

Progressive Die Strip Layout Design (SL9)

BPA Delivery 7 for V5R19 (V5.7)

Implementation Guide

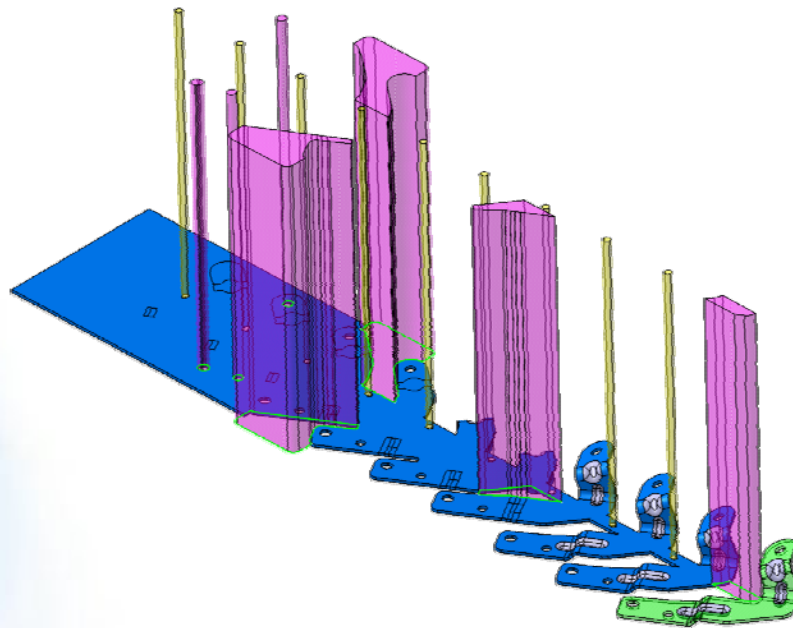


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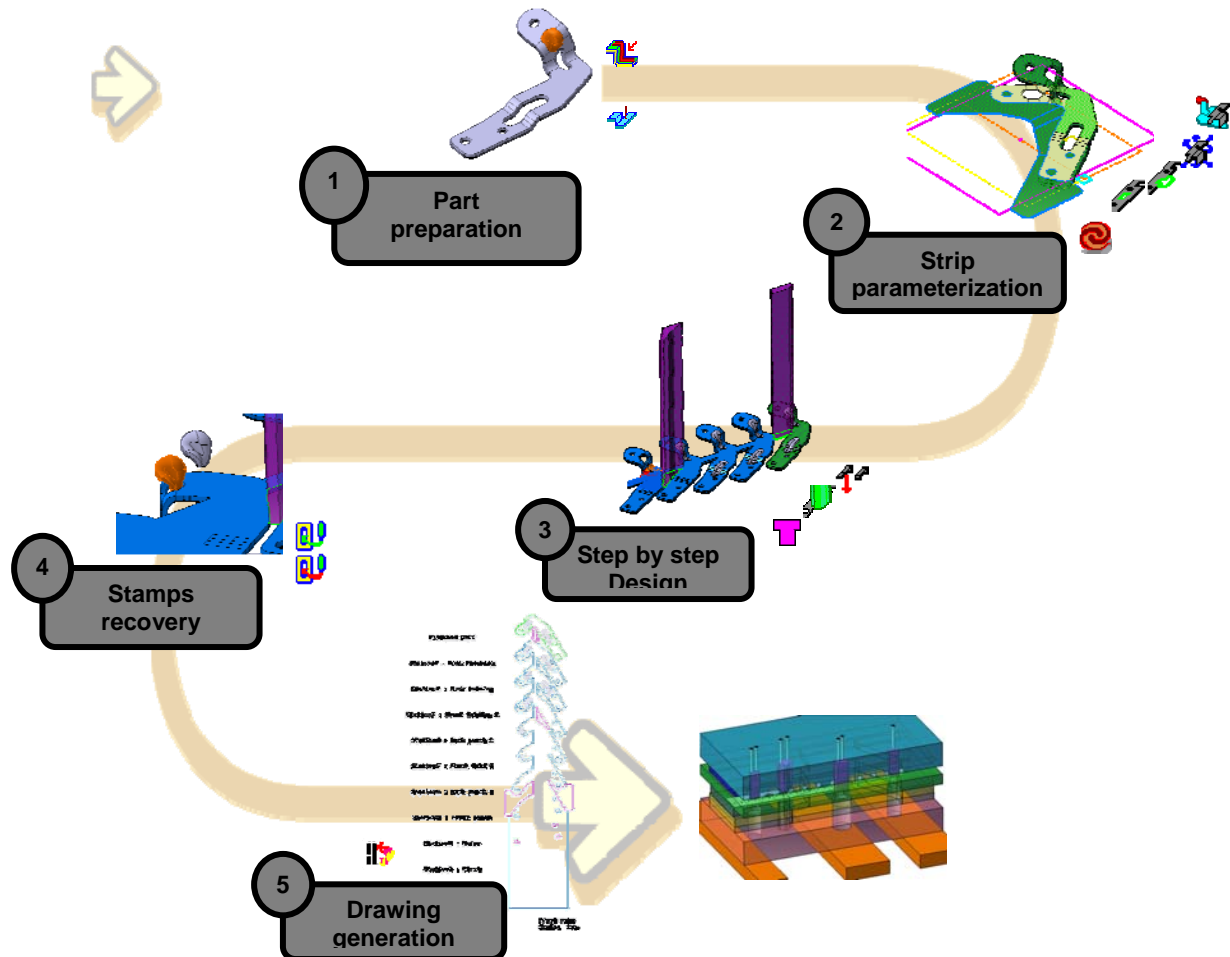
Introduction

- This document is part of a set of documents that will develop a CATIA V5 methodology to define a Strip Layout for Progressive Dies business.
- These documents will describe
 - 👤 All the specific functions developed for the Strip Layout methodology,
 - 👤 A scenario that will be used for demo or a step by step exercise for training, **this document**
 - 👤 A different scenario, could be used as speaker text to support the delivered video,
 - 👤 Some hints and tips,
- The user will run all the developed functions thru CATIA automatisms like Assembly Templates, Power Copies, User Defined Features, Macros.
- The user needs to know how to use the basic functions of CATIA if he wants to understand this entire guide.

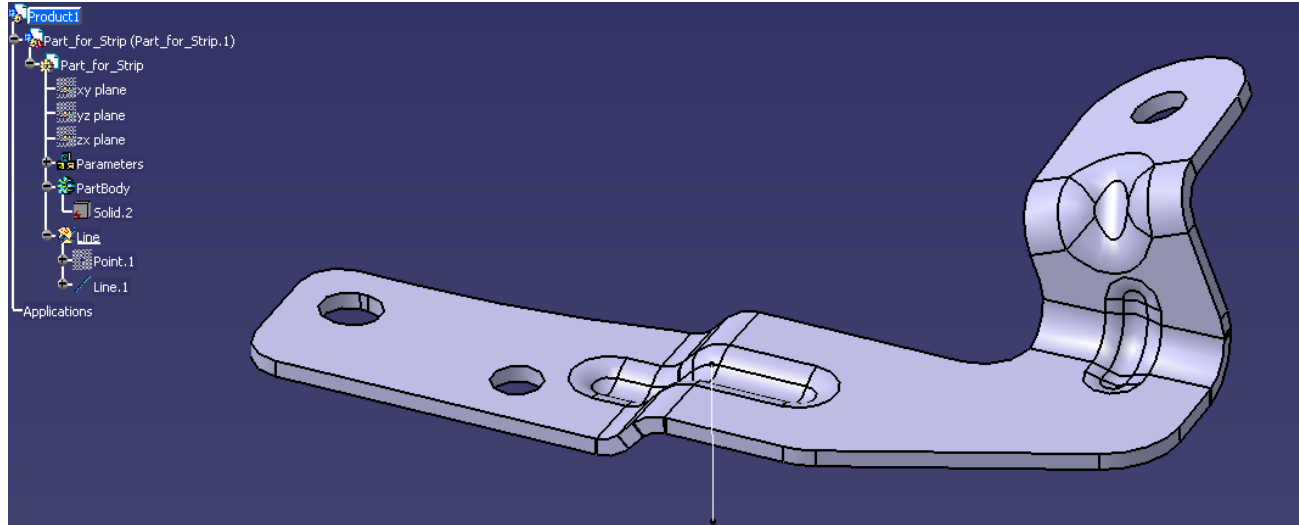
Aim of the methodology

- The aim of this methodology is to build the strip layout design of a metal part. Starting from an imported IGES part or from a V5 part, this methodology, based on knowledge template and macro tools, will enable the user to design the strip layout with many automatisms.
- A Part respecting lengths and dimensions will be created or an existing IGES or CATIA V4/V5 Part will be used.
- The strip layout design will be created in reverse direction from the finished part to the starting plate.
- Then the final result may be used to design the Progressive Stamping Die.

Strip Layout Methodology description

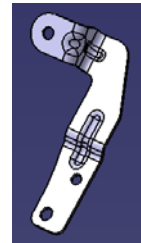


Part to produce creation



Introduction

- **This methodology works in opposite direction of the industrial process:**
 - 👤 In the industrial process, the strip layout begins with a metal plate to end with finished product
 - 👤 Whereas, in the strip layout methodology, we begin with the finished product and we go back step by step until the starting plate in order to design all the strip
- **So, the finished part is required to begin the methodology**
- **To have the finished part, you have a lot of possibilities:**
 - 👤 Create the part with Generative Sheet Metal Design features,
 - 👤 Create the part with CATIA V5 Part, GSD, ... features,
 - 👤 Or, retrieve the part from IGES, STEP, ... or CATIA V4
- **The creation or the import of this part will not be detailed in this methodology because it is not specific to strip layout**



Creation of the finished part

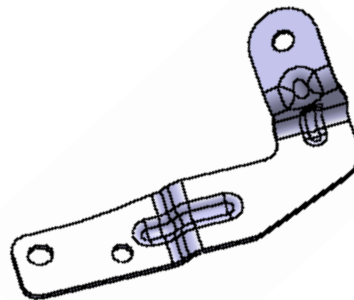
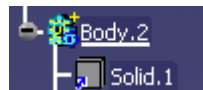
- To create this part with **Generative Sheetmetal Design** features:

- 👤 In the Generative Sheetmetal Design workbench:
 - 🔧 Use *Generative Sheetmetal* tools:
 - 🔧 Wall, Wall on Edge, Extrusion...
 - 🔧 Flanges, Stamps...

Or

- Import this part from an **IGES or V4**:

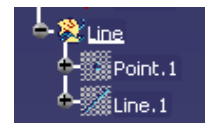
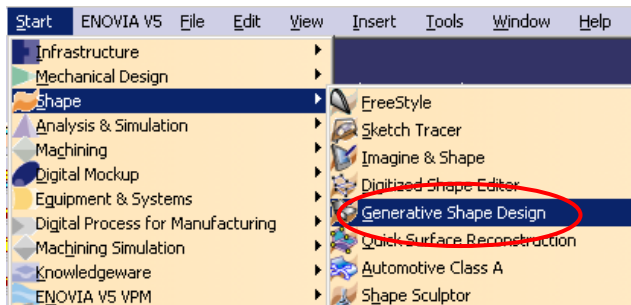
- 👤 The imported solid is a datum solid.



- **MANDATORY:** If you want to use a part with more than one body, do not use "Removed" as body name. This name is used by the automation functions, using it may interfere the complex geometries treatment.

Methodology starting point creation





- In order to use some automatisms, the Part must have some specific publications (A solid and a Line). So, whatever it is, a IGES, a V4 or a Sheetmetal Design Part, we must retrieve these data.
Be sure that the part is in the correct orientation,
- **MANDATORY, the upper face of the final strip will be coincident with the XY plane.**
- In Generative Shape Design workbench create a line which will manage the orientation :
 - 👤 Switch to Generative Shape Design workbench.
 - 👤 In the body containing the previously created or imported solid (or in a geometrical set), **create a point** which will be used for the line creation
 - 👤 In the same body (or geometrical set), **create a line** passing by the previously created point and along Z-Axis in order to define the strip orientation (This line will be the rotation axis used to place the part in the good position for the future X-Axis strip design). It will be done later with the instantiation of a template







Part preparation for strip layout

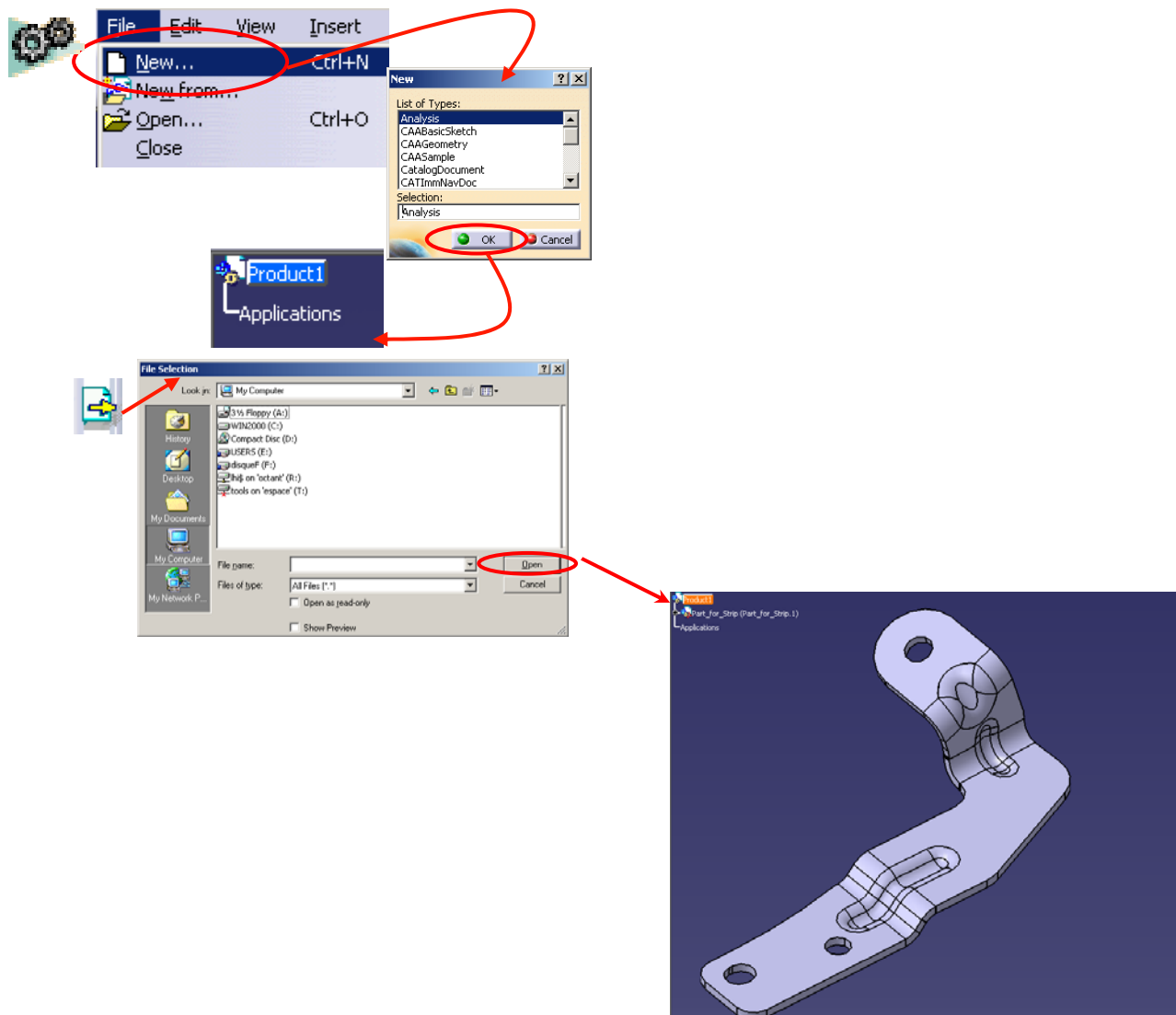
Preparation of the structure (1 / 2)

Open a new Product:









-  Select **File / New...**
 -  The "New" window is displayed.
-  Select **Product** in the "New" window and validate with **OK**
 -  An empty product is displayed in CATIA in the Assembly Design workbench.

