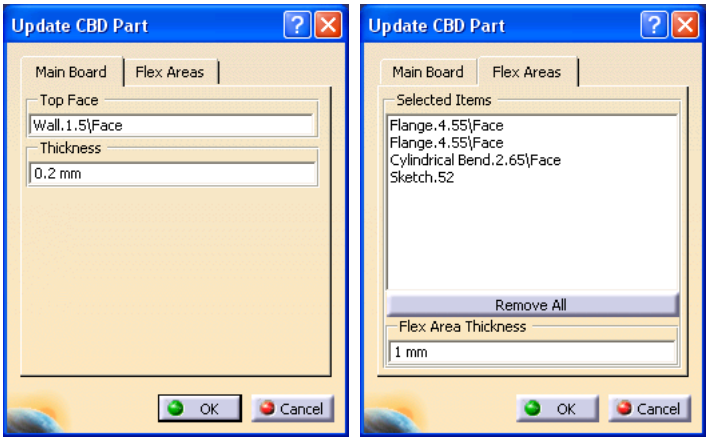
There are 7 types of constraint area that could be used for flexible areas when generating CBD board (see Constraint areas default configurations – page 7).

Update of the Flexible PCB Part

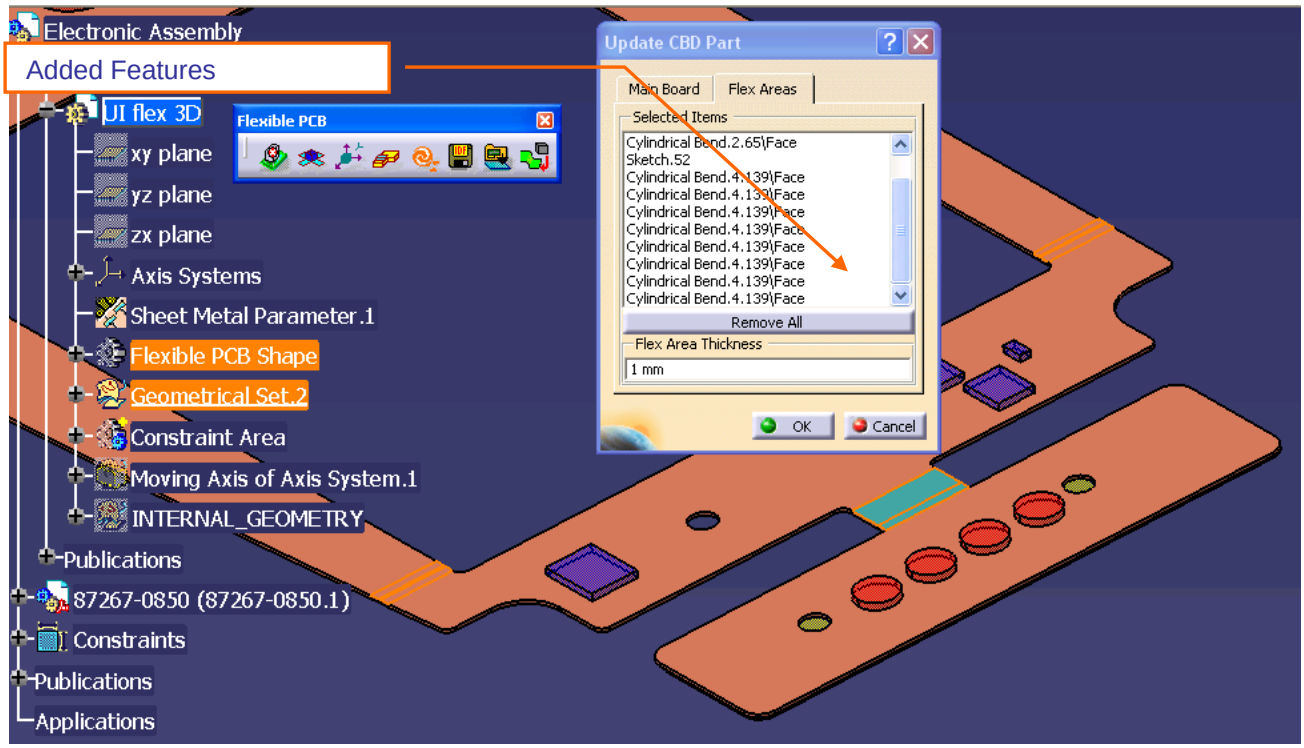
When modifications have occurred in the sheet metal part, the flexible PCB Part must be updated. Let us add some flexible area to the flexible PCB.

