

Flexible PCB Automation (FP9)

BPA Delivery 6 for V5R19 (V5.6)

Implementation Guide

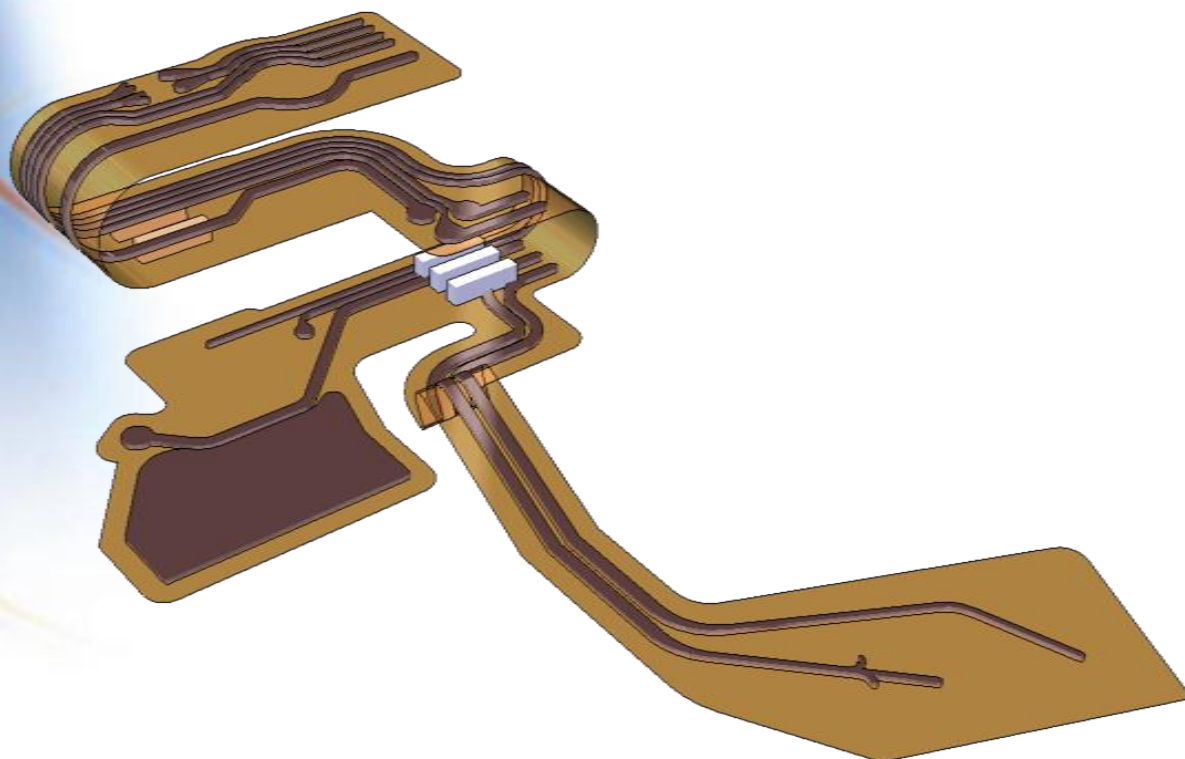


Table of Contents

Table of Contents	0
Copyright Notice	1
About Printed Circuit Board (PCB) Design	2
Flexible PCB deliverables.....	4
Generation of Constraint Area	6
Switch to Unfold View	12
Generation of the Flexible PCB Part.....	13
Export IDF File	18
Placing Standard Component with Input sketch	19
Placing Standard Component with Axis Systems	21
Placing Standard Component with import from external part file	24
Import IDF in Context of Master Flexible CBD Part	25
Refold Flexible Board	27

Copyright Notice

© 2008. Dassault Systèmes, All Rights Reserved.

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

CONFIDENTIAL - This document contains unpublished, confidential and proprietary information of Dassault Systèmes.

This document or any part thereof shall not be reproduced or transferred to other documents or formats, disclosed to others or used for any purpose other than that for which it is furnished, without the prior written consent of Dassault Systèmes.

It shall be returned to Dassault Systèmes upon request.

Dassault Systèmes is a registered trademark of Dassault Systèmes.

All other trademarks belong to their respective owners.

CATIA is a registered trademark of Dassault Systèmes SA.

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 Mechanics design, so that Mechanics 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.

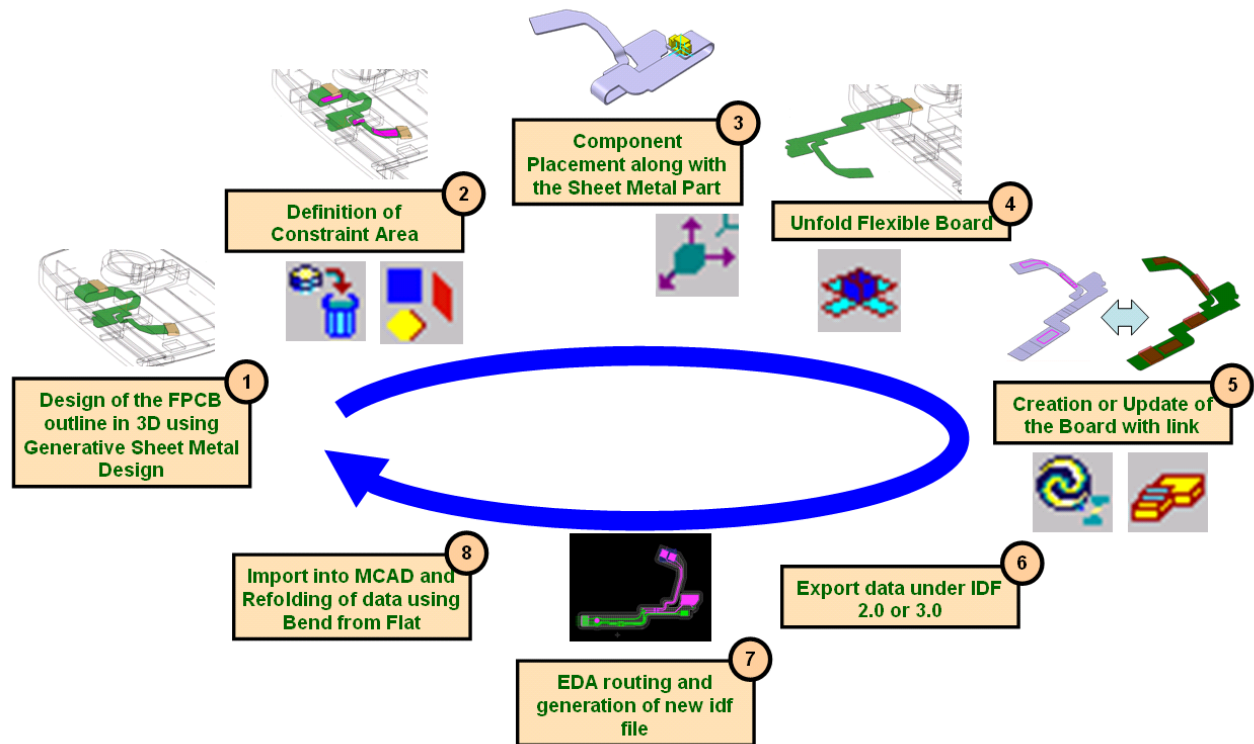
All updates are sent to ECAD. Because library files cannot be exported, all area and component space reservations must be done by Constraint Areas.

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

WORKFLOW Reminder

1. Hardware assembly contains Flexible PCB board, Mechanics components divided into Top and Bottom side subassemblies and component subassembly from ECAD
2. Flexible PCB board outline and Constraint Areas are created by Mechanics 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 (flex) areas?
2. Fold/Unfold Board 


It flattens and folds a master sheet metal part, attached components and constraint area.
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 

Formatting of the Sheet Metal model into a ready for export flattened board. The specification tree structure allows an easy drafting generation.
5. Update CBD Part 