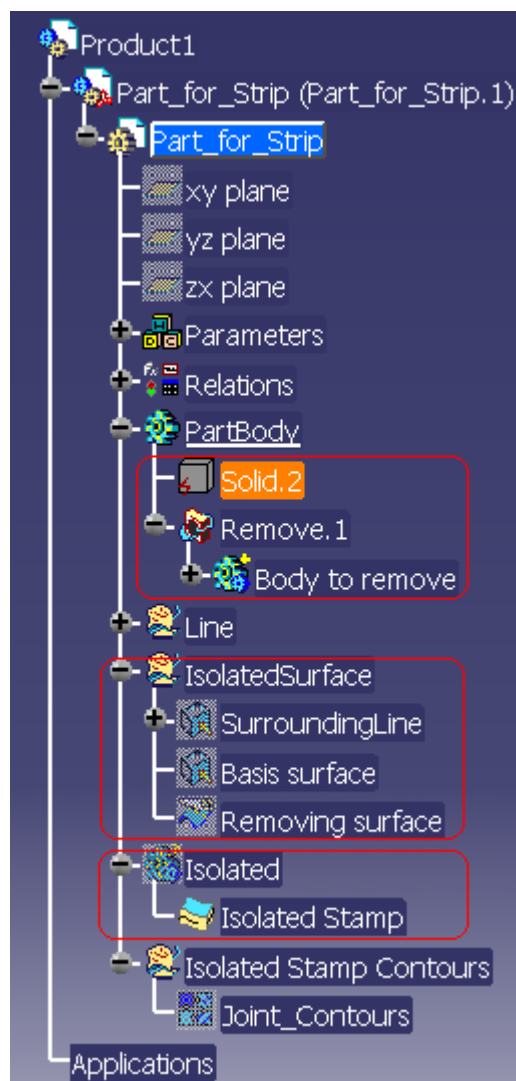
Insert the finished part previously created or the file Part_for_strip.CATPart given with the methodology:

-  Select the **Existing Component** icon and select the new product in the tree
 -  The "File Selection" window is displayed.
-  Open the part to be striped.
 -  The component is inserted in the new product



Isolation of the complex geometries

-  As the Recognize function is unable to recognize all the sheet metal geometries, you need to separate those areas of your part. You will use for that 3 specific function:
 -  3D stamp removing
 -  This function helps you to isolate 2D and 3D stamps
 -  Rib removing
 -  This function helps you to isolate ribs
 -  Contour removing
 -  This function helps you to remove any geometry from a joined contour. The two previous functions are particular applications of this one.
-  **The result of this function is always the same. You will find, as result, a remove feature in your part's solid, a geometrical set containing the needed surfaces for the operation and a body named "Removed" containing the isolated area.**

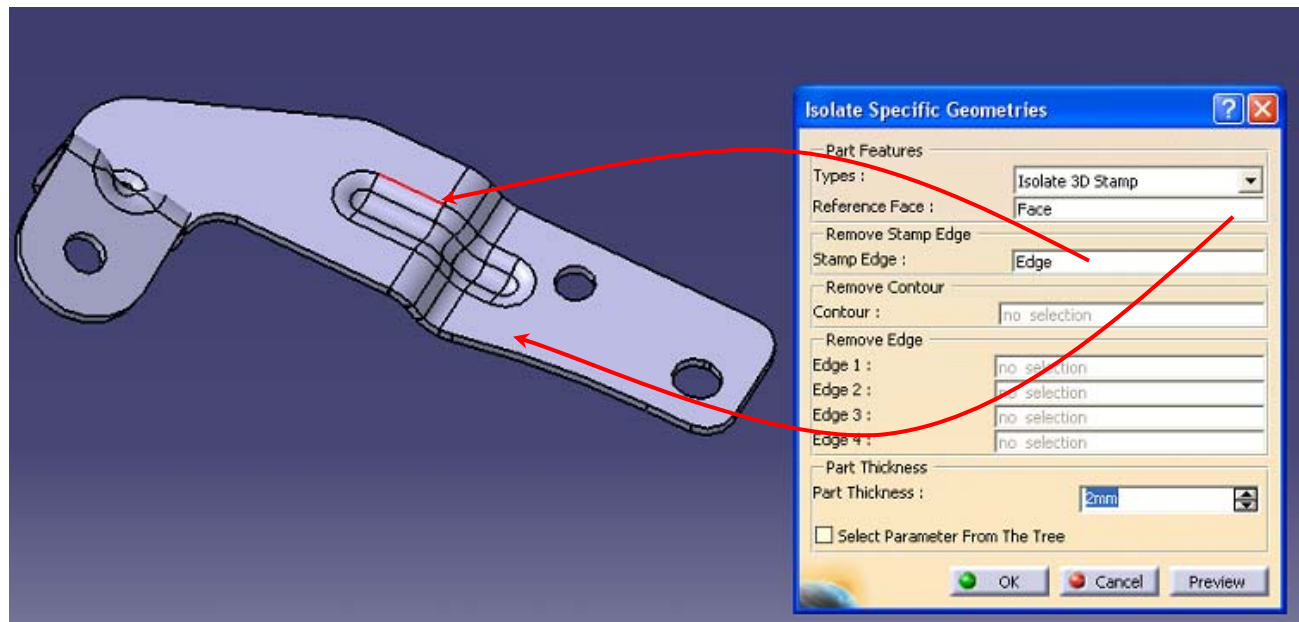


-  **Mandatory: You need to work in a product structure for all the functions to work.**

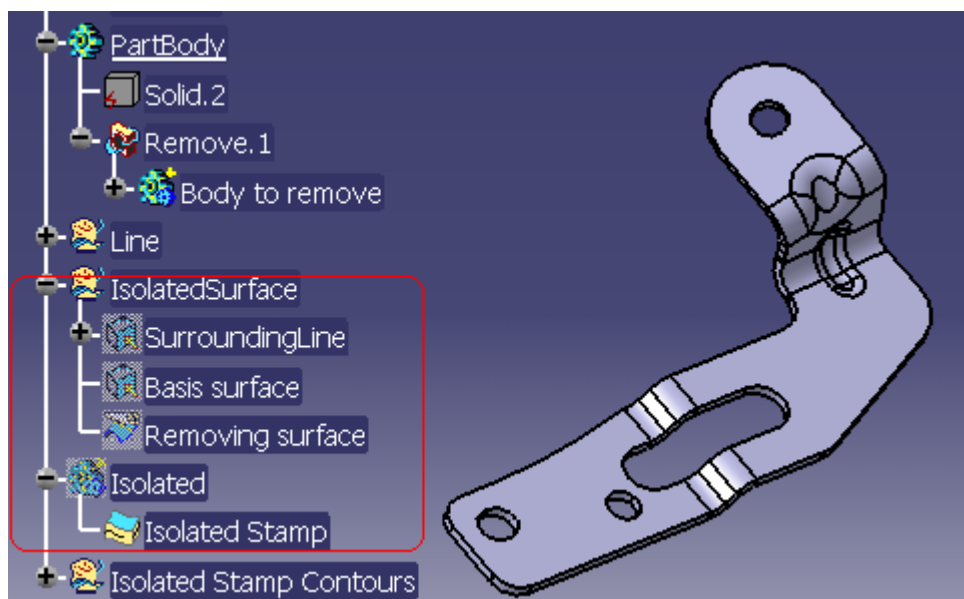
Removing Stamps

➤ This function is specialized in 2D and 3D stamps removing.

- 👤 Launch the "Isolate Specific Geometries" function
- 👤 Select the body in which your working part is.
- 👤 Select a stamp edge, a reference face, and pick the part thickness parameter.



- 👤 Press OK
- 👤 The stamp is isolated

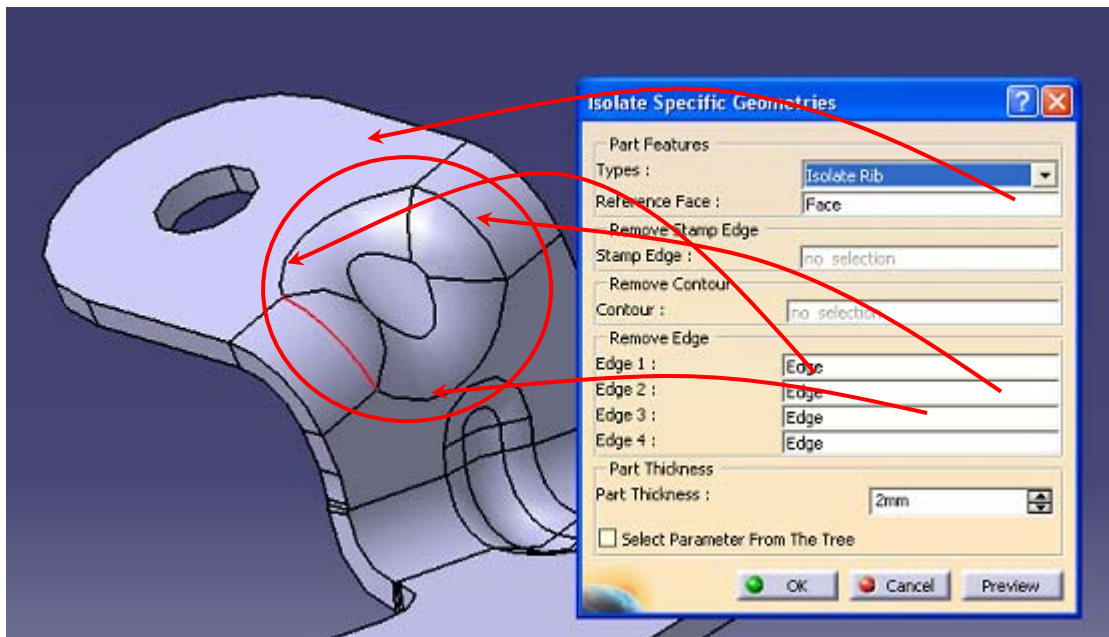


Removing Ribs

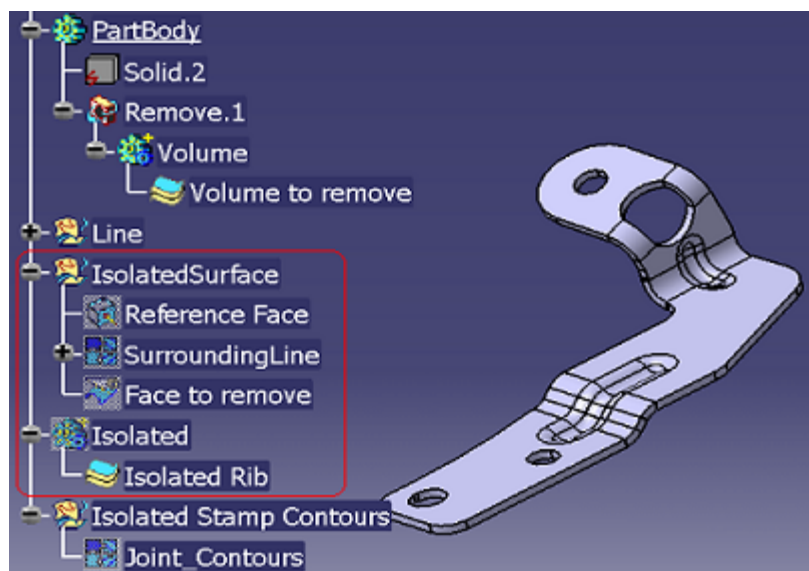


➤ This function is specialized in rib removing.

- 👤 Launch the “Isolate Specific Geometries” function
- 👤 Select the body in which your working part is.
- 👤 Select a reference face, the four edges of your ribs, and pick the part thickness parameter.



- 👤 Press OK
- 👤 The Rib is isolated

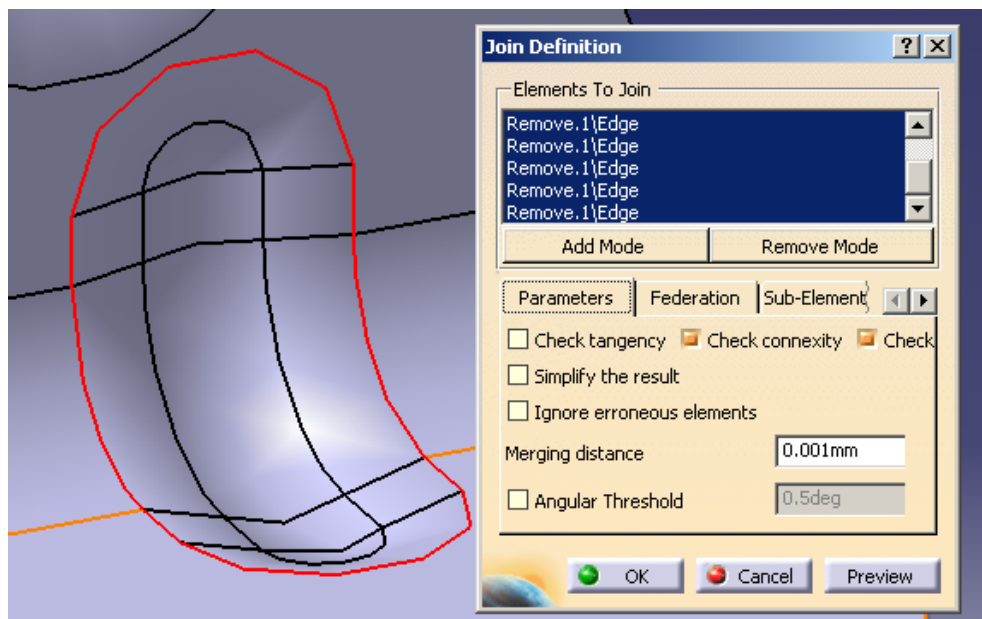







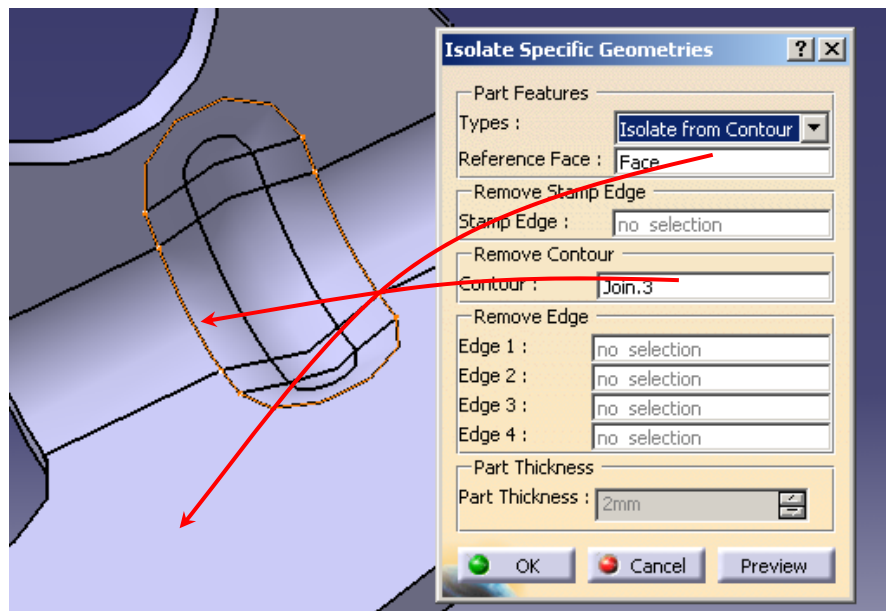
Removing complex geometries

- This function enables you to isolate hardly every kind of geometry, just entering its joined contour and the part's thickness.