1. Select function “Update CBD Part”

The features already selected and set parameters are retrieved automatically.



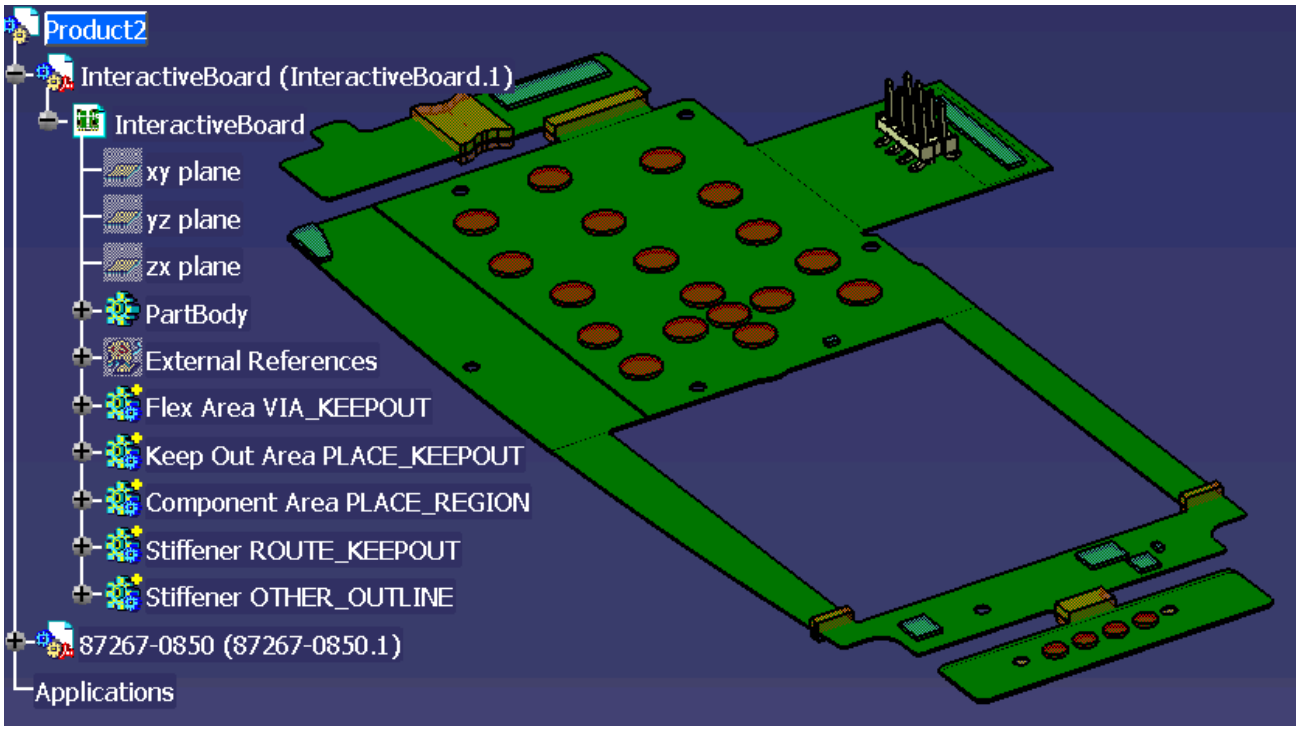
2. Select the flexible Area to add in the flex area index



For any modification of the Sheet Metal Master Part, addition or deletion of constraint area, walls and bends, the function “update CBD Board” must be used to update the flexible PCB Part