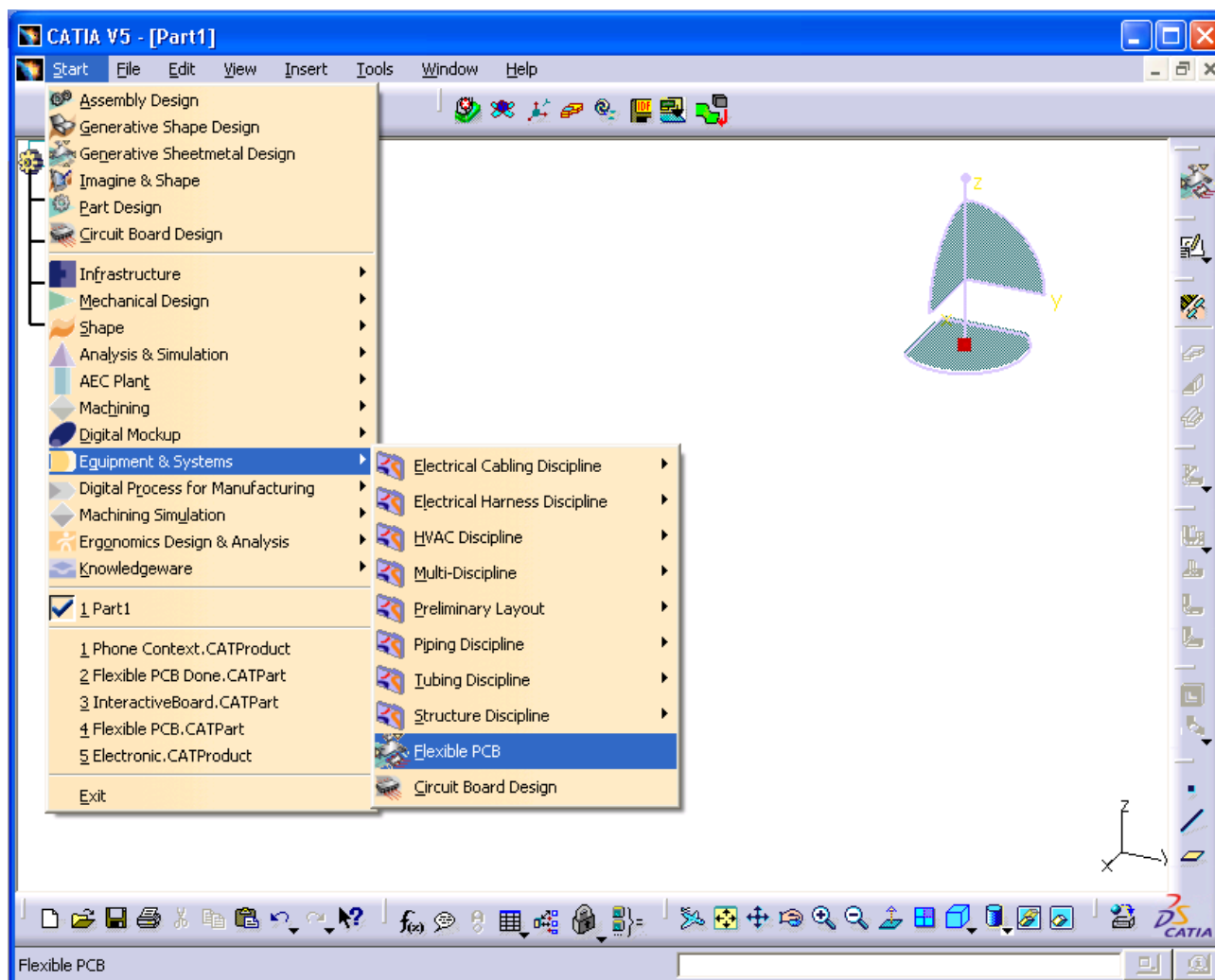
The master sheet metal model and the formatted model are fully associative through the use of this function
6. Export As IDF File 

It generates an IDF files that can be read by any EDA tool.
7. Import In Context 

It imports an IDF files and places it at the same location as its flexible PCB master model.
8. Refold Flexible Board 

It folds the constraint area and components of the imported IDF file according to the master flexible PCB part

These capabilities can be accessed 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.

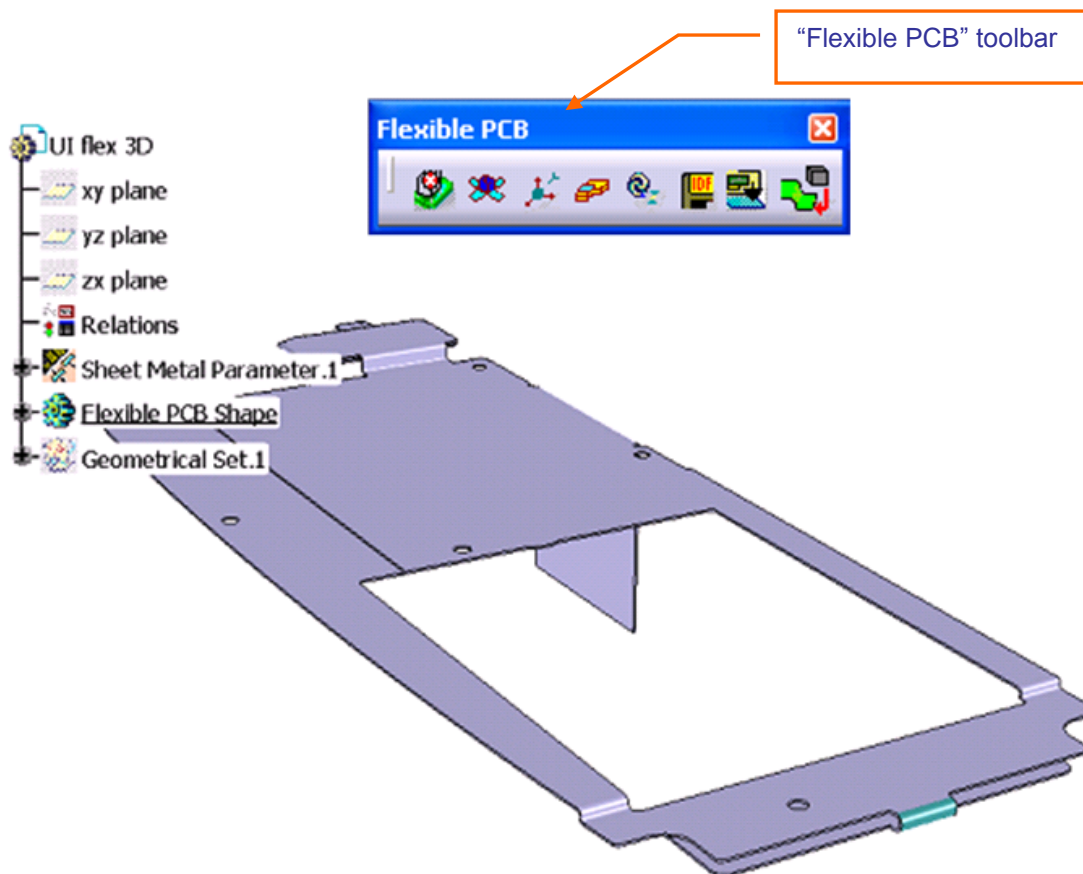


Figure 1

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.

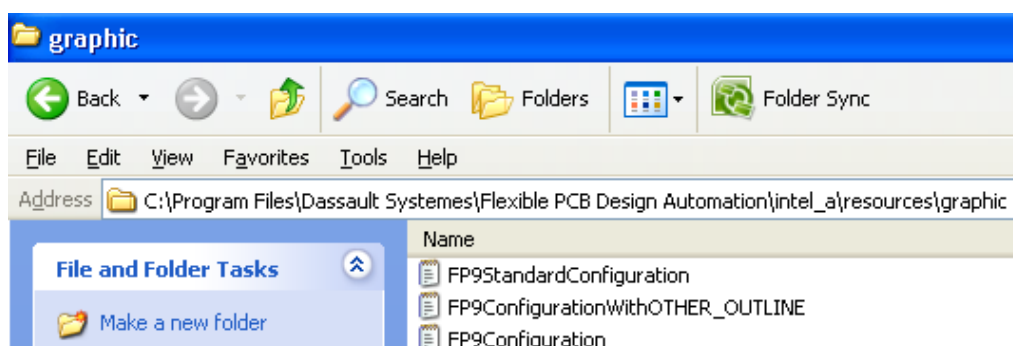
By default, there are 4 different types of constraint area available: Component Area, Keep out Area, Cover Layer Opening and Stiffener.



Constraint type	IDF Equivalent	Default Limit	Default Color
Component Area	PLACE_REGION	0.4mm	Blue
Keep out Area	PLACE_KEEPOUT	0.4mm	Red
Stiffener	ROUTE_KEEPOUT	0.1mm	Yellow
Cover Layer Opening	VIA_KEEPOUT	0.05mm	Red

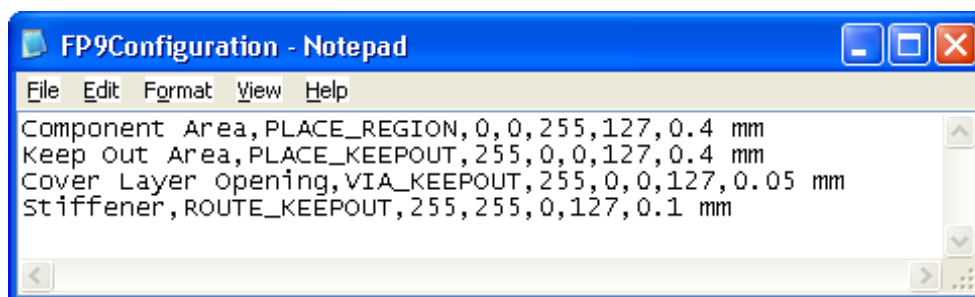
Those parameters are customized through a text file named “FP9Configuration”.

There are several configuration files delivered. These files are located in:



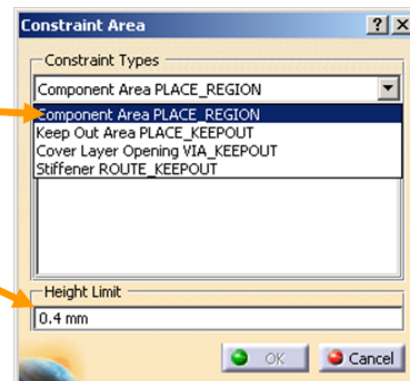
C:\Program Files\Dassault Systemes\Flexible PCB Design Automation is the installation directory defined during the FP9 BPA installation.