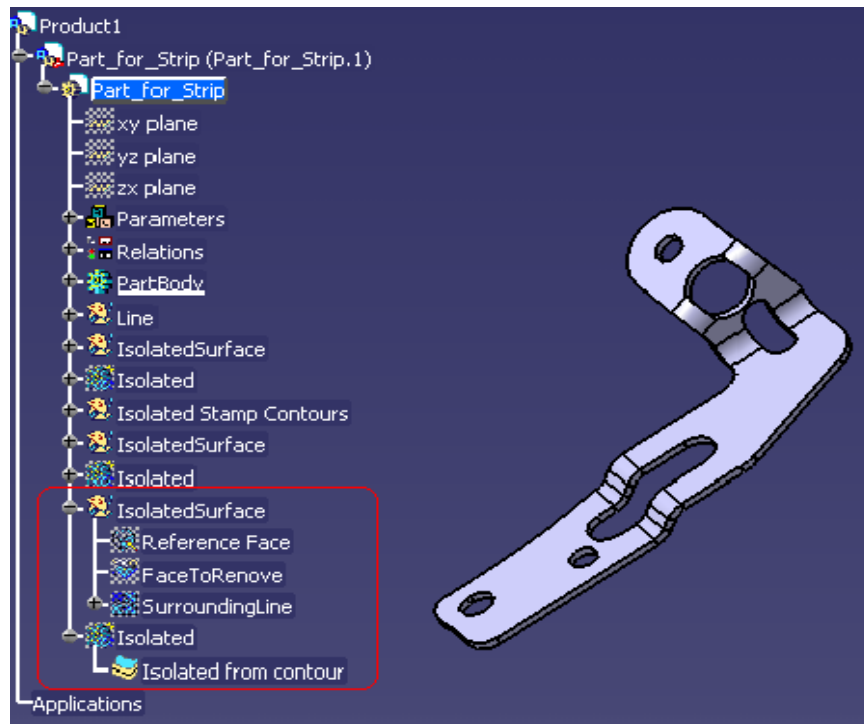
 Click on the "Join" icon and pick all the edges of your geometry's contour.



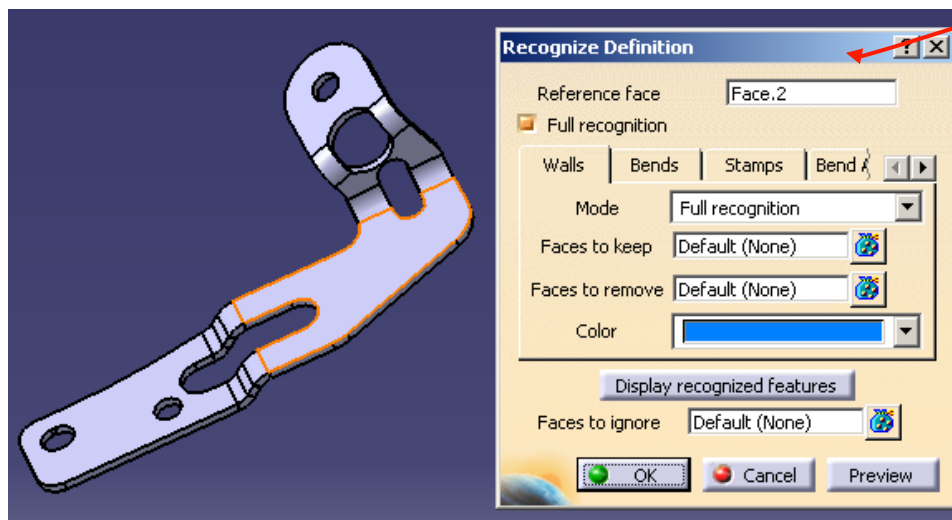
-  Launch the "Isolate Specific Geometries" function
-  Select the body in which your working part is.
-  Select a reference face, the contour of your geometry, and pick the part thickness parameter.



- Click OK
- The geometry is isolated.



- Mandatory:** After isolation of complex geometries, Recognize has to be done if part is not sheet metal solid for further operations.

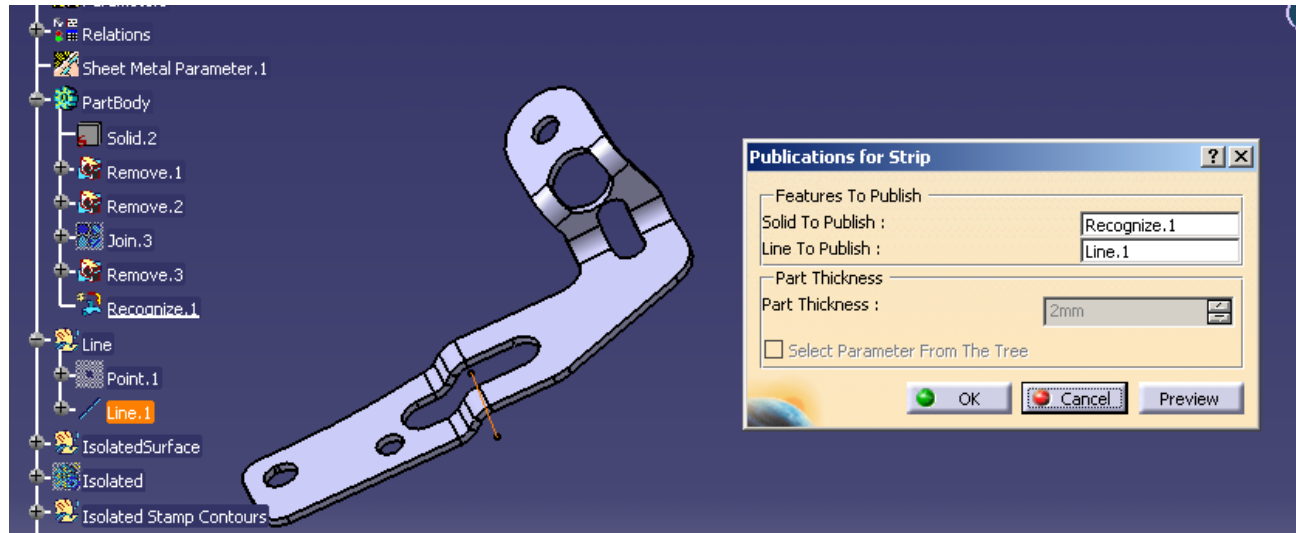


Publishing the elements



In order to use the previously created elements (a solid and a line), we must publish them.

Launch the “Prepare for strip” function.



Select your work body and your rotation line.

The work body is copied and pasted as result with link.

The work body, Join_Contour and the rotation line are published.



Save the part

If other solutions exist to create the finished part, the process is the same:

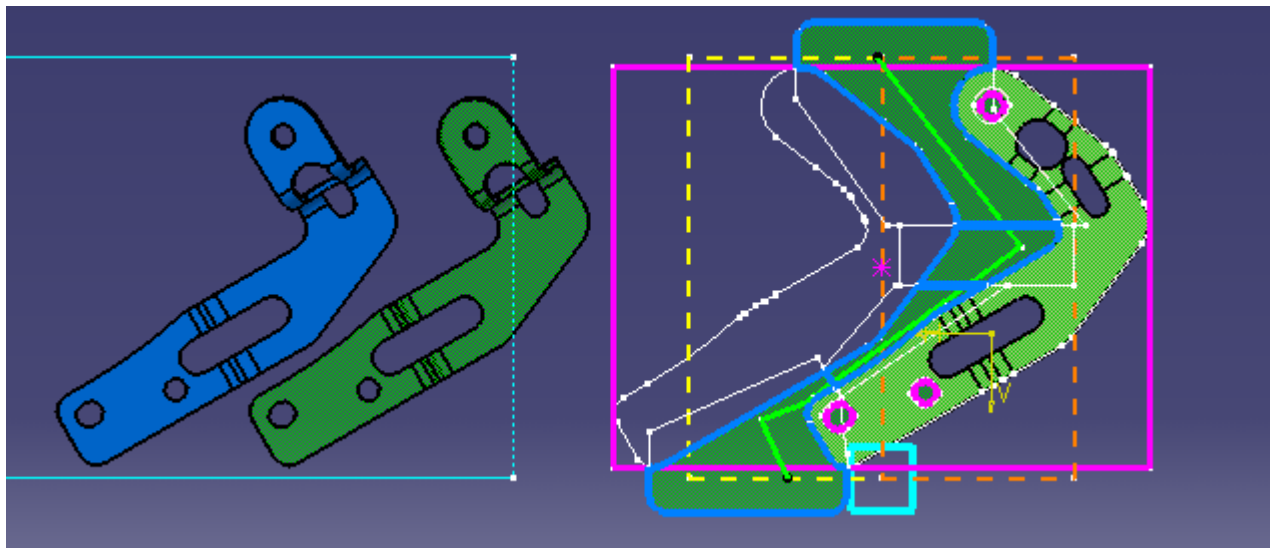
Create a Line

Make the publications of the created Solid and Line.

Strip Layout Preparation

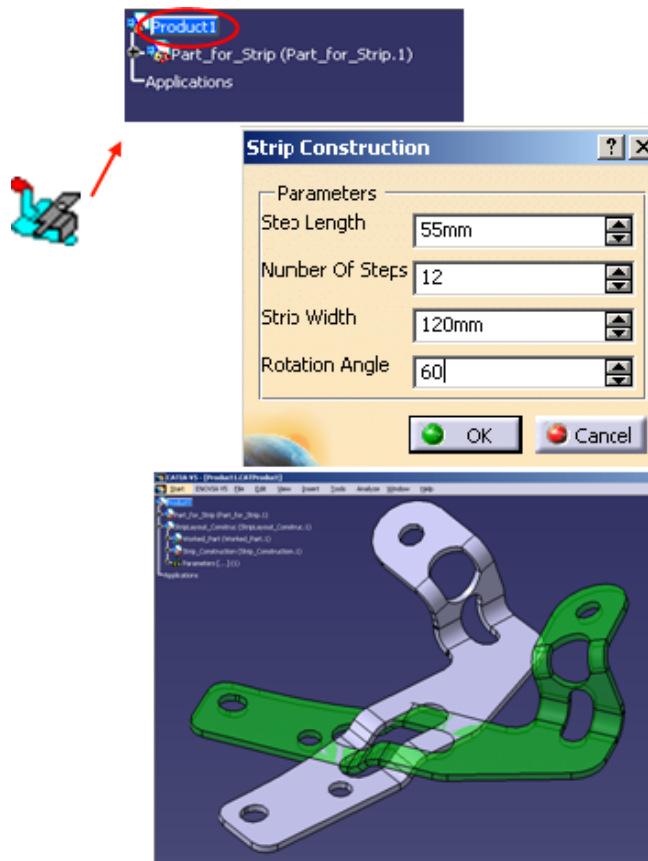
Introduction

- One of the great advantages of this methodology is the fact that many parameters can be changed when needed by the user
 - ⤴ The length between two consecutive steps can be parameterised if you have internal rules.
 - ⤴ The width of the strip layout can be changed,
 - ⤴ **The orientation of the part in relation to the X-axis can be chosen.**
 - ⤴ The user can customise embedding of two steps.
 - ⤴ The user can chose between two construction modes.
- So, the preparation phase of the strip layout is essential in order to lay the finished part as wanted and then design the strip layout.



Preparation of the structure (2 / 2)

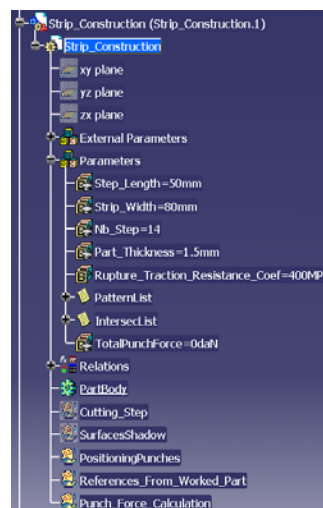
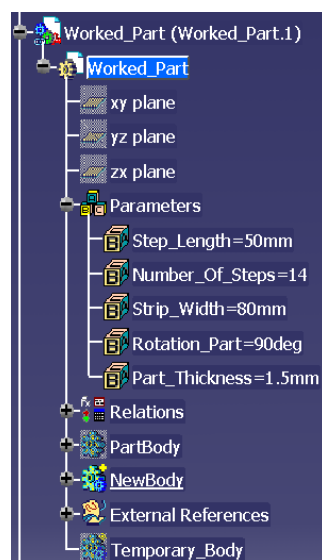
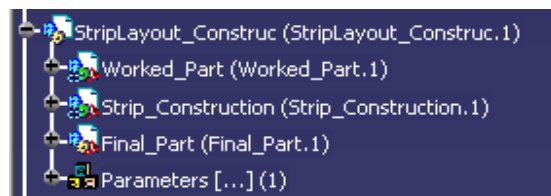
- Instantiate the Document Template published under the name of “StripLayoutConstruction”:
 - 👤 Edit the new product by double-clicking on it in the tree.
 - 👤 *The new product is highlighted in blue in the tree*
 - 👤 Select the **Strip Layout Construct** function
 - 👤 Set the **Step_Length** parameter to **55mm**.
 - 👤 Set the **Number_Of_Steps** parameter to **12**.
 - 👤 Set the **Strip_Width** parameter to **120mm**.
 - 👤 To orientate the part along X-Axis set the **Rotation_Part** parameter to **60°**
 - 👤 **Close** the “Parameters” window
 - 👤 Validate with the **OK** button
 - 👤 *The instantiated “Worked_Part” is displayed (in transparency) and the strip layout tree is built.*



Structure description

- The “StripLayoutConstruction” document template instantiates a product document “StripLayout_Construc.CATProduct” made of two parts documents “Worked_Part.CATPart” and “Strip_Construction.CATPart”
- “Worked_Part.CATPart” contains the “Worked_Part” part on which the operations of retrieving punches contours and organization of strip layout will be done

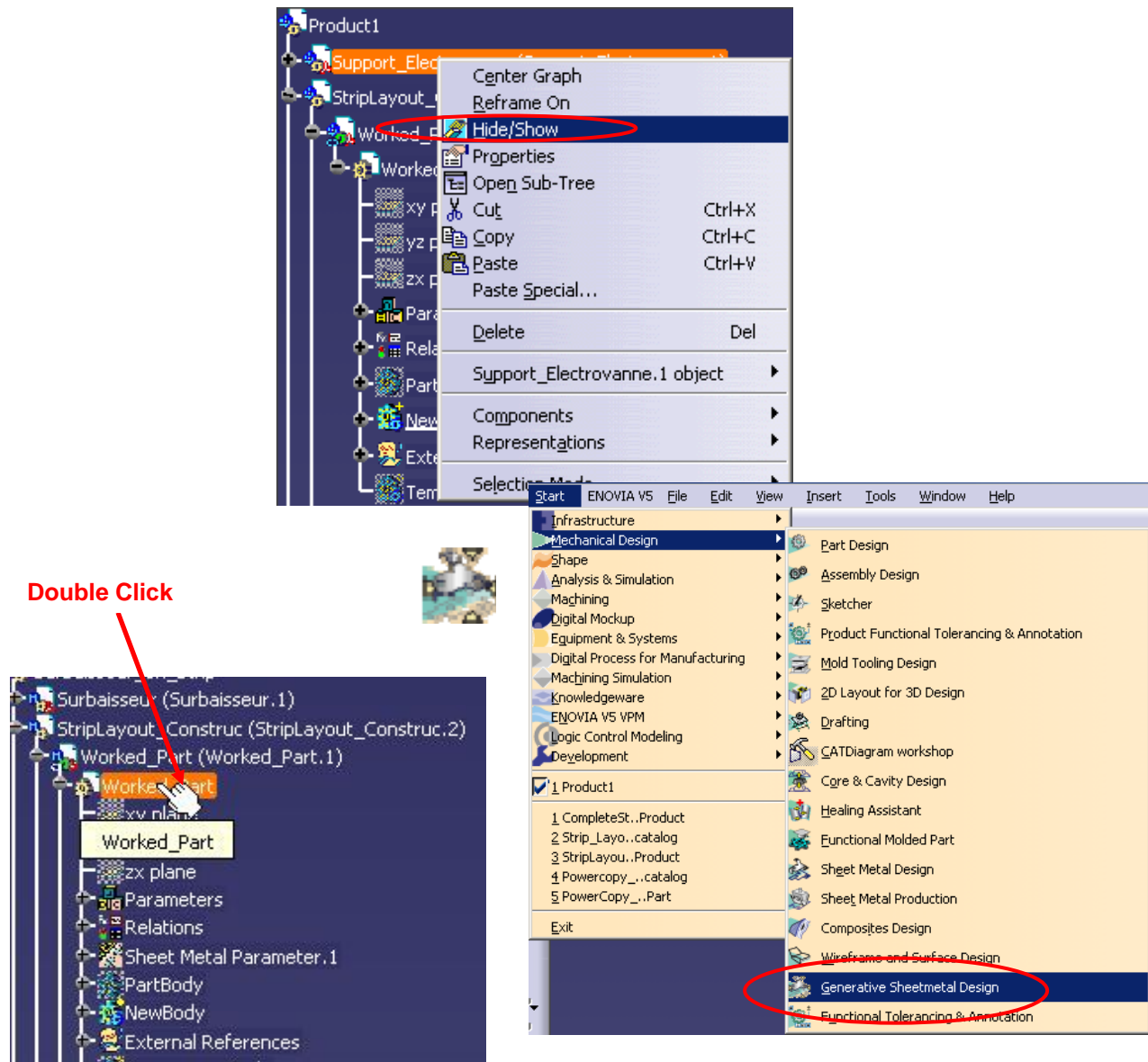
- “Strip_Construction.CATPart” contains the “Strip_Construction” part on which the different steps (mainly stamping and folding) will be defined one by one in order to design the Strip_Layout.
- “Final_Part.CATPart” will receive the final strip at the end of the process.
- Be sure the main partbody name is “PartBody”. This is mandatory in any language, if needed, rename it**