3. Select OK

The Flexible PCB Part is then updated

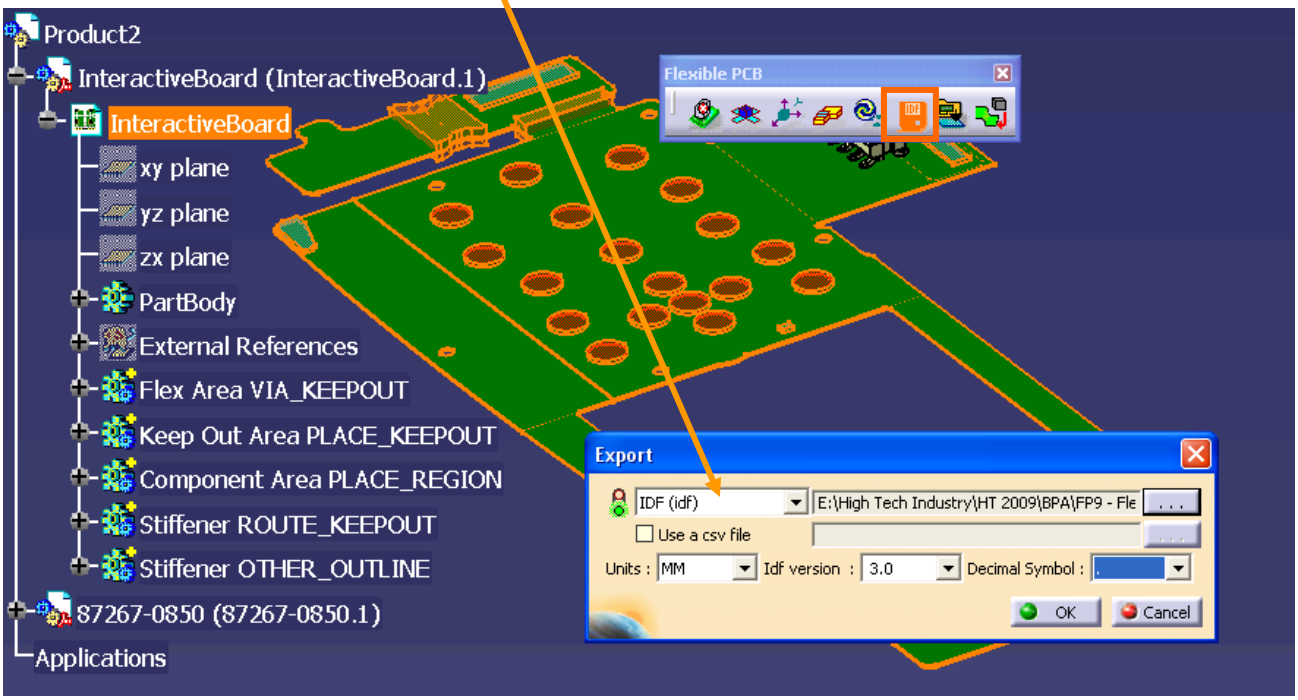


“Generate CBD Part” function supports flexible PCBs whose flat representation lies on top of a basic plane (XY, YZ or XZ) or a plane which is an offset of a basic plane

When deleting a Sheet Metal Feature, Update Board is not supported. The geometrical set “internal geometry” must be deleted and recreated using function “Generate CBD Part”

Export IDF File

1. Select Function « Export As IDF file » in toolbar “Flexible PCB”
2. Set up relevant option for the IDF



You can choose between three extensions for the IDF type:

- .idf: the general format extension
- .brd: the Mentor Graphics specific IDF extension
- .bdf: the Allegro specific IDF extension

You can choose between:

- two unit systems: MM: millimeters, THOU: 1/1000 of inch
- two IDF versions: IDF 2.0, IDF 3.0
- And two decimal symbols: point (.) or comma (,)

For other options, please refer to the CATIA Circuit Board Design User documentation

3. Click OK to perform the export. An Information message is displayed:

Import IDF in Context of Master Flexible CBD Part

1. Open Assembly containing original master Flexible CBD Part
2. Activate master Flexible CBD Part
3. Switch into Folded view using function “Fold/Unfold Board” 

4. Select “Import in Context” function 

5. Select idf files, lib files and if necessary standard component detailed catalog

The .idf file contains the location of the components in the circuit board assembly.
You can also use a library and/or a catalog to get a better geometry of the components:

- the .lib file contains the 2D footprint geometry of the components
- the .catalog contains the 3D (exact) geometry of the components.

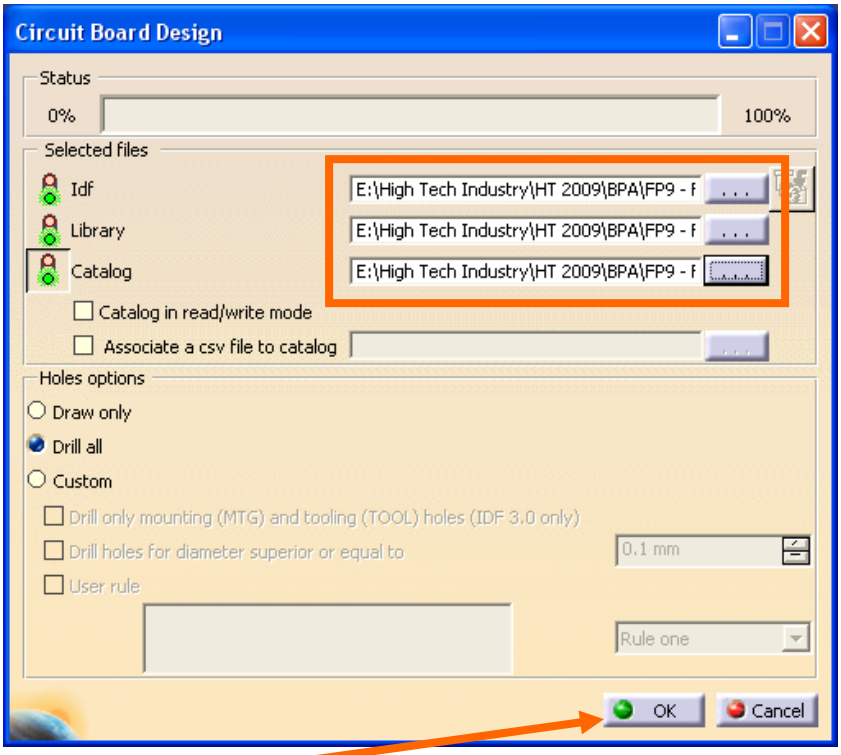
However, none of these two files are mandatory.