FP9Configuration.txt file is the default configuration file. Here is the format of the text file:

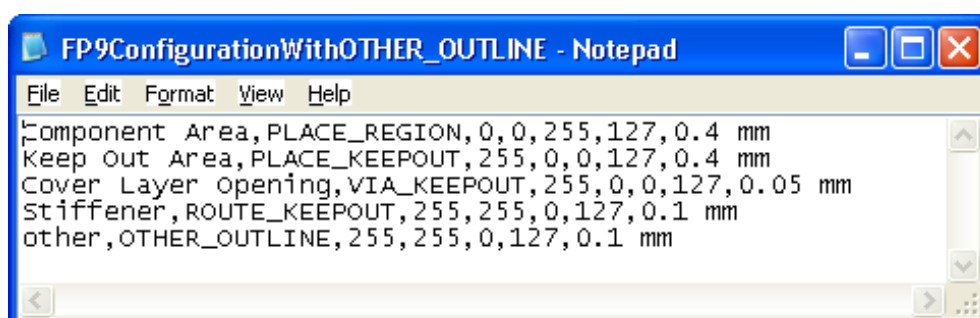


Constraint Type		Color				Height Limit
MCAD Name	EDA Name	R (Red)	G (Green)	B (Blue)	A (Transparent)	
Component Area	PLACE_REGION	0	0	255	127	0.4 mm

Constraint Type		Height	Color	
MCAD name	ECAD name	Limit	RGBA	Color
Component Area	PLACE_REGION	0.4 mm	0, 0, 255, 127	Blue
Keep Out Area	PLACE_KEEPOUT	0.4 mm	0, 0, 255, 127	Red
Cover Layer Opening	VIA_KEEPOUT	0.05 mm	0, 0, 255, 127	Red
Stiffener	ROUTE_KEEPOUT	0.01 mm	0, 0, 255, 127	Yellow

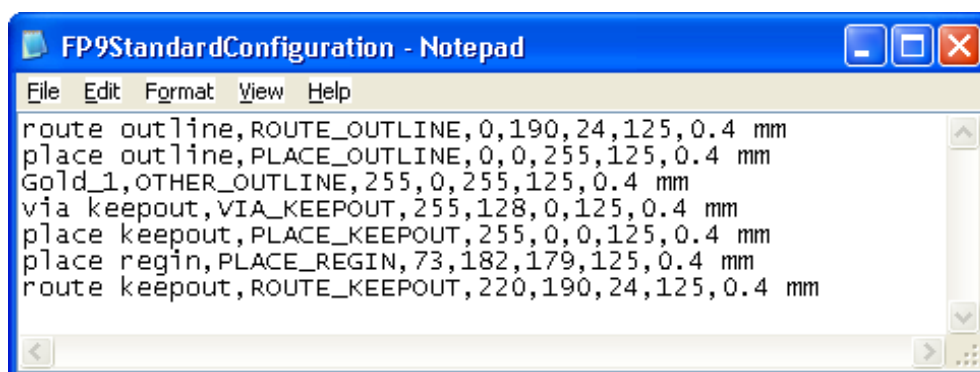


FP9ConfigurationWithOTHER_OUTLINE.txt file is the file to be able to create OTHER_OUTLINE constraint areas type. Here is the format of the text file:



FP9StandardConfiguration.txt file is the configuration file with the same constraint areas than CATIA Circuit Board Design workbench (CBD). It should be used if you want to maintain consistency between constraint areas created on rigid boards with CBD and constraint areas created on flexible boards with FP9.

Here is the format of the text file:



In the case you want to use FP9ConfigurationWithOTHER_OUTLINE.txt or FP9StandardConfiguration.txt as the default configuration file, please follow the following procedure:

1. Rename the existing FP9Configuration.txt file
2. Copy the configuration file you want to use and rename it as FP9Configuration.txt

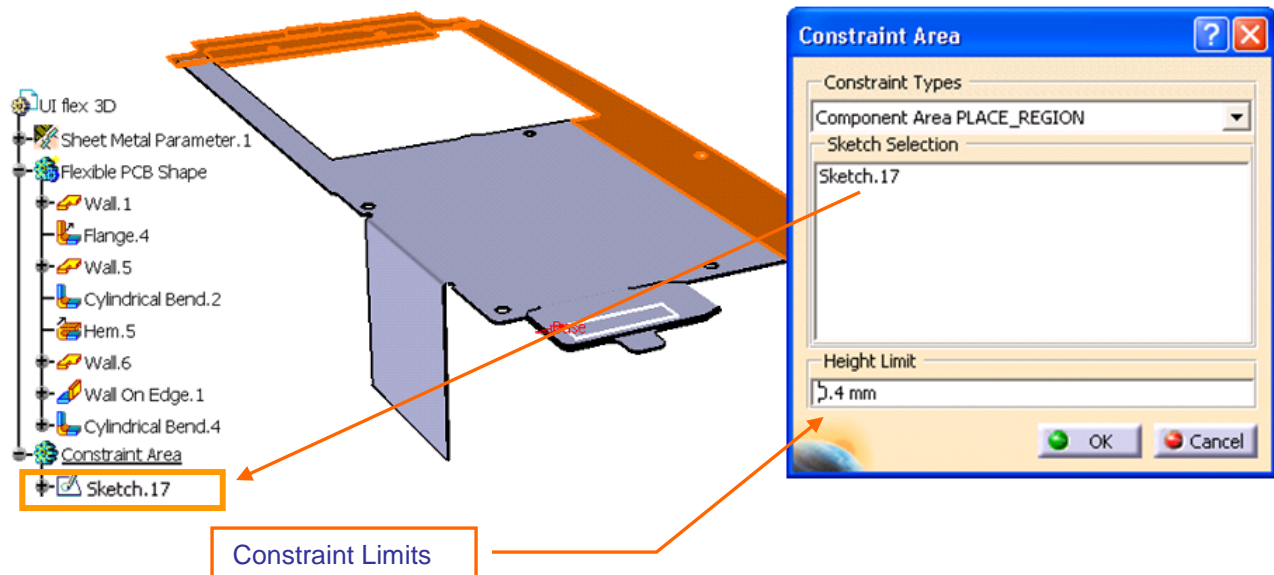
Component Area Generation

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”



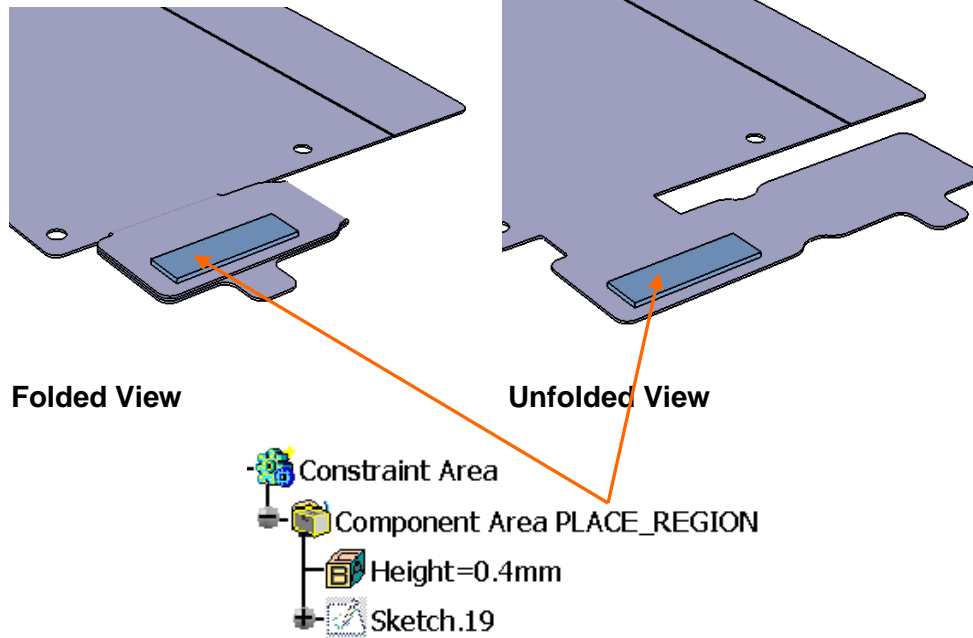
2. Select inputs required to create this area