Parameters of the strip layout (1 / 4)

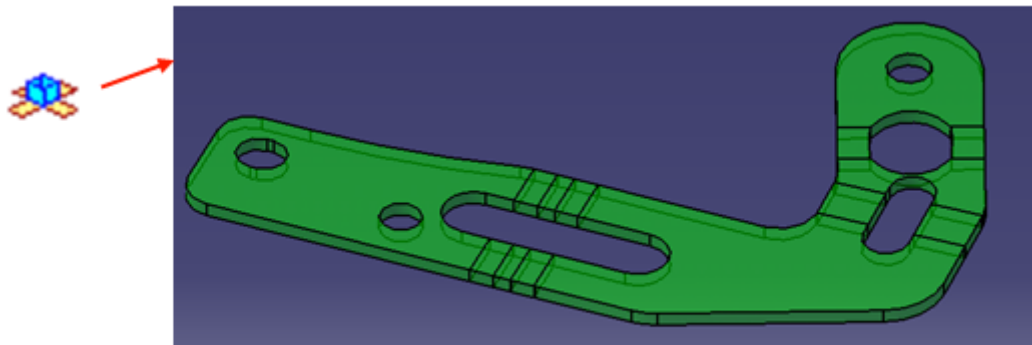
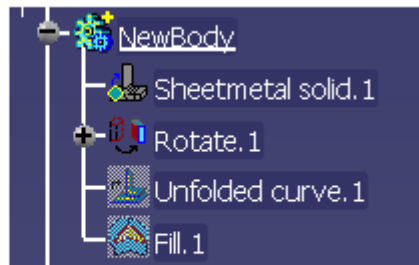
- Hide the component containing the finished part:**
 - Right-click on it and select **Hide/Show**
 - The component is no longer displayed
- Edit the “Worked_Part” part by double-clicking on it:**
 - The “Worked_Part” part is highlighted in blue.
 - You are no longer in “Assembly Design” workbench.
- Be sure to be in “Generative Sheetmetal Design” workbench**

- 👤 If it is not the case, switch to “Generative Sheetmetal Design” workbench by selecting **Start \ Mechanical Design \ Generative Sheetmetal Design**



Parameters of the strip layout (2 / 4)












- 👤 Verify that the body “NewBody” is in work object (underlined), if not right-click on it in the tree and select “Define in Work Object”.
 - 👤 The body “NewBody” is underlined.
- 👤 Unfold the part to get the unfolded result:
 - 👤 Select the **Fold/Unfold** icon
 - 👤 The unfolded result is displayed.

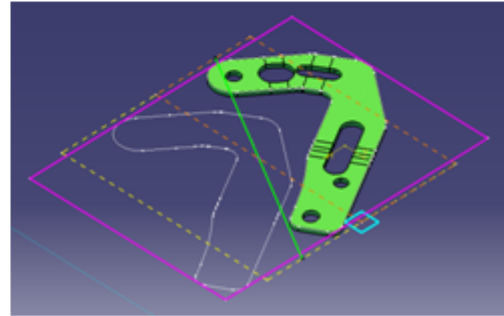
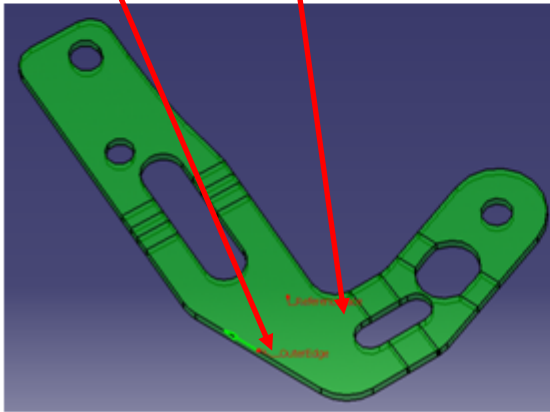
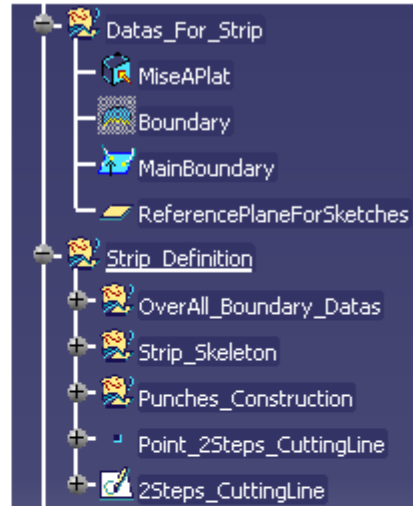
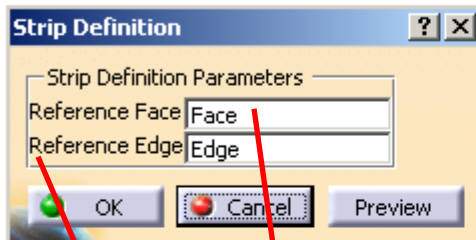


Parameters of the strip layout (3 / 4)



Creation of the strip layout preparation area:

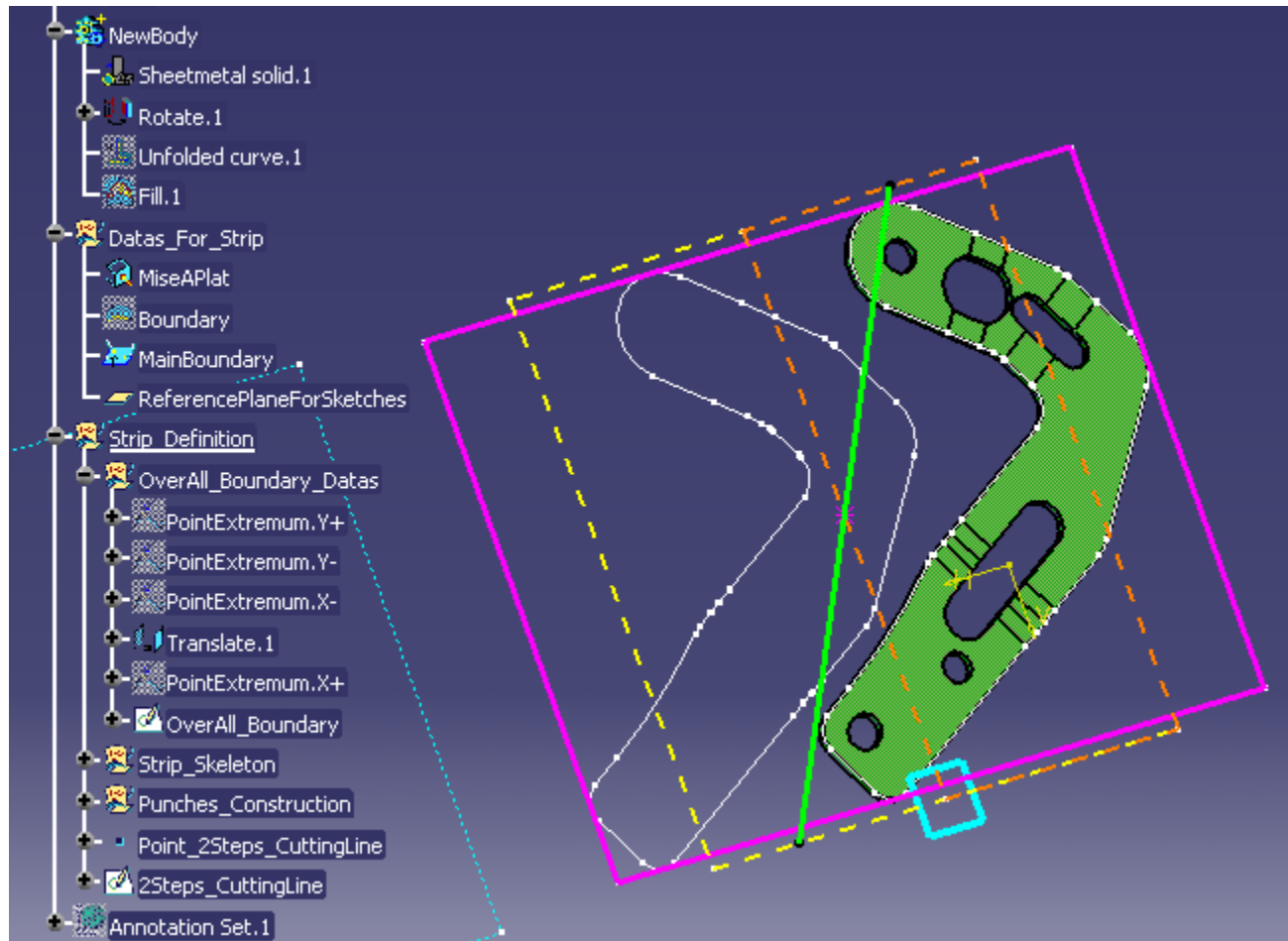
-  Select the function **Strip Definition**
 -  The "Insert Object" window is displayed
-  Select the **Use identical name** button
 -  Inputs have been selected.
-  Select the **face** as shown on the picture below
 -  The same face selected for the previous recognize operation
 -  The selected face is displayed as "ReferenceFace" in the list.
-  Select the **edge** as shown on the picture on the right
 -  The selected edge is displayed as "OuterEdge" in the list.
-  Validate with the **OK** button
 -  The strip layout preparation area is now displayed with the visualisation of the strip length in blue.



Parameters of the strip layout (4 / 4)

Just for information, inside “Worked_Part” part, the powercopy has created:

- Data_For_Strip to get the Part's boundary
 - In Strip_Definition, there are:
 - OverAll_Boundary_Data to define the overall boundary.
 - Strip_Skeleton to have the boundaries of the unfolded and positioned part
 - Punches_Construction which has 2 hybrid bodies
 - Sketch_Punches where will be created the sketches for punches (direct user action)
 - Cutting_Shapes required for the creation of the stamping tools contours.
 - The “2Steps_CuttingLine” which enable the embedded customization of the strip.
 - The “Point_2Steps_CuttingLine” which also enable the embedded customization.

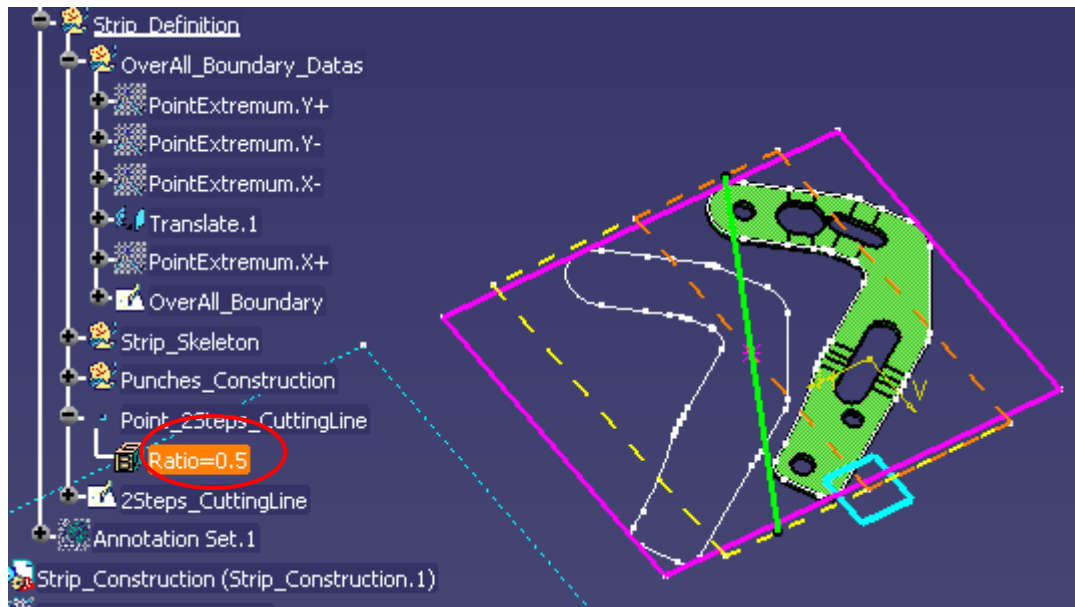


Modification of the cutting line

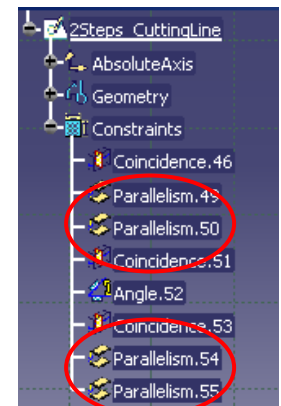
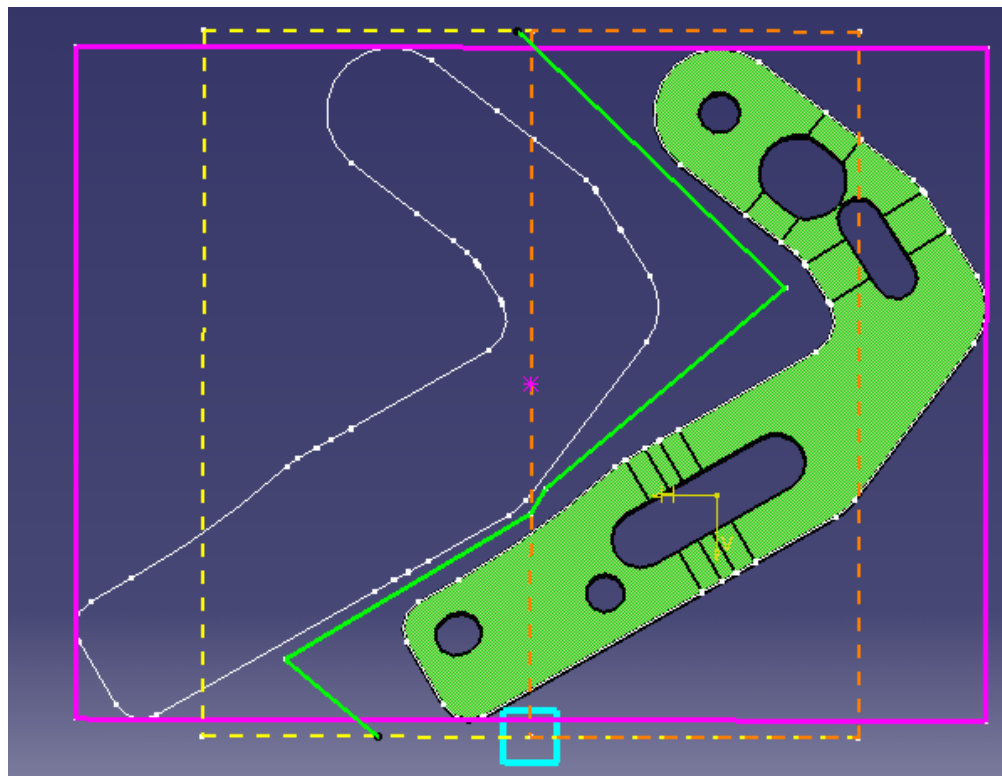
- As you can see on the previous page picture, the cutting line (in green) intersects the boundary of the two consecutive steps. In the chapter, we will see how to modify this cutting line for it to separate the two steps without intersecting the boundaries.