You can choose between several alternatives:

- Open a library only: the 3D geometry is extruded from the 2D geometry
- Open a library and use a catalog.
- Open no library and no catalog, the component has no associated geometry.

The light shows, according to a color code:

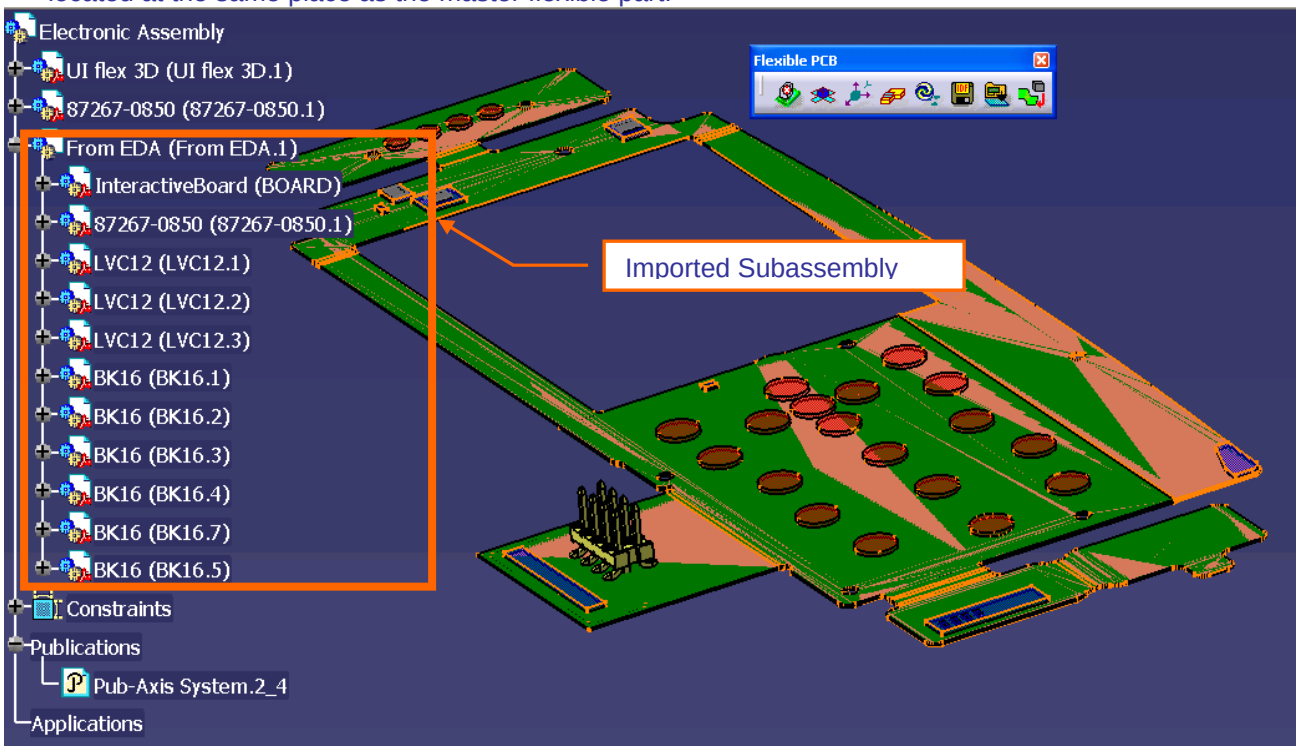
- if the file is mandatory, the light is red 
- if the file is optional, the light is yellow 
- if your selection is valid, the light turns green. 