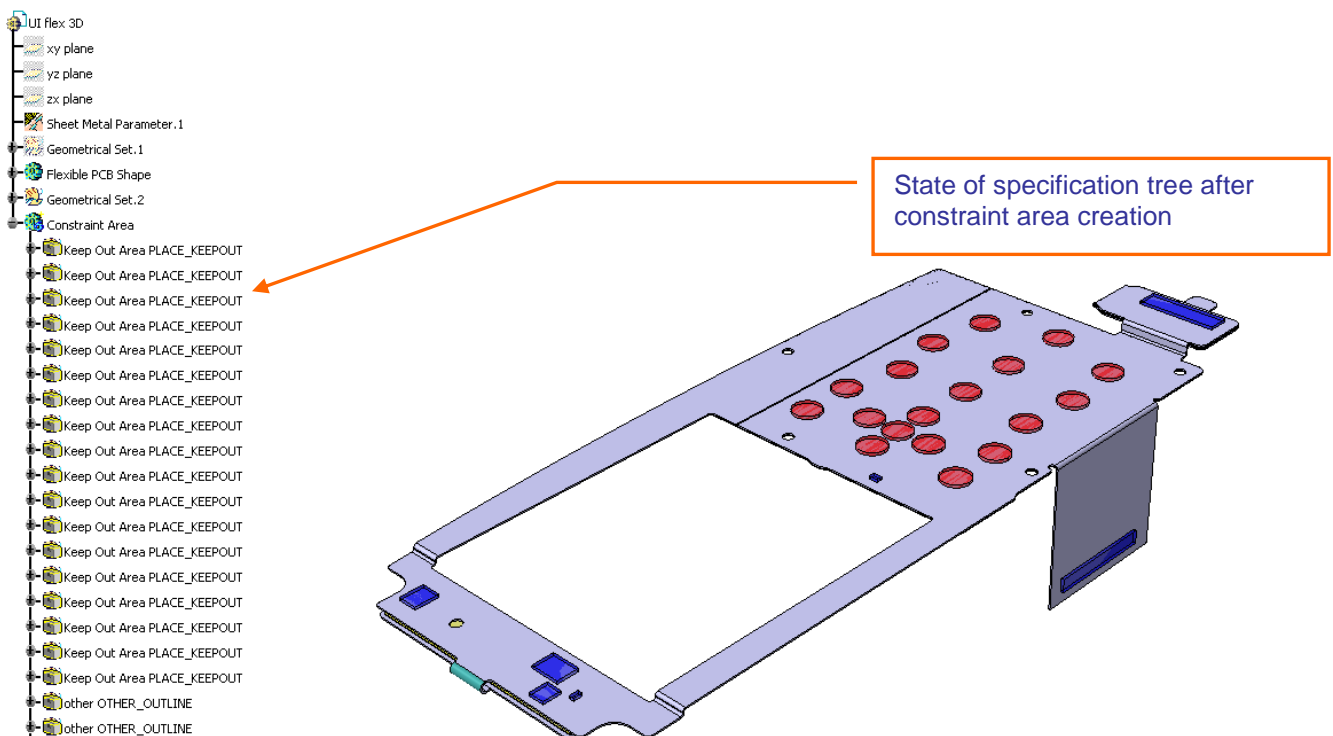
3. If necessary, set another area's limit

4. Press OK

A Group 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".



It must lie within the flexible PCB's outline

Selection of isolated Sketches is forbidden

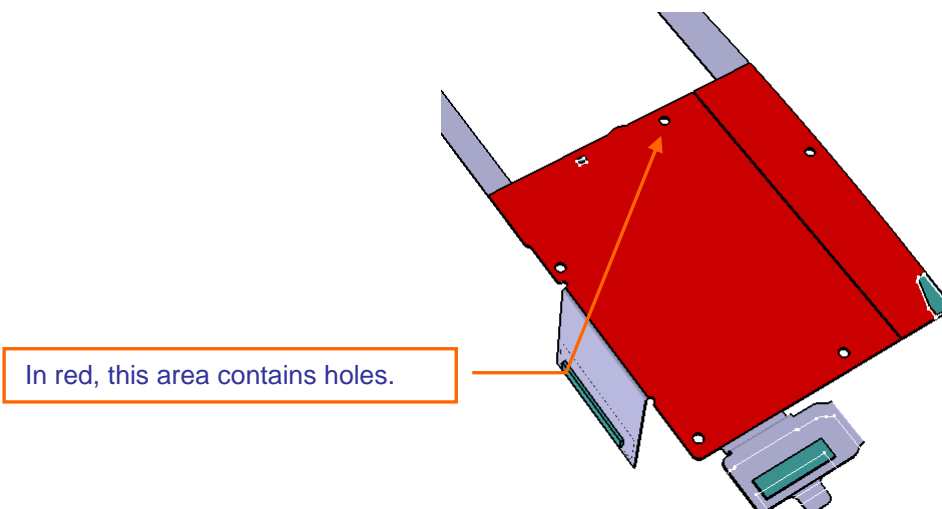
Selection of closed profile sketches with holes were forbidden (not compliant with IDF standard) in V5.5. Only OTHER_OUTLINE could be defined with holes constraints as it is supported by IDF.

This limitation can be removed in V5.6 by setting FP9Configuration.txt. The column 8 is added and the meaning is as following:

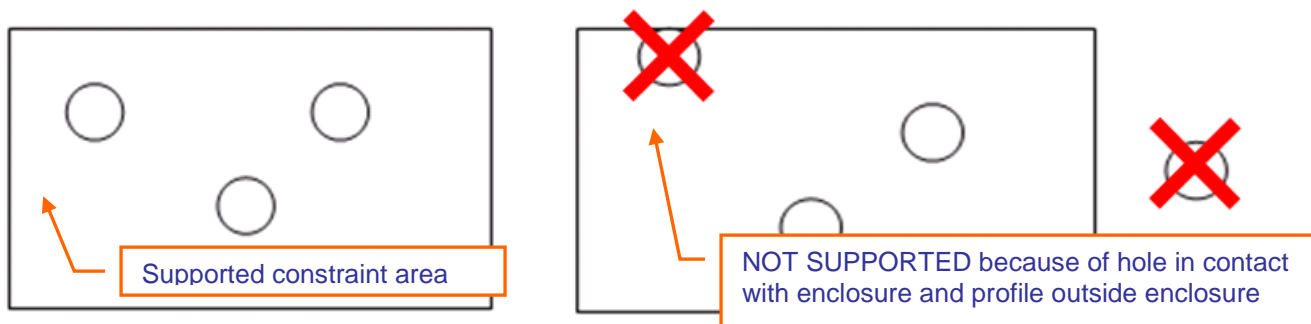
- 0 : One Closed Profile
- 1 : Multiply Domain
- F : f or flex area

Warnings:

1. If there is no value in column 8 or the value in the column 8 is 0, the selection of closed profile sketches with holes is forbidden except OTHER_OUTLINE
2. If the value of column is 1, then the Multi profile sketches can be selected for the constraint type of that line.
3. The standard constraint areas (all but OTHER_OUTLINE) with holes will lose the holes when export to IDF.



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

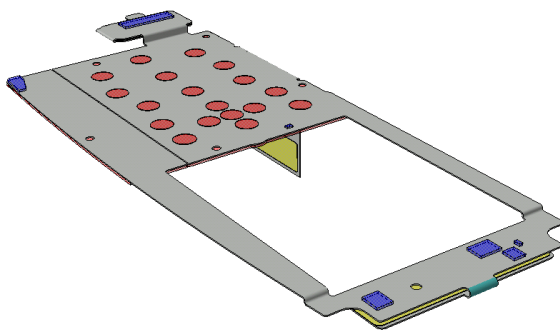


Switch to Unfold View

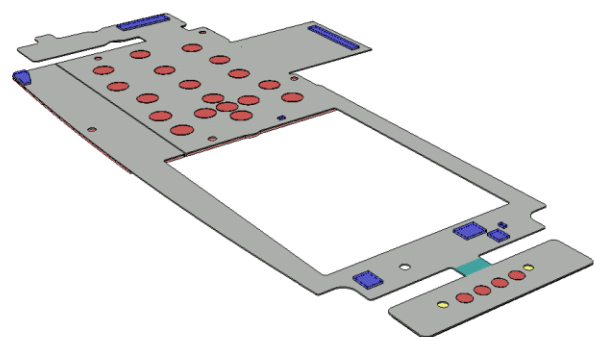


1. Select function "Fold/Unfold" in "Flexible PCB" toolbar to flatten the flexible PCB part.

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



Folded View



Unfolded View

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.

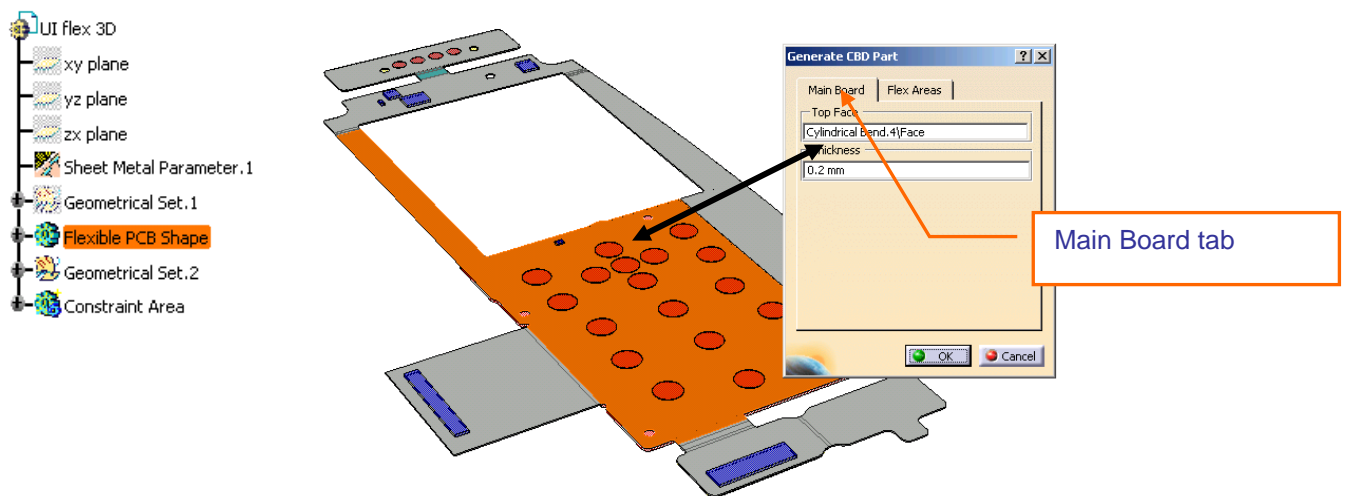
Generation of the Flexible PCB Part

The creation of the flexible PCB Part must be done in the unfolded view. Make sure you are 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