- Edit the "Point_2Steps_CuttingLine" ratio and set it to 0.69.

- The "Point_2Steps_CuttingLine" is between the two steps

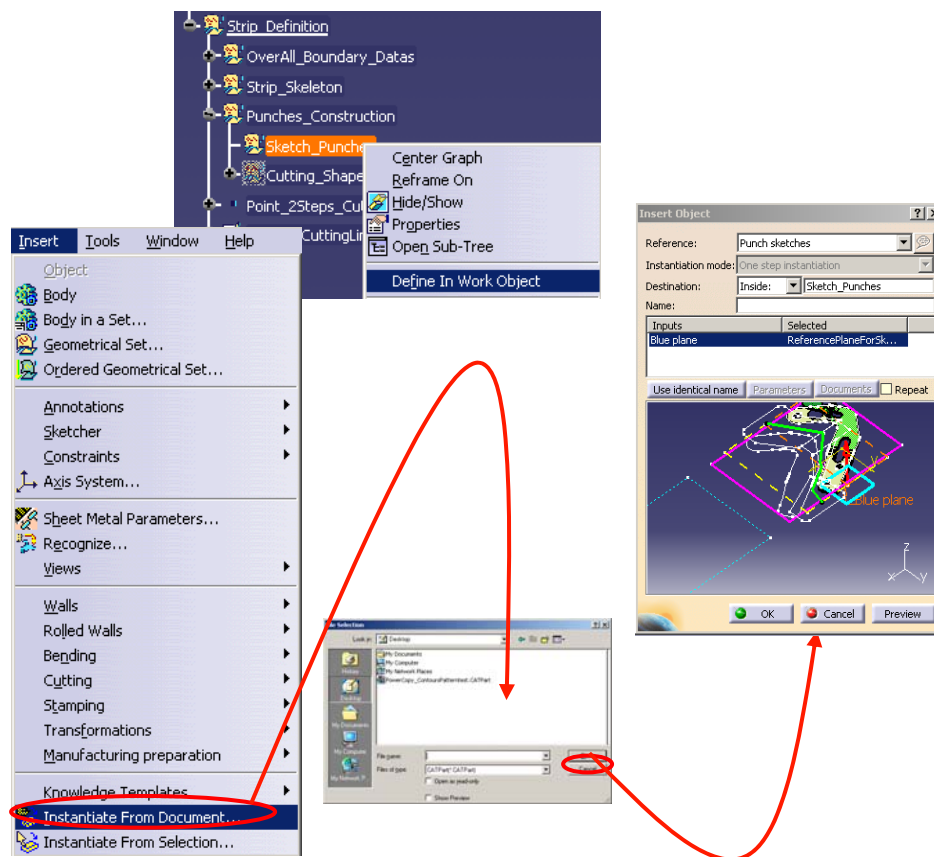


- ✎ Edit the 2Steps_CuttingLine sketch
- ✎ Delete the four Parallelism constraints.
- ✎ Modify the cutting line for it not to intersect the two steps boundaries.







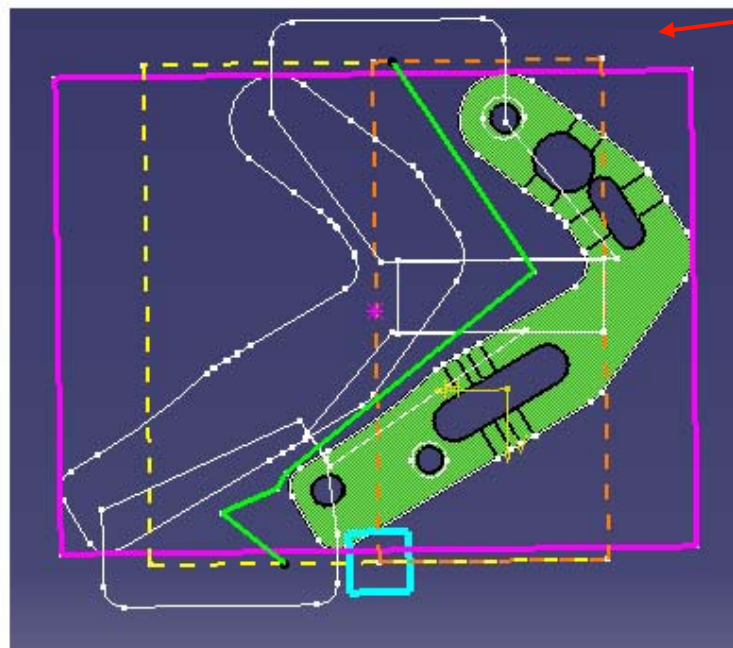
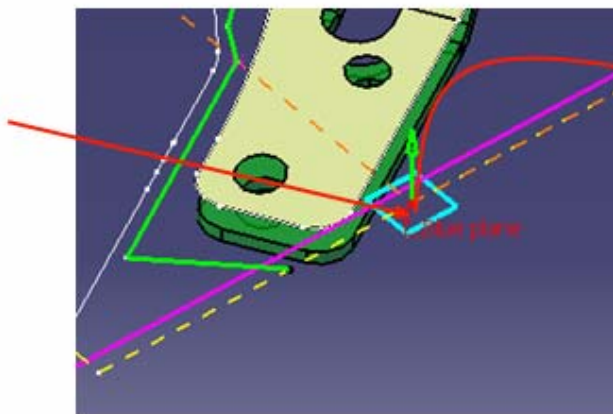
Creation of punch sketches (1 / 2)

- **Mandatory:** Define the “Sketch_Punches” open body as in work object to create the sketches in it:
 - 👤 Click on **Sketch Contour** icon.
 - 👤 The “Sketch_Punches” open body is underlined.
- To create all the sketches, it is possible to manually use the sketcher tools and the “ReferencePlaneForSketches” as support (blue plane). As the aim of this methodology is not to build sketches, the time of sketches creation will be saved by instantiating a powercopy containing the punch sketches.
 - 👤 Select **Insert / Instantiate from Document...**
 - 👤 The “File Selection” window is displayed.
 - 👤 Open the “...\\Part_for_strip_punches_contours.CATPart” file
 - 👤 The “Insert Object” window is displayed

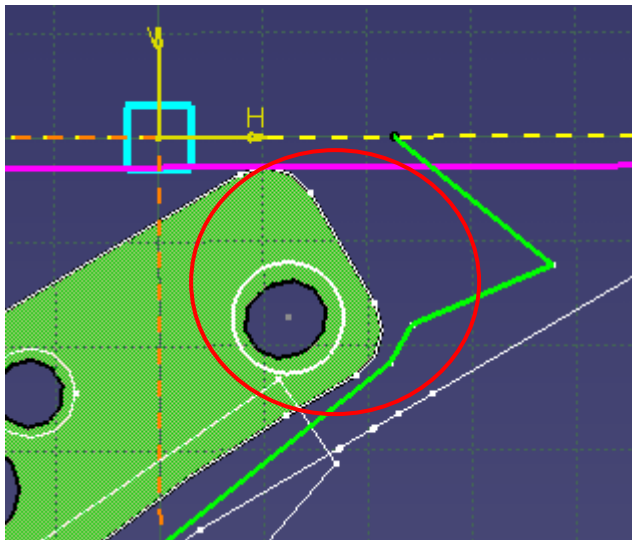


Creation of punch sketches (2 / 2)

- 
 Select the **ReferencePlaneForSketches** plane (the blue plane)
 - 
The plane has been correctly selected.
- 
 Select the **OK** button to validate
 - 
The sketches are displayed and the sketches are available in the tree



- 
 Click on **Sketch Contour icon**.
 - 
The "Sketch_Punches" open body is underlined.
 - 
Create a Sketch.



All the punch sketches should be created under “Sketch_Punches” geometric set. Sketch Contour function automatically defines in work object to “Sketch_Punches” and makes a sketch in editable mode.

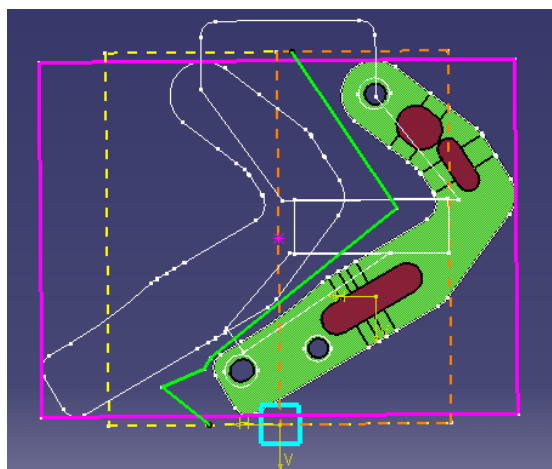


Show protected area

In the Strip Layout Design methodology, the stamps and other complex geometries that are not recognized by the “Recognize” function of the GSM workbench are isolated using a dedicated function. Due to this isolation, those regions get removed. At the step of creating punch contours, the user may get confused while differentiating between natural holes “protected area” in the part and the removed area in Isolation.






Hence a function is required to identify those regions for which punches are not to be created.

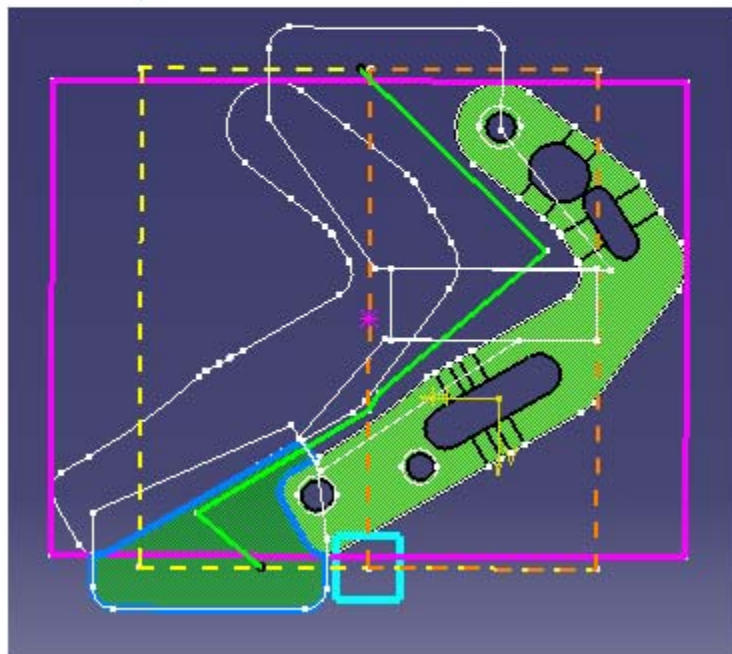
- Click on “Show Protected Area” function.
- Protected area will be shown red colour.
- Click on “Show Protected Area” function again protected area will be hidden.









Definition of the punch contours (1 / 4)

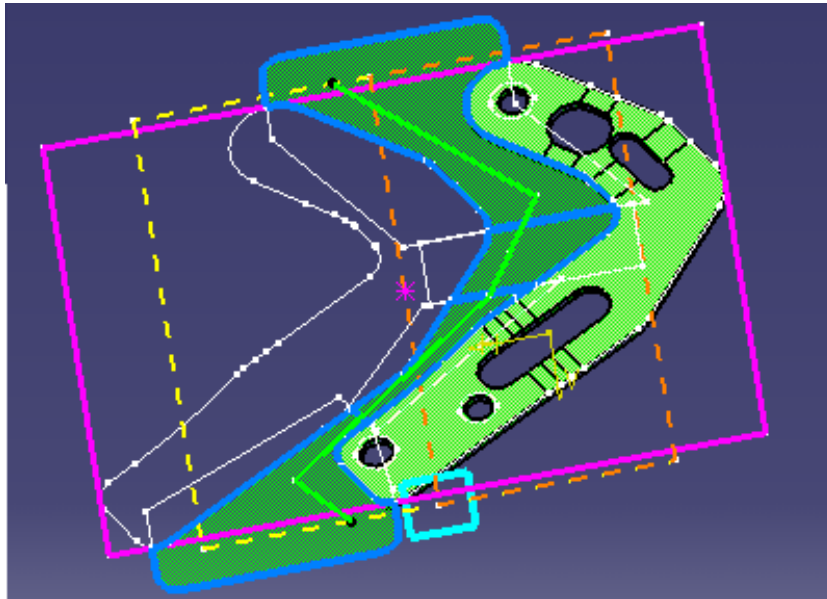
Create the external Punch Contours:

-  Select the **Punch contours** function.
-  Select the Punch type (External).
-  Select on the screen (or in the tree inside "Sketch_Punches" geometrical set) the **first sketch**.
 -  The "Sketch Punch" input has been selected and displayed in the "Punch Sketch".
-  Validate with the **OK** button.









Definition of the punch contours (2 / 4)

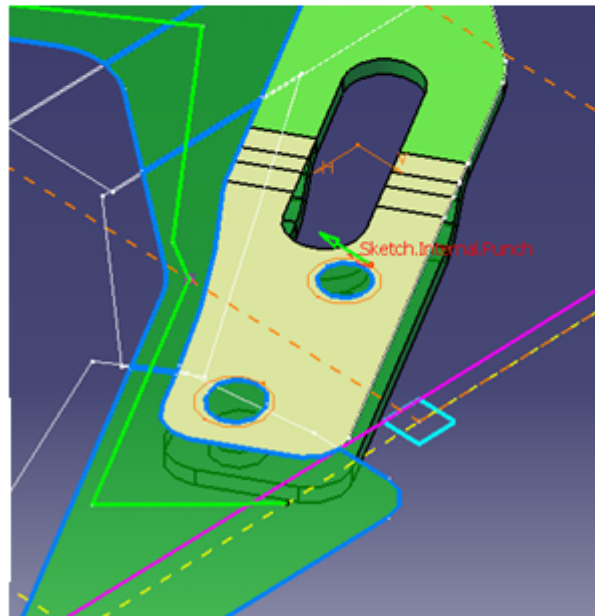
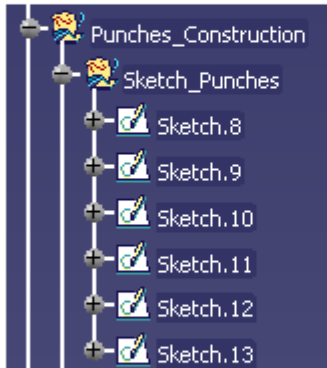
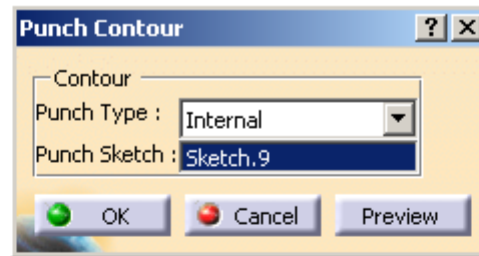
-  The first punch contour is displayed and an open body named "Punch" containing the features has been created.
-  Select (or pick in the tree), **one by one, all the other external sketches** of the "Sketch_Punches" geometrical set
-  After each selection, validate with the **OK** button
 -  *All the punch contours will be displayed and an open body named "Punch" containing the features for each sketch selected will be created*
-  When all the contours are created, Close the function with the **Cancel** button (Mandatory in order to avoid duplicate the contours)
 -  *Every external punch contours have been well designed*