6. Launch the import

7. Close the imported assembly view, the imported structure is a subassembly of the flexible CBD Part's root product and located at the same position in flat view.

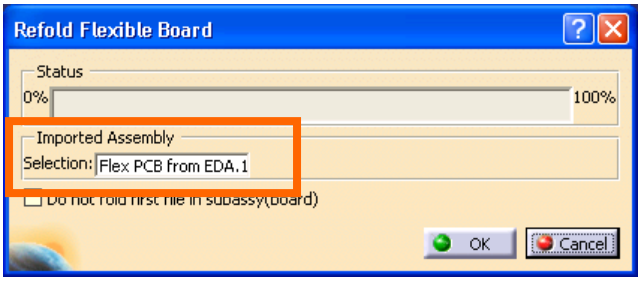
A new subassembly is created in the master flexible part assembly, representing the idf file import. It is located at the same place as the master flexible part.



Refold Flexible Board

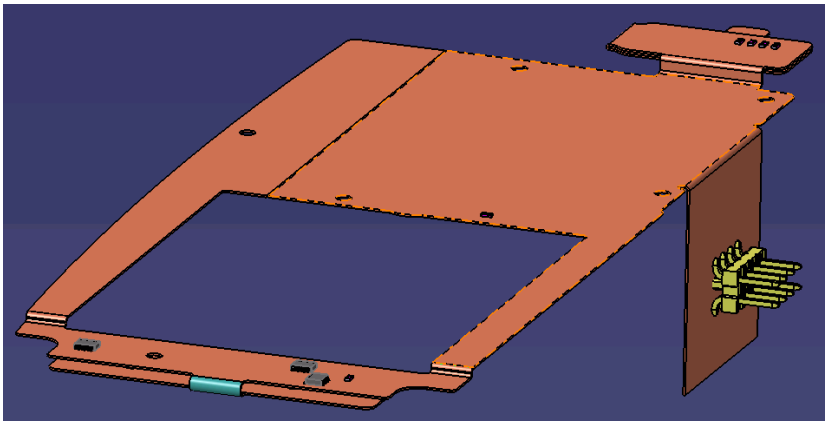
8. Select “Refold” function 

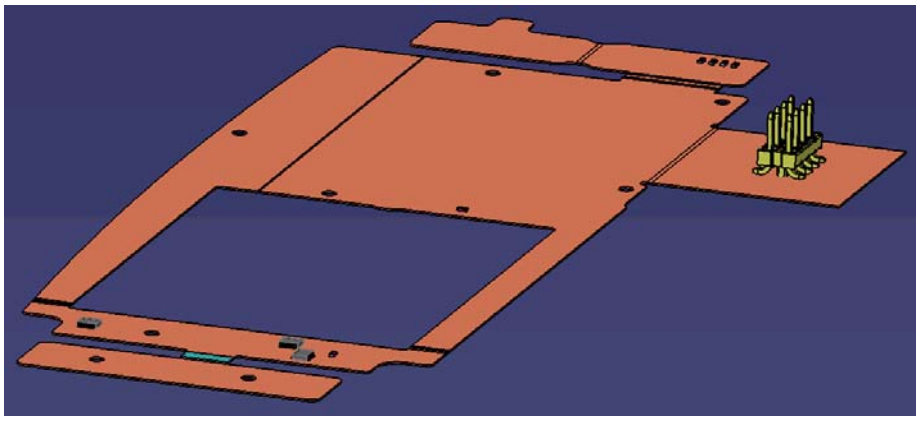
9. Select the imported subassembly and click OK



A special geometrical set is created with two subsets containing the moving axis for the components and the constraints area.

At the end of the procedure all the imported components and constraint area are attached to the master flexible PCB. The function “Fold/Unfold Board” impacts the position of the imported components and constraint area.





In case where you want to refold only specific components, please follow the following steps:

1. Create a product that aggregates all components which will be refolded with sheet metal.
2. Switch the sheet metal to flat model if necessary
3. Move all components onto sheet metal
    a. The default axis or the published position axis must be laid on sheet metal
4. Launch the command “Refold Flexible board”
5. Select the product that aggregates the components
6. Press OK button
7. The components are placed on the sheet metal.

New in CATIA Flexible PCB Automation (FP9) – v5.7

- **Generic Tool Options Management**
- **Constraint Areas Default Configurations**
- **Flexible PCB Integration with Zuken Board Interchanger**
  - **Include components when generating CBD board**
    - Addition to the existing command, which supports only Board and Constraint Areas
  - **Refold BI board**
    - Provide Flexible capability to the model created by Zuken's BI