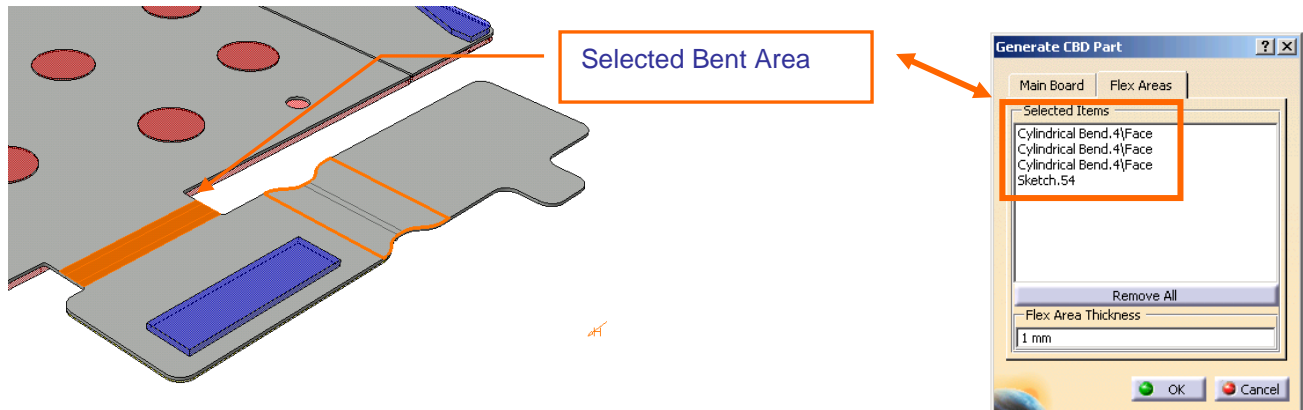


3. Choose the Top of the Board by selecting any face on it

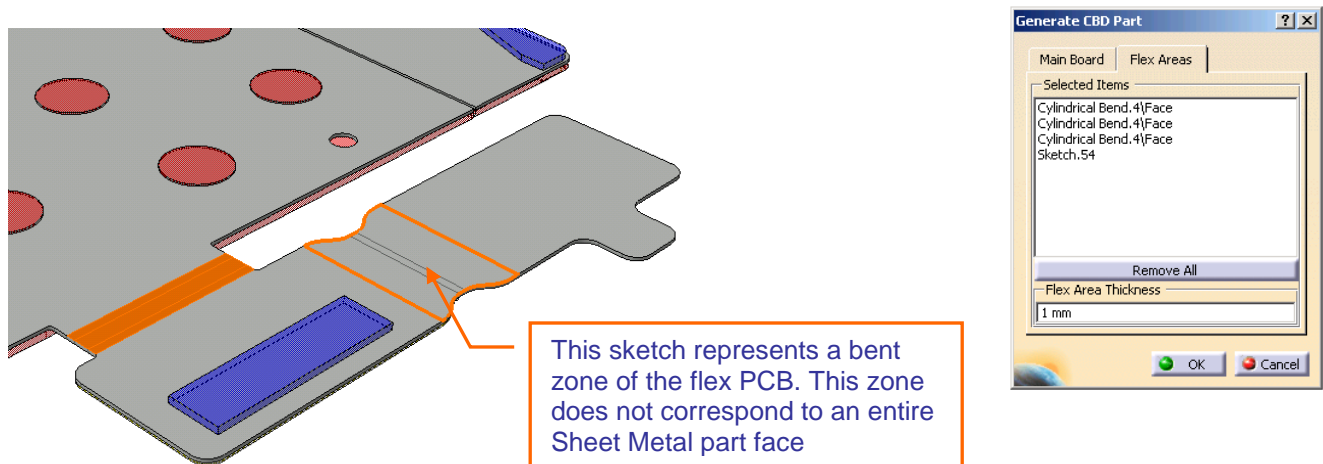
By default, the thickness of the Flexible PCB in the panel is equal to the Sheet Metal Part's thickness. It can be modified.

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



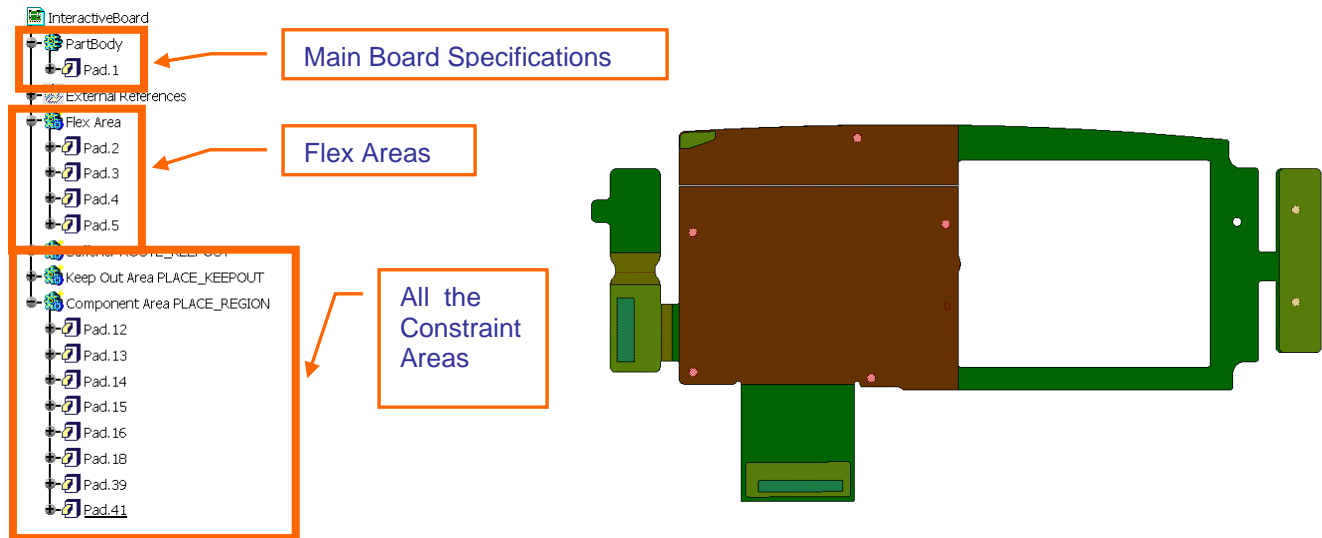
6. For a bent area which does not correspond to an entire face of the sheet metal model, it can be done through a sketch created in the flat view. This sketch can then be selected in the same Flex Area Tab.



7. The default thickness is set to 1mm and can be modified
8. Select OK to validate selection

Generation of the Flexible PCB Part

A new 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 part 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". User can change this type by modifying the constraint area configuration file FP9Configuration.txt. The column 8 is added and the meaning is as following:

- 0 : One Closed Profile
- 1 : Multiply Domain
- F : f or flex area

Warnings:

1. The default type of Flexible Area is VIA_KEEPOUT
2. In Constraint Area type configuration file, if the value in column 8 is F, system will use this constraint area as flex area when creating CBD board.

Constraint areas will be created in InteractiveBoard.CATPart into body with the same naming as defined in the FP9configuration text file.

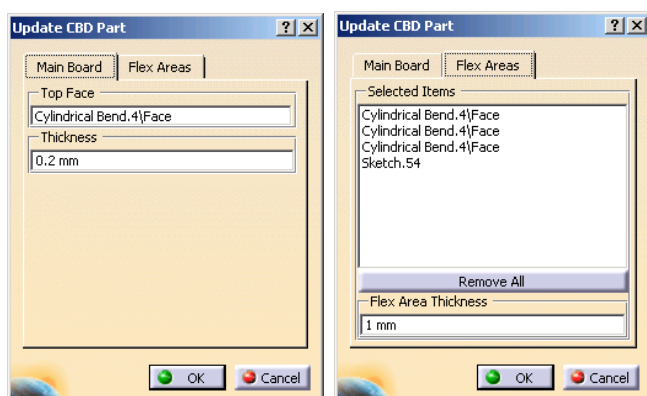
Update of the Flexible PCB Part

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.

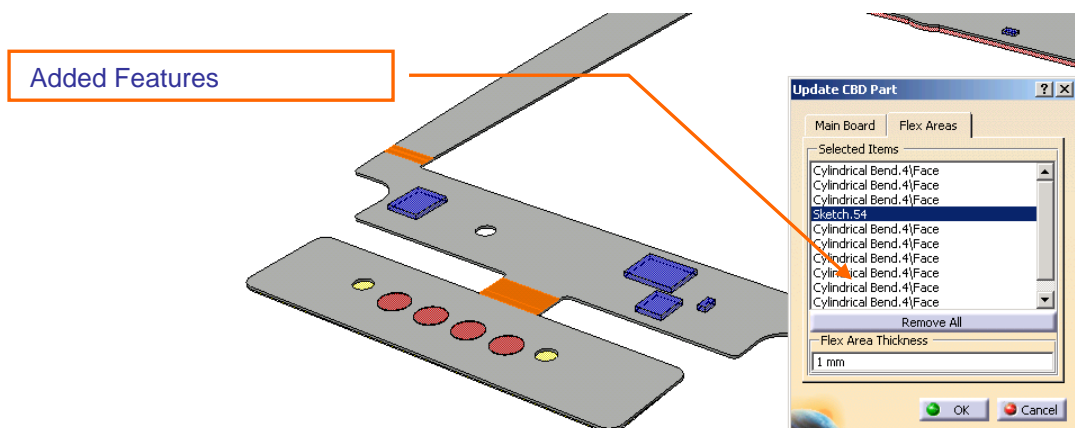
1. Select function "Update CBD Part"



The features already selected and set parameters are retrieved automatically.