Definition of the punch contours (3 / 4)

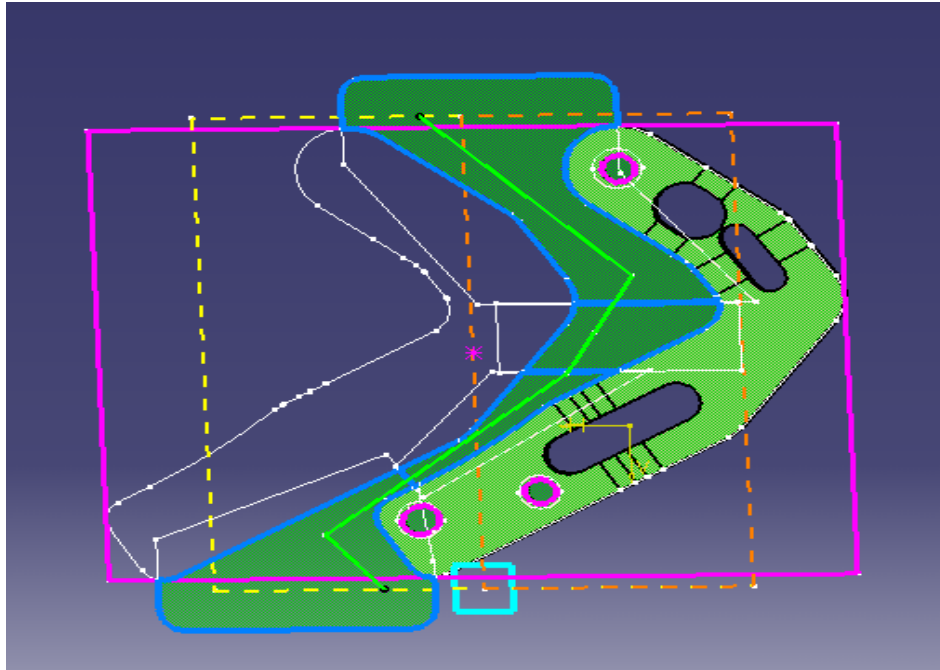


-  **Create the internal Punch Contours:**
 -  Select the **Punch contours** function.
 -  Select the Punch type (Internal).
 -  Select on the screen (or in the tree inside "Sketch_Punches" geometrical set).
 -  *The "Sketch Punch" input has been selected and displayed in the "Inputs" list.*
 -  Validate with the **OK** button



Definition of the punch contours (4 / 4)

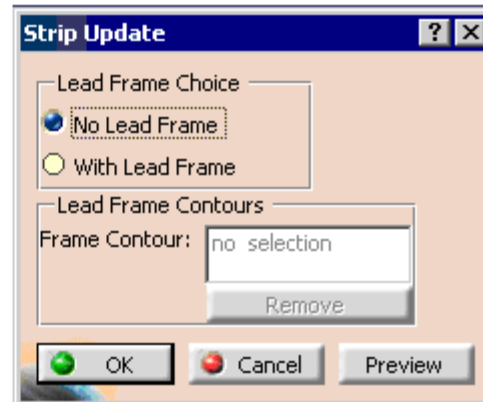
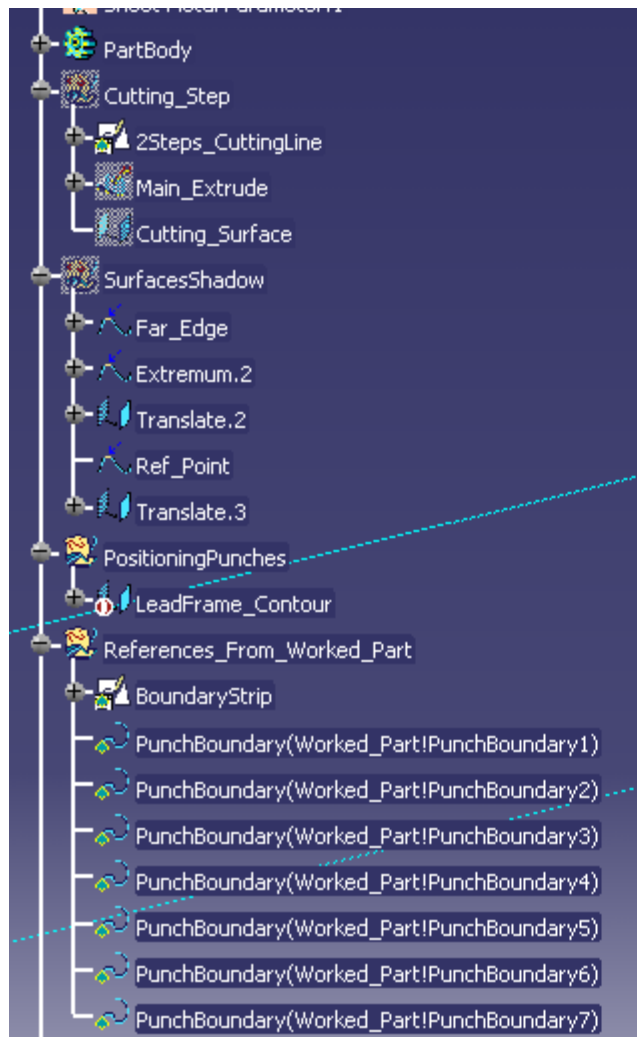
-  Select (or pick in the tree), **one by one, all the other internal sketches** of the "Sketch_Punches" geometrical set
-  After each selection, validate with the **OK** button
 -  *All the punch contours will be displayed and an open body named "Punch" containing the features for each sketch selected will be created*
-  When all the contours are created, Close the function with the **Cancel** button (Mandatory in order to avoid duplicate the contours)
 -  *Every external and internal punch contours have been well designed*



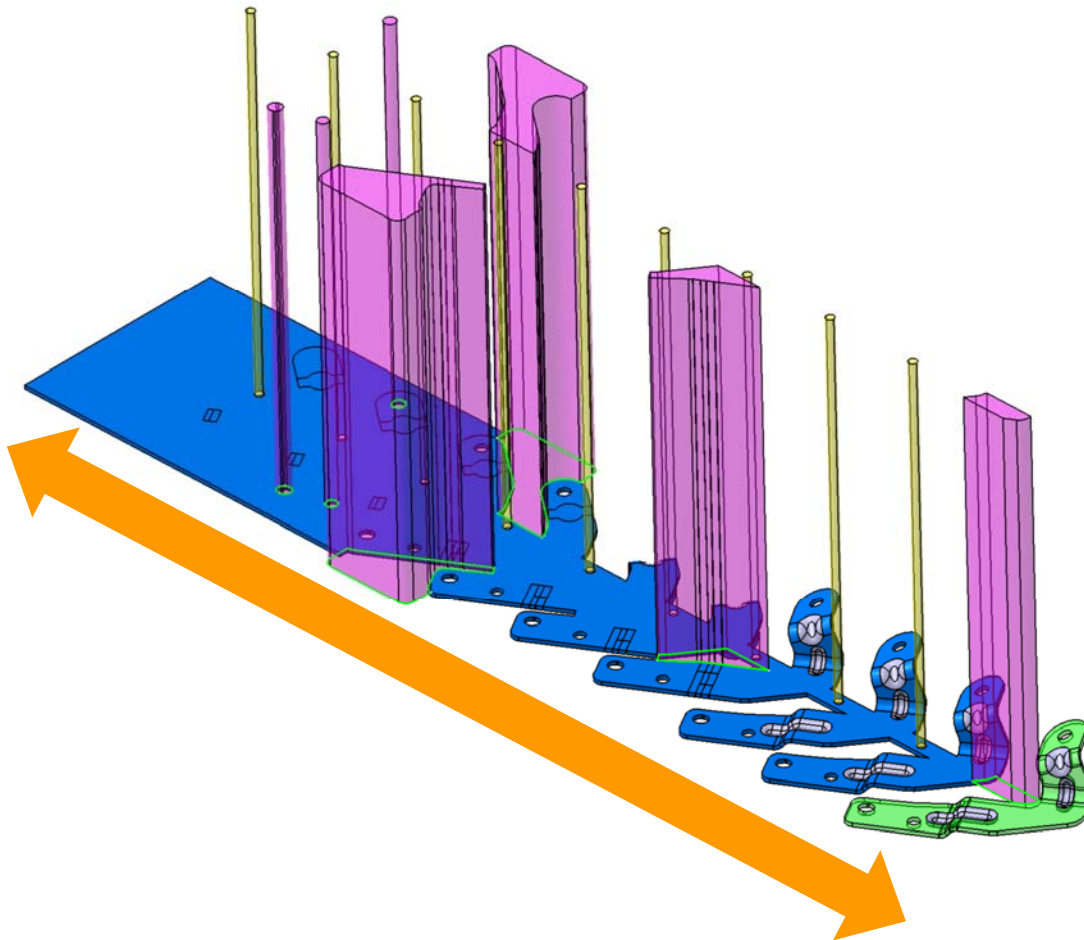
Strip Update



- Click on Strip Update.
- Select No Lead Frame.
- Click OK



Strip Layout Design



Introduction

- Now, as the strip layout is completely prepared, the phase of designing the strip layout can be started
- As it was already explained, the start point of the methodology is the finished part, and step by step the punching operations and the folding operations will be designed to reach the starting metal plate
- The scenario presented here is one possible scenario. The objective here is to present all the capabilities available thru the dedicated functions developed for the Strip Layout methodology.
Of course it is possible to define a different strip

Creation of the first step: A Punching Step (1 / 2)

- Edit the “Strip_Construction” part by double-clicking on it:

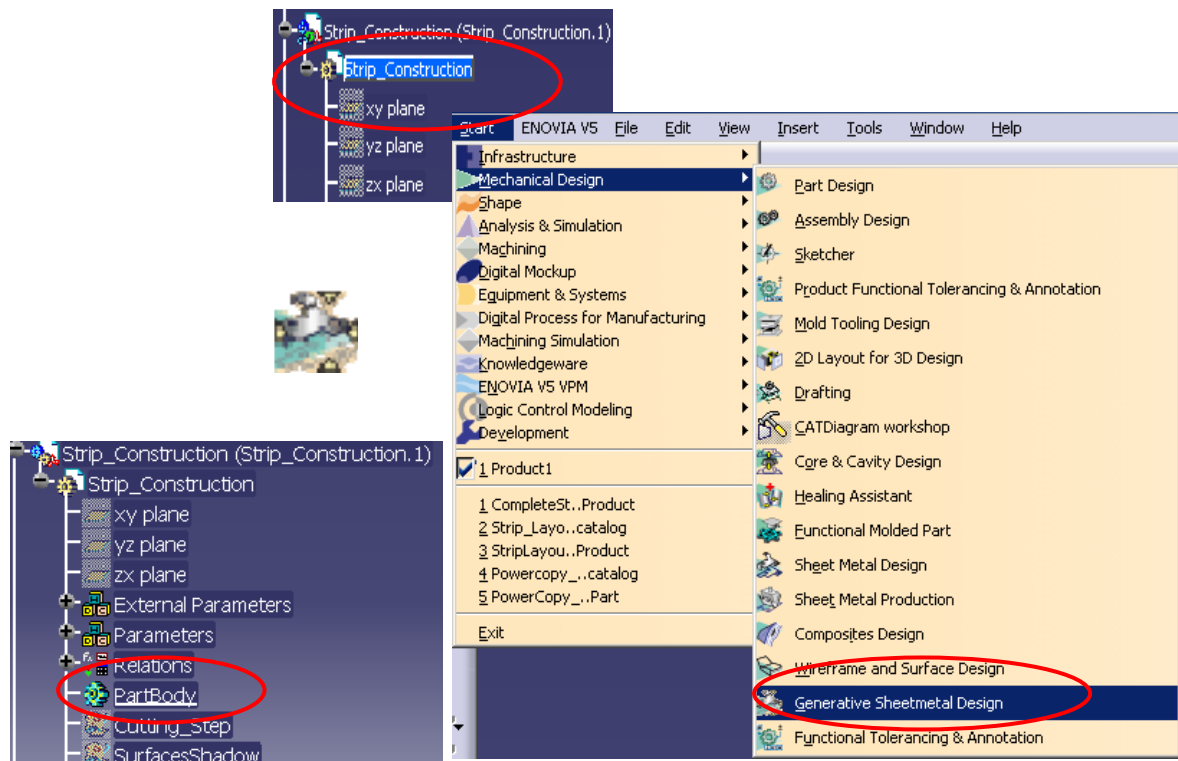
- 34 -

 The **Strip_Construction** part is highlighted in blue.

 Verify to be in “Generative Sheetmetal Design” workbench.

 If it is not the case, switch to “Generative Sheetmetal Design” workbench by selecting “Start \ Mechanical Design \ Generative Sheetmetal Design”.

Mandatory: Be sure to have PartBody name on the PartBody, this name is mandatory whatever your CATIA language is, if needed, rename it

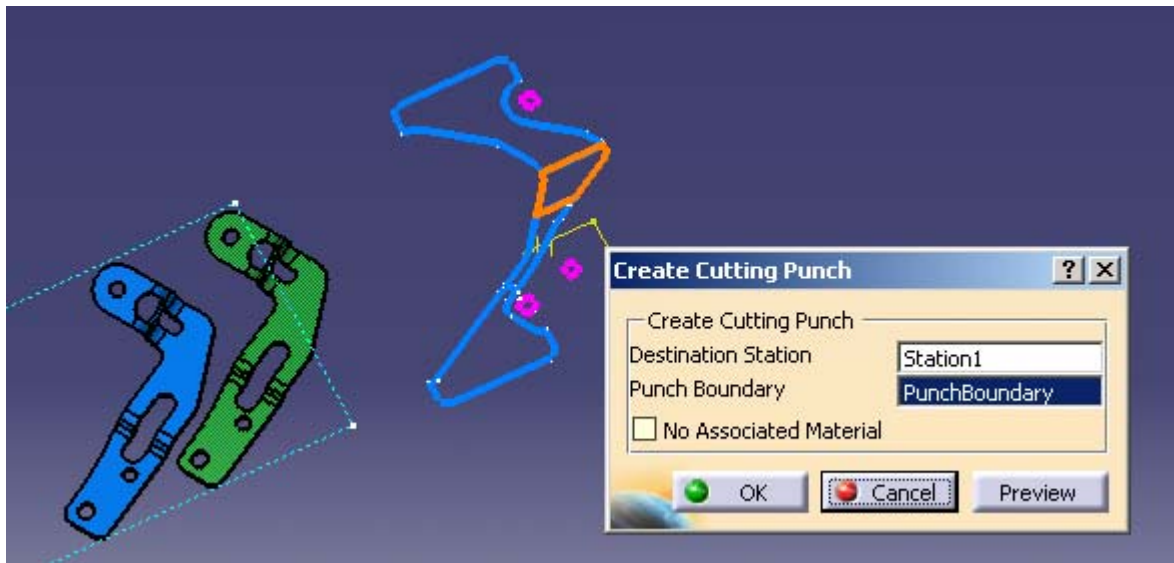


Creation of the first step: A Punching Step (2 / 2)

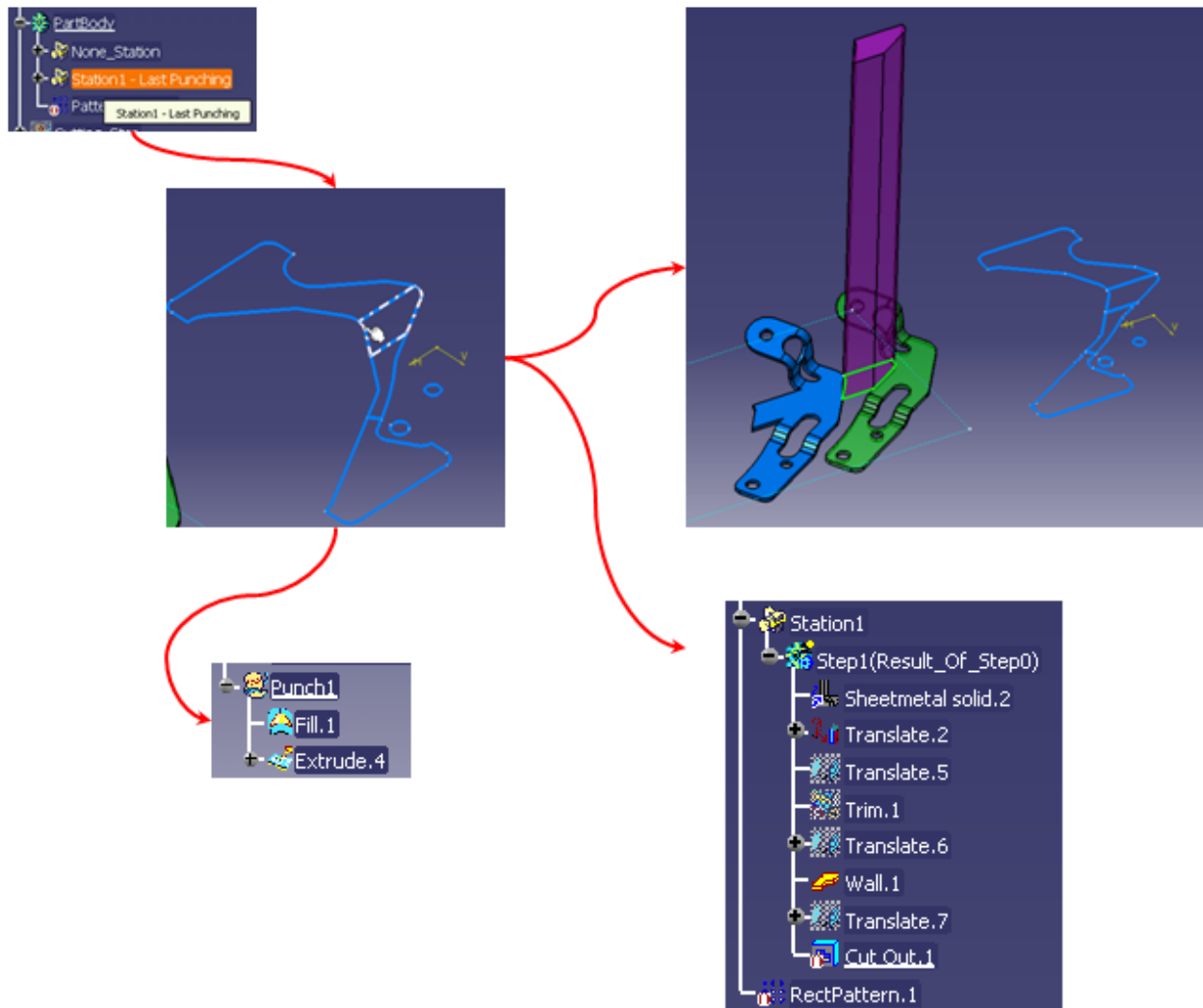
 Create the punch tool and the material removed by the tool:

 Use the **Punch trim operation** function by selecting the dedicated icon.

- The “Create Cutting Punch” window will be displayed in top left corner of the screen.

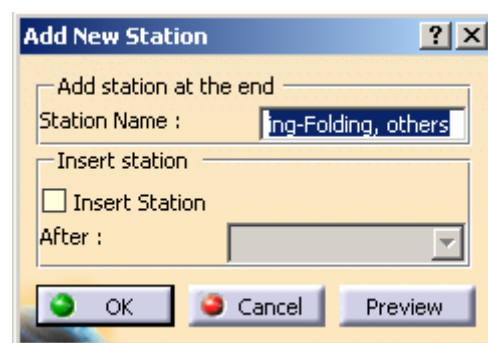


- Select the “Station1 – Last Punching” assembly feature.
- Select the right **blue punch contour** that will instantiate the punch to separate the finished part from the strip
 - The material removed by the tool has been added, the punch tool is displayed in purple and these new features have been created in “Punch1” and “PositionningPunches” open bodies and in the “Step1 (Result_Of_Step0)” body.*

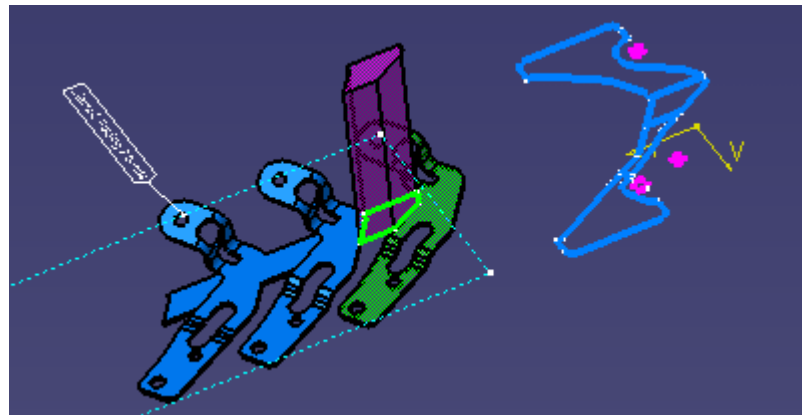


Creation of the second step: Unfolding two bends (1 / 2)

- **Add the step which will receive the unfolding operation:**
 👤 Select the **Add Step** function by selecting the dedicated icon




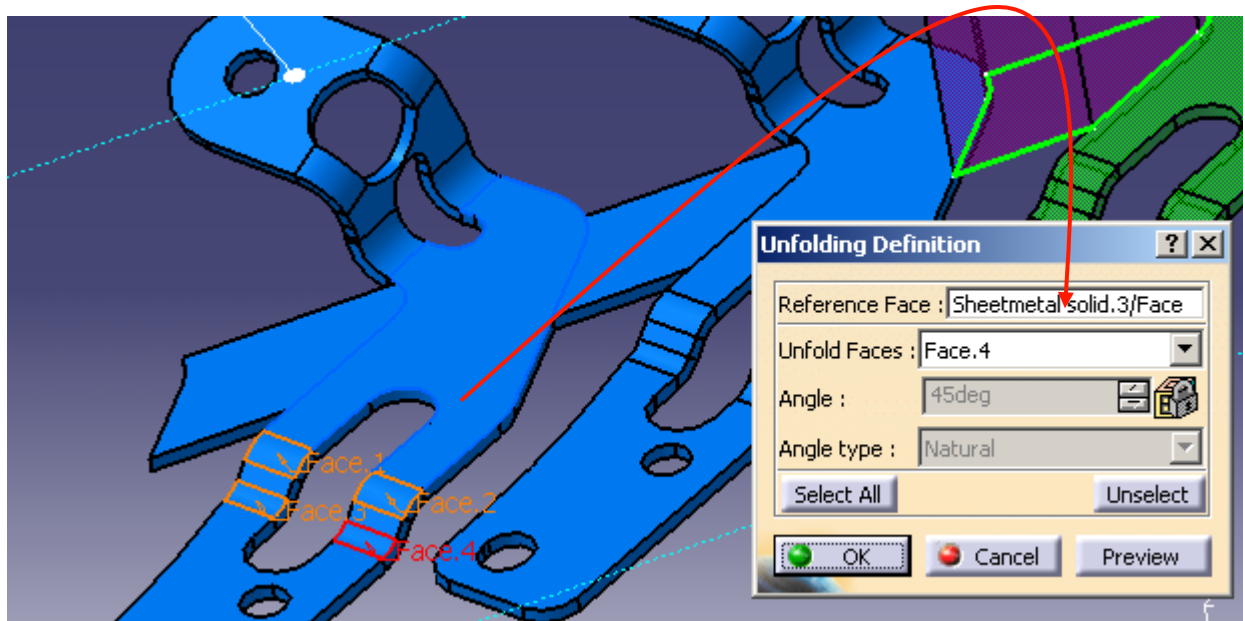
- Enter the name of the station (i.e. "Folding 2 bends").
- Validate with the **OK** button.
 - The "Station2: Folding 2 bends" is created in the "PartBody" body.
 - A parameter "Station2_FreeStep" has been created and set to 0.



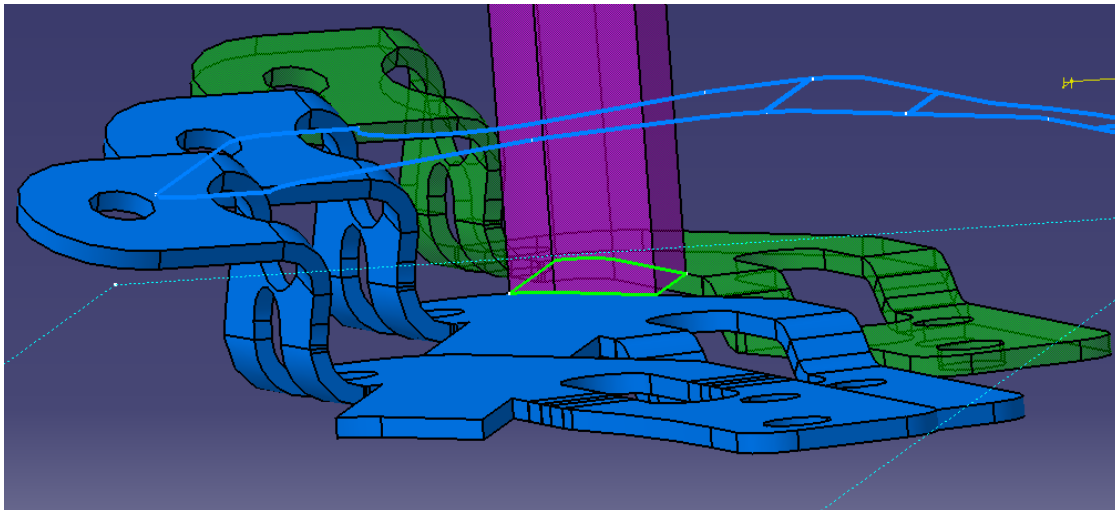
Creation of the second step: Unfolding two bends (2 / 2)

Unfold the two bends:

- Select the **Unfolding** icon ().
 - The "Unfolding Definition" window is displayed.
- Select the faces as shown:
 - The blue face as **Reference Face**, the red faces as **Unfold Faces** (either the internal or the external face of the bend can be chosen)



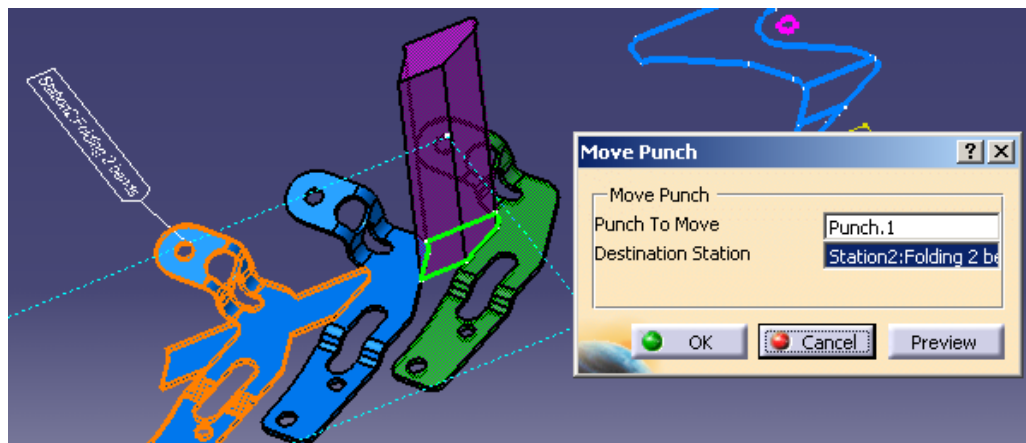
- 👤 Select the **OK** button to validate
- 👉 The bend is unfolded



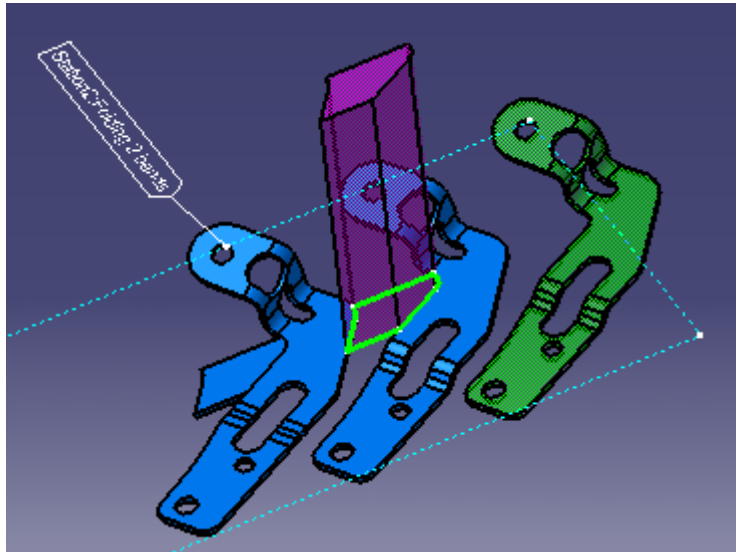
Move punch



- 👉 Click "Move Punch" function.
- 👉 Select "Punch.1".
- 👉 **Select** "Station2: Folding 2 bends".



- 👉 Click "Preview", Punch will be moved to station 2 from station 1.



Click "Cancel".

Creation of the third step: A punching step

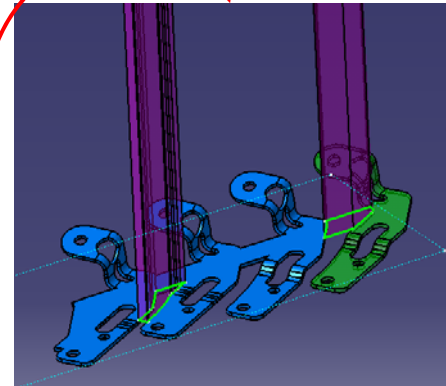
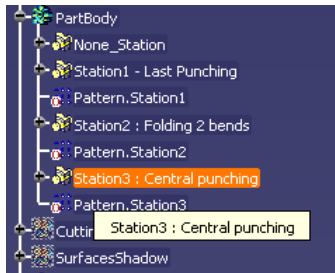
- **Add the step which will receive the punching operation:**

 - 👤 Select the **Add Step** function by selecting the dedicated icon and follow the same process as in the previous steps.
 - 👤 Enter the name of the station (i.e.: "Central punching")
 - 👤 Validate with the **OK** button.

 - 🔗 The "Station3: Central punching" is created in the "PartBody" body.
 - 🔗 A parameter "Station3_FreeStep" has been created and set to 0.
- **Create the punch tool and the material removed by the tool:**

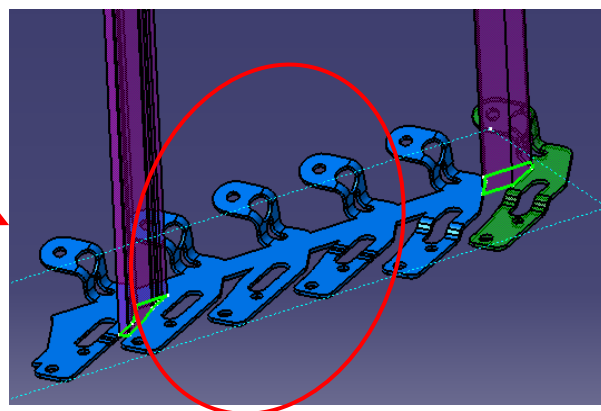
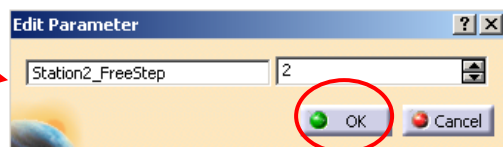
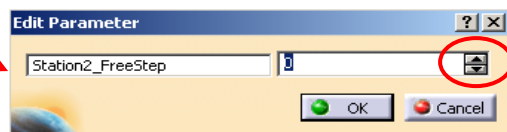
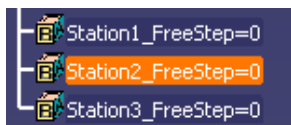
 - 👤 Select the **Punch Trim Operation** function by selecting the dedicated icon.
 - 👤 Select in the tree, the **Station3: Central punching** assembly feature which has just been created.
 - 👤 Select the **blue curve** as shown (or in the "References_From_Worked_Part" open body in the tree)

 - 🔗 The material has been added, the punch tool is displayed in purple and these new features have been created in the "Punch2" and "PositionningPunches" open bodies and in the "Step3 (Result_Of_Step2)" body.