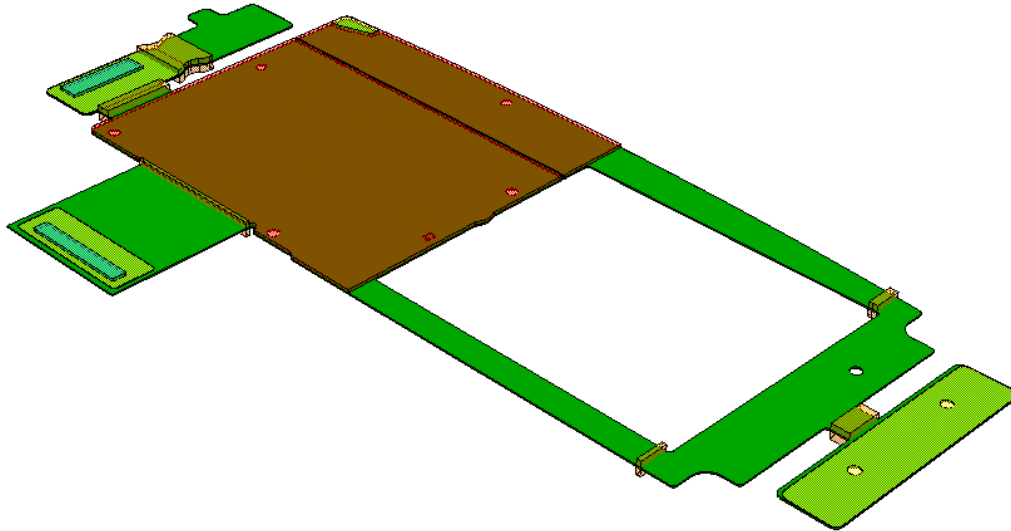
2. Select the flexible Area to add in the flex area index



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



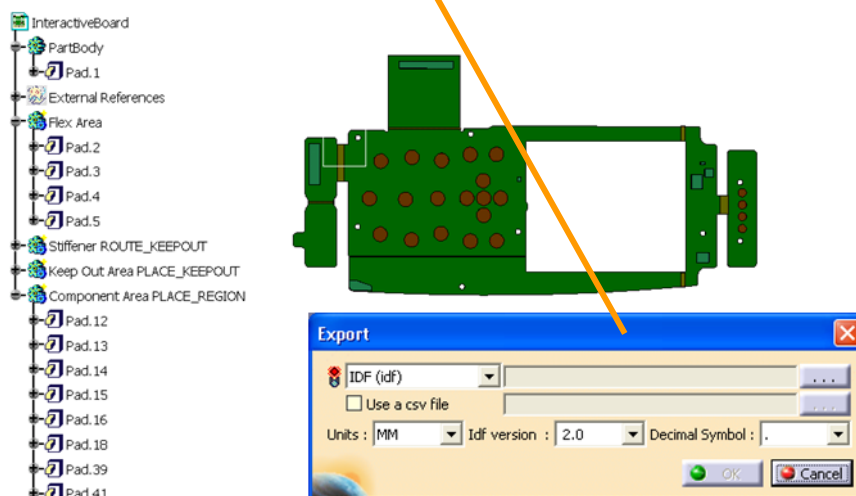
“Generate CBD Part” function supports flexible PCBs whose flat representation lies on top of a basic plane (XY, YZ or XZ) or a plane which is an offset of a basic plane

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



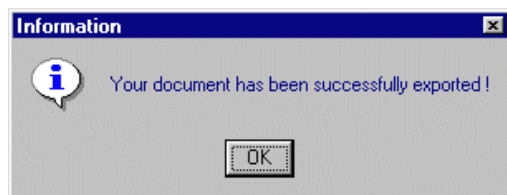
You can choose between three extensions for the IDF type:

- .idf: the general format
- .brd: the MG specific format
- .bdf: the Allegro specific format

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 (,)

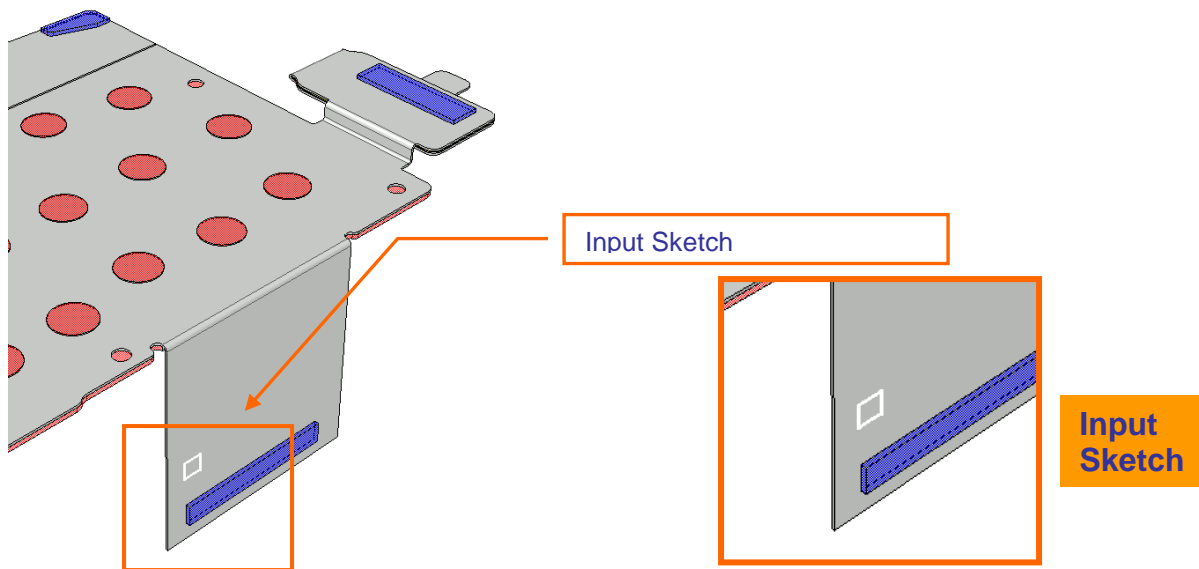
3. Click OK to perform the export. An Information message is displayed:



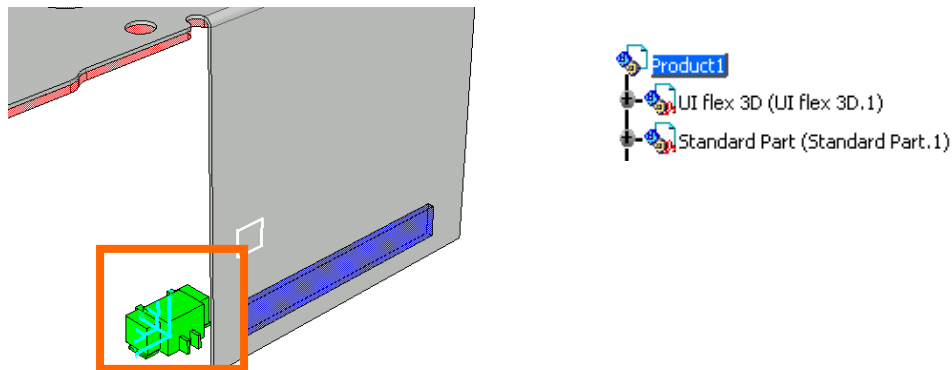
Placing Standard Component with Input sketch

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

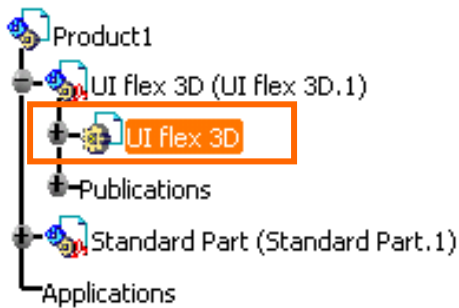
This sketch corresponds to the location of the standard part to be instantiated. It can be a profile already used for a Component placement constraint area. The standard part local axis system will be centered on the barycenter of the sketch with Z axis orthogonal to the sheet metal face.



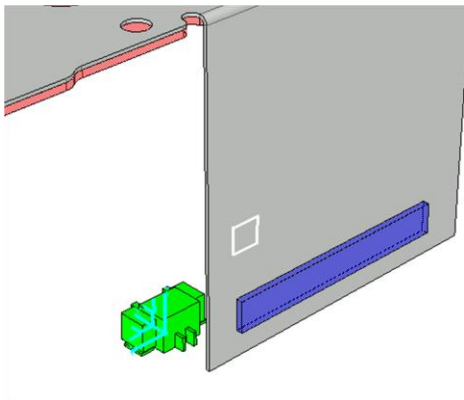
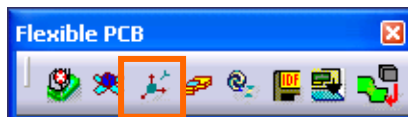
2. Instantiate the standard component at the assembly level



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



When the Axis of the standard part is not published, it must be selected by hand by the end user

Option to hide axis created by the command

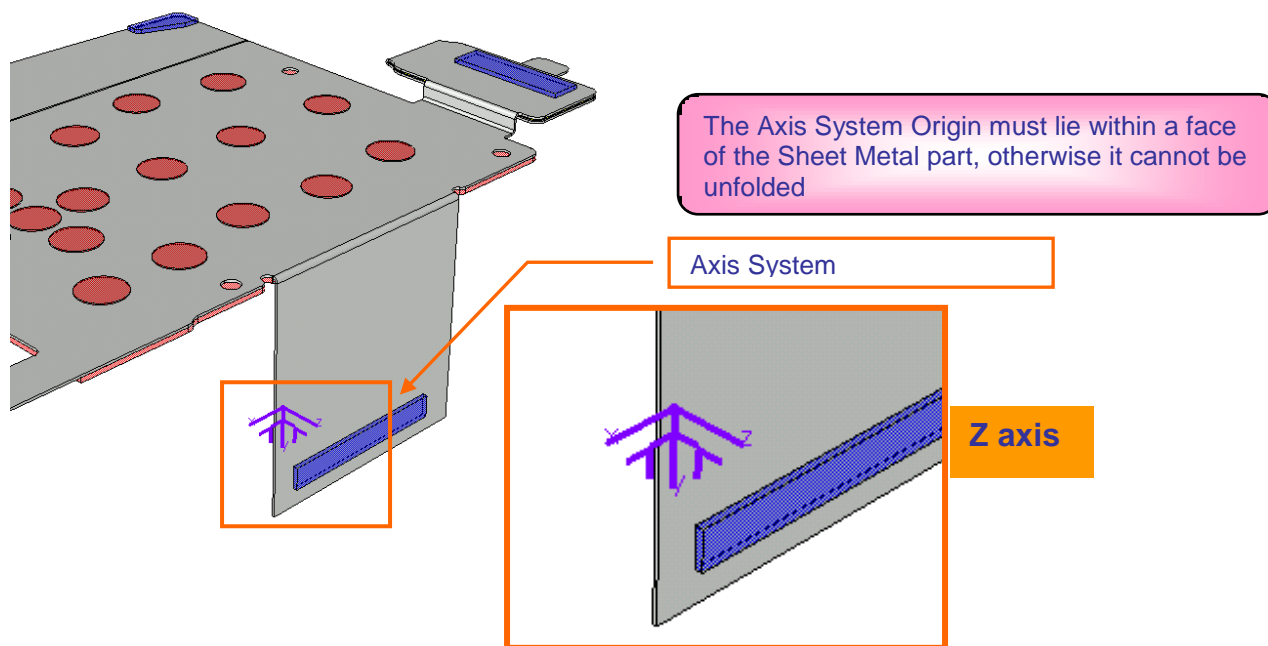
7. Select OK

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

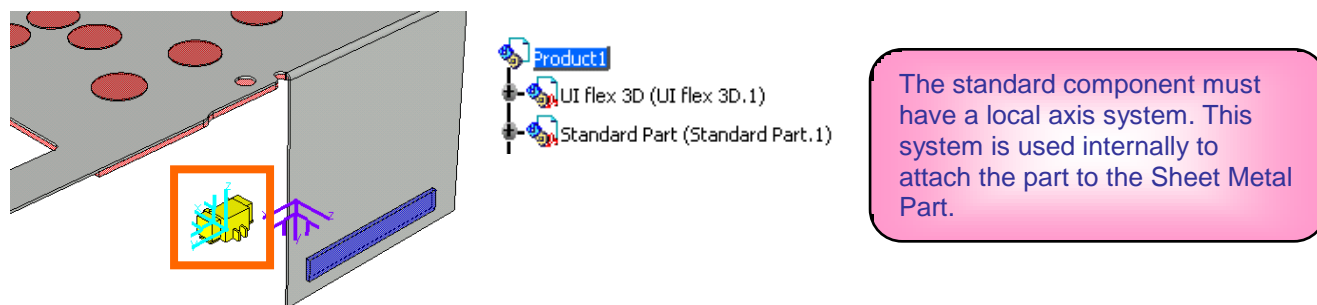
Placing Standard Component with Axis Systems

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.

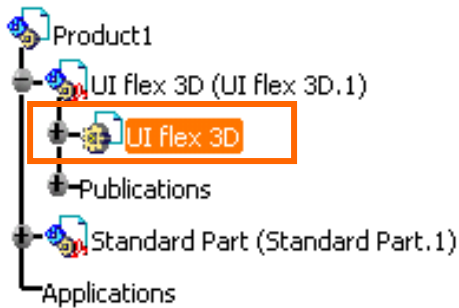


2. Instantiate the standard component at the assembly level or into a sub assembly



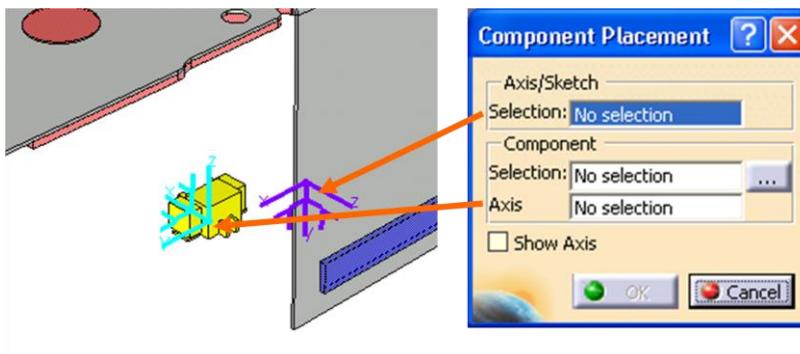
Everything is ready for component placement on the Sheet Metal Master part

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



The attachment of the standard component is done at the sheet metal part level

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

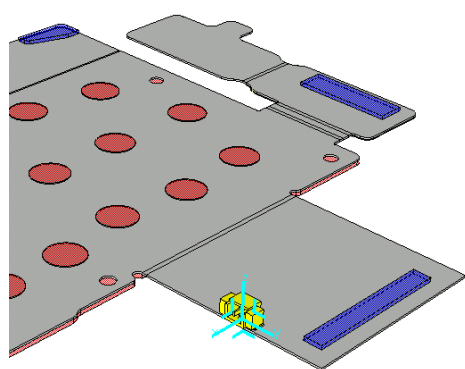
A group is created in the sheet metal master part. This group is linked to the axis system.1.

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

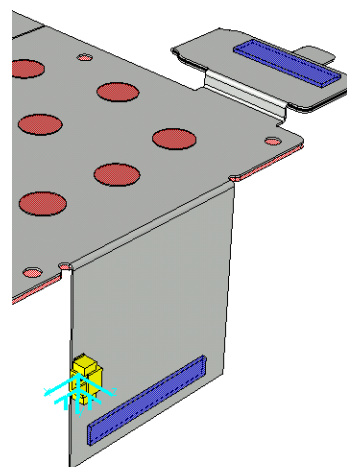
An assembly constraint, Coincidence “Axis to Axis” is created between the Standard Part Axis system and the Sheet Metal part “moving axis”. This coincidence can be modified manually with any other assembly constraints such as offsets.



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




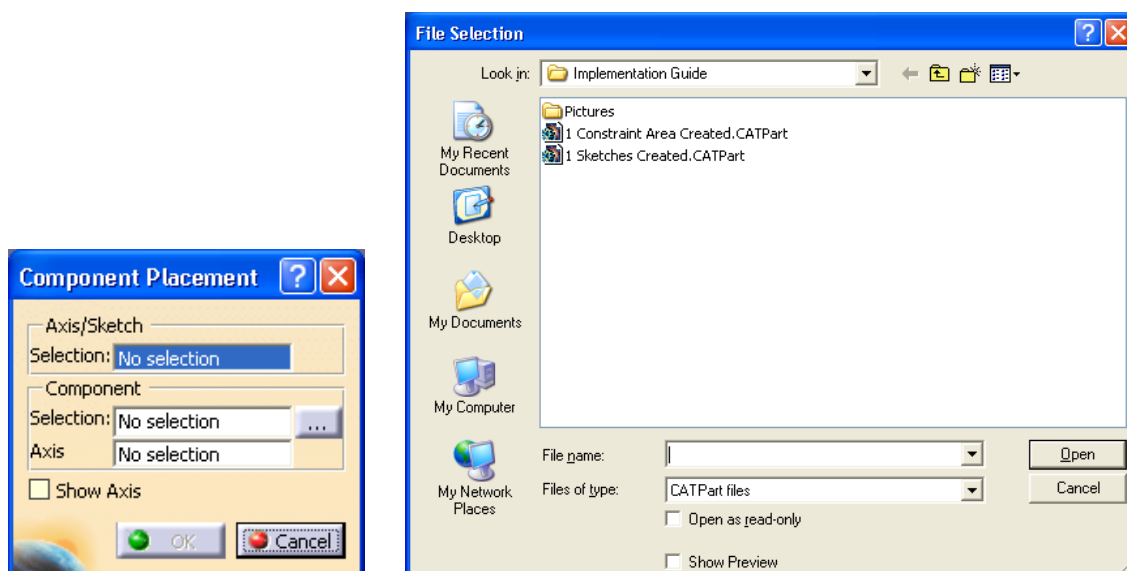
Unfolded view



Folded view

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

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.