Creation of two free steps

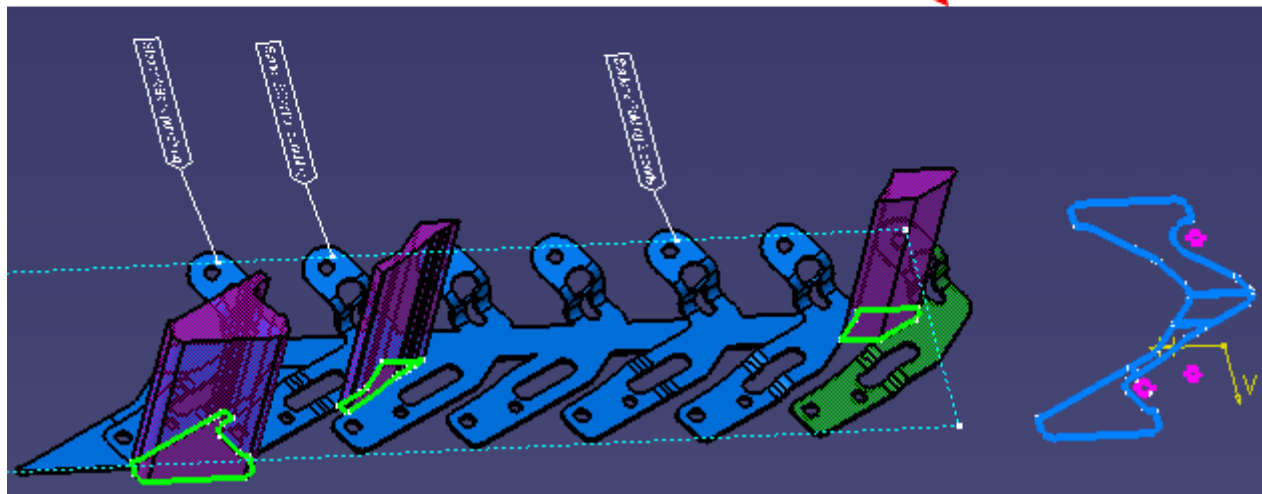
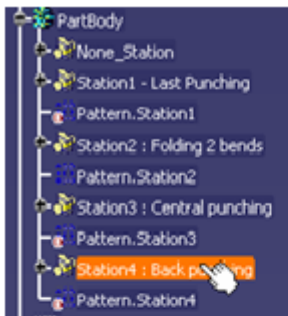
- To prevent the collision between folding and cutting punches, we will add two free steps between station 2 and station 3.
 - 👤 Select the **Station2_FreeStep** parameter of the "Strip_Construction" parameters
 - 👤 The "Edit Parameter" window is displayed.
 - 👤 Change the value from "0" to **2**.
 - 👤 Validate with the **OK** button.
 - 👤 The "Station2_FreeStep" parameter is now equal to 2.
 - 👤 The "Pattern.Station2" pattern is activated..
 - 👤 Two more steps have been added to the strip layout.
 - 👤 The Pattern of the station is automatically activated thanks to the Rule RuleManagingFreeSteps.
 - 👤 If we need, free steps (or station) will be managed this way. This value may be set for each station when needed. The strip will be updated and the associativity between the stations will be taken into account.




Creation of the fourth step: A punching step




- **Add the step which will receive the punching operation:**
 -  Select the **Add Step** function by selecting the dedicated icon and follow the same process as in the previous steps.
 -  Enter the name of the station (i.e.: "Back punching")
 -  Validate with the **OK** button.
 -  The "Station4 : Back punching" is created in the "PartBody" body.
 -  A parameter "Station4_FreeStep" has been created and set to 0.
- **Create the punch tool and the material removed by the tool:**
 -  Select the **Punch Trim Operation** function by selecting the dedicated icon.
 -  Select in the tree, the **Station4 : Back punching** assembly feature which has just been created.
 -  Select the **blue curve** as shown (or in the "References_From_Worked_Part" open body in the tree)

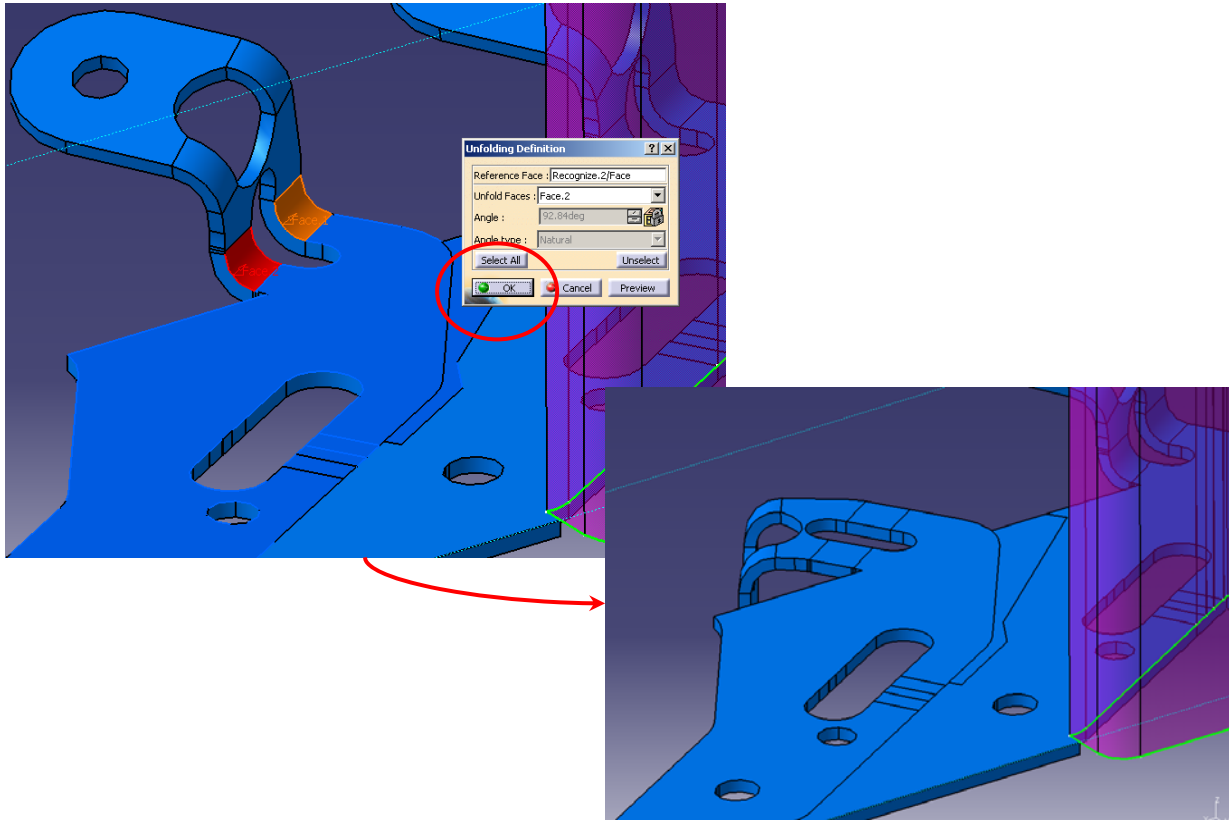
- The material has been added, the punch tool is displayed in purple and these new features have been created in the “Punch3” and “PositionningPunches” open bodies and in the “Step4 (Result_Of_Step3)” body.

















Creation of the fifth step: A folding step

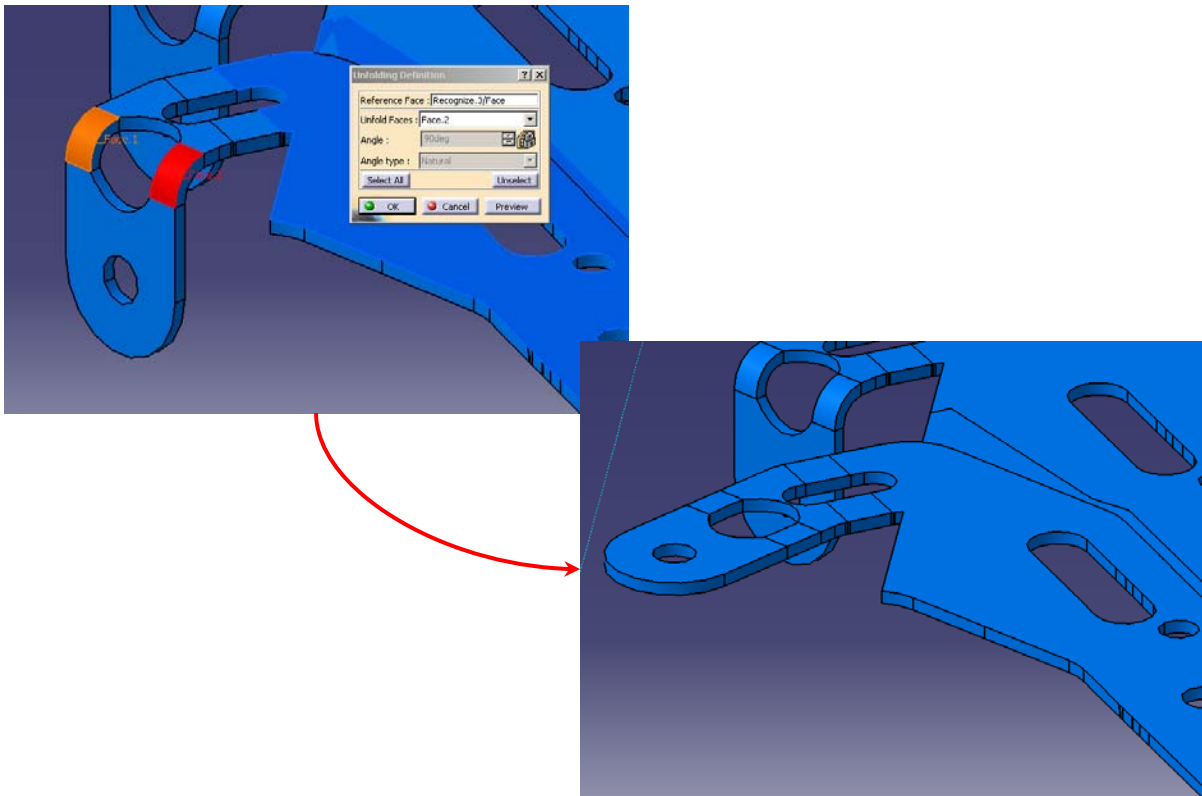
- Add the step which will receive the folding operation:**
 - Select the **Add Step** function by selecting the dedicated icon and follow the same process as in the previous steps.
 - Enter the name of the station (i.e.: "Front folding 2")
 - Validate with the **OK** button.
 - The “Station5: Front folding 2” is created in the “PartBody” body.
 - A parameter “Station5_FreeStep” has been created and set to 0.
- Unfold the second front bend:**
 - Select the **Unfolding** icon ()
 - The “Unfolding Definition” window is displayed.
 - Select the faces as shown:

-  The blue face as **Reference Face** (it should be the same as the selected one for the "Recognize" operation), the red faces as **Unfold Faces** (either the internal or the external face of the bend can be chosen)
-  Select the **OK** button to validate
 -  The bend is unfolded












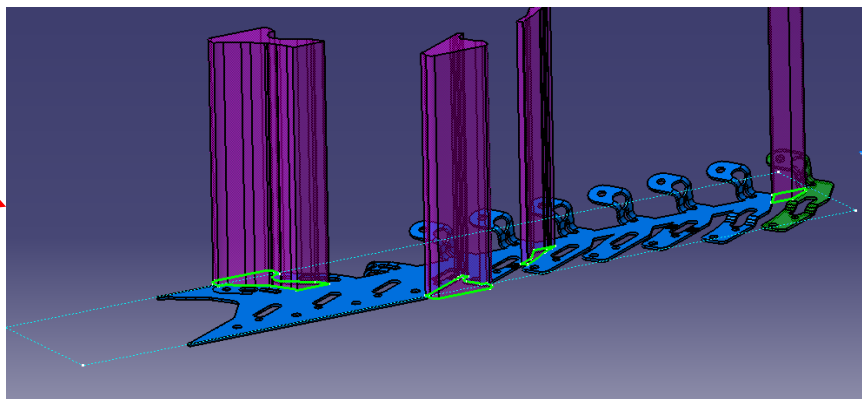
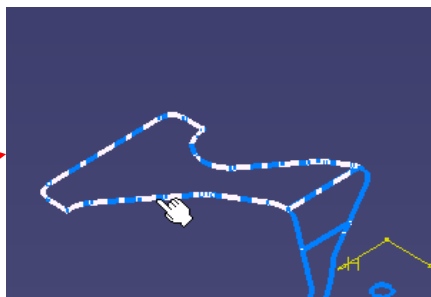
Creation of the sixth step: A Folding step

-  **Add the step which will receive the folding operation:**
 -  Select the **Add Step** function by selecting the dedicated icon and follow the same process as in the previous steps.
 -  Enter the name of the station (i.e.: "Front folding 1")
 -  Validate with the **OK** button.
 -  The "Station6: Front folding 1" is created in the "PartBody" body.
 -  A parameter "Station6_FreeStep" has been created and set to 0.
-  **Unfold the first front bend:**
 -  Select the **Unfolding** icon ()
 -  The "Unfolding Definition" window is displayed.
 -  Select the faces as shown:
 -  The blue face as **Reference Face** (it should be the same as the selected one for the "Recognize" operation), the red faces as **Unfold Faces** (either the internal or the external face of the bend can be chosen)
 -  Select the **OK** button to validate
 -  The bend is unfolded










Creation of the seventh step: A punching step

- **Add the step which will receive the punching operation:**
 -  Select the **Add Step** function by selecting the dedicated icon and follow the same process as in the previous steps.
 -  Enter the name of the station (i.e.: "Front punching")
 -  Validate with the **OK** button.
 -  The "Station7: Front punching" is created in the "PartBody" body.
 -  A parameter "Station7_FreeStep" has been created and set to 0.
- **Create the punch tool and the material removed by the tool:**
 -  Select the **Punch Trim Operation** function by selecting the dedicated icon.
 -  Select in the tree, the **Station7: Front punching** assembly feature which has just been created.
 -  Select the **blue curve** as shown (or in the "References_From_Worked_Part" open body in the tree)
 -  The material has been added, the punch tool is displayed in purple and these new features have been created in the "Punch4" and "PositionningPunches" open bodies and in the "Step7 (Result_Of_Step6)" body.




Creation of the eighth step: An internal punching step

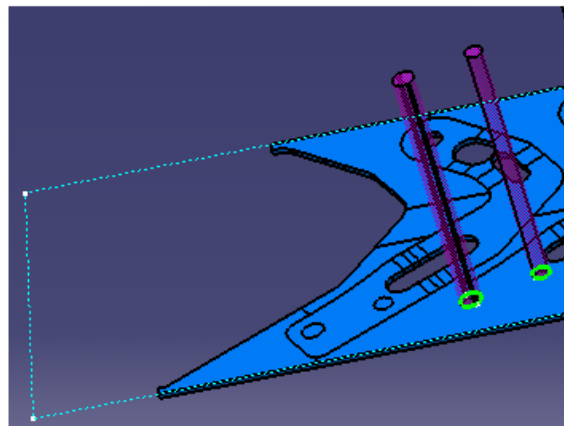