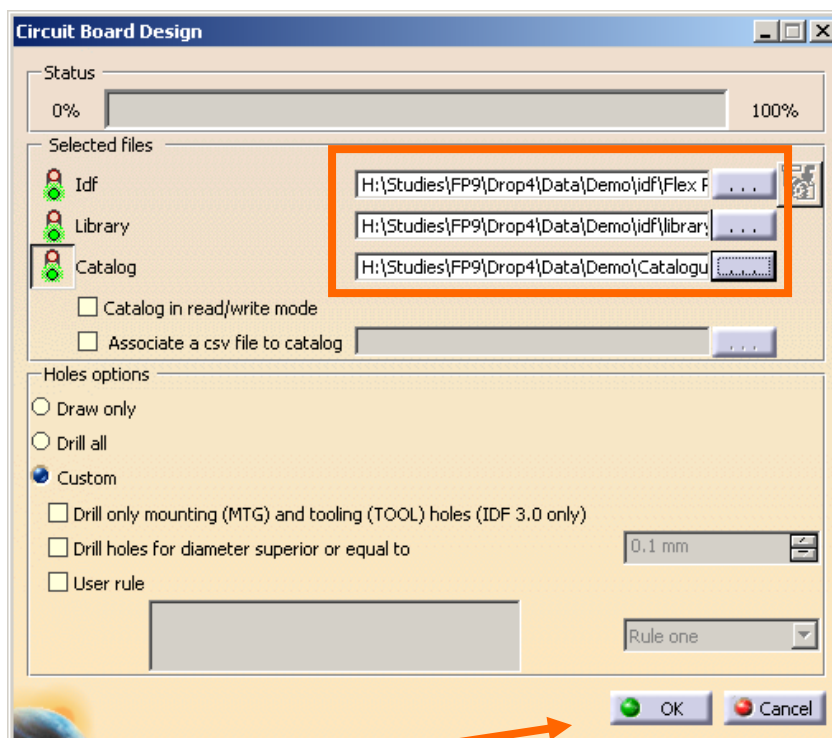
However, none of these two files are mandatory.

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:

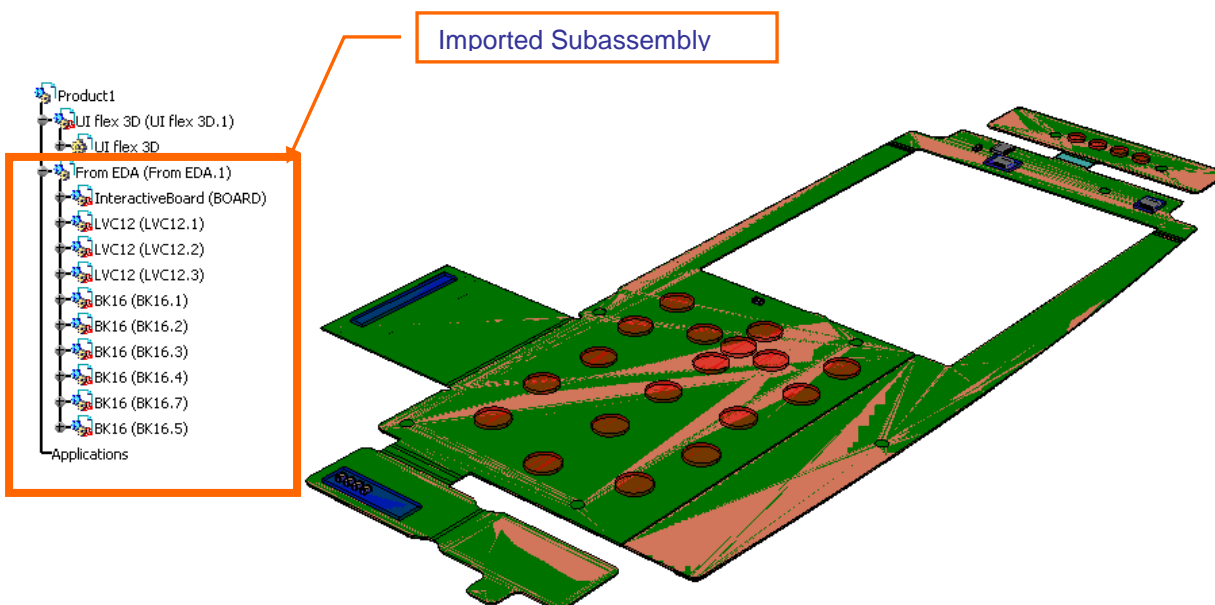
- if the file is mandatory, the light is red 
- if the file is optional, the light is yellow 
- if your selection is valid, the light turns green. 



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.

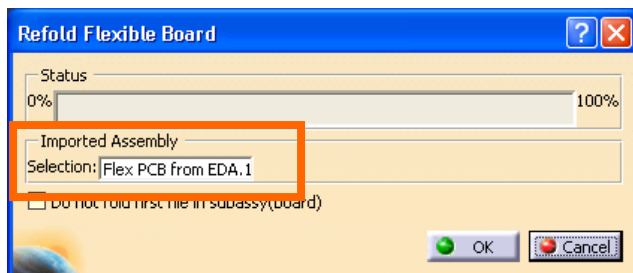


Refold Flexible Board

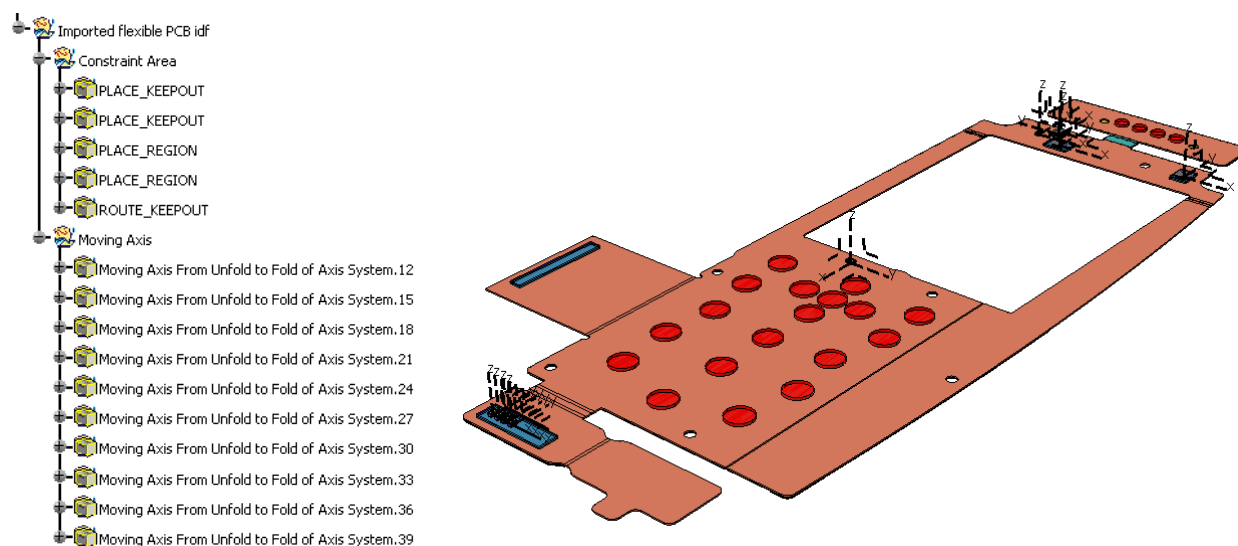
8. Select "Refold" function



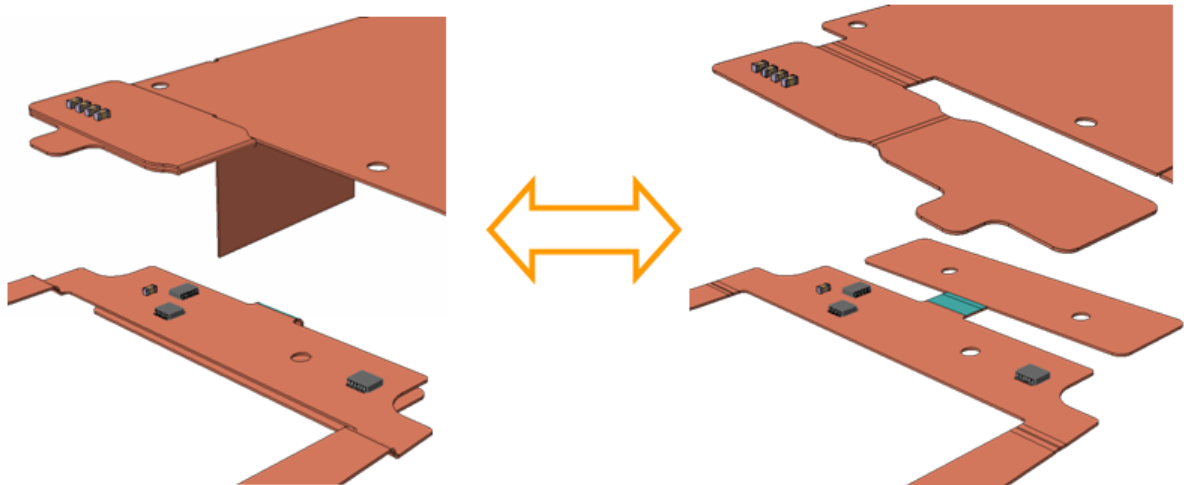
9. Select the imported subassembly and click OK



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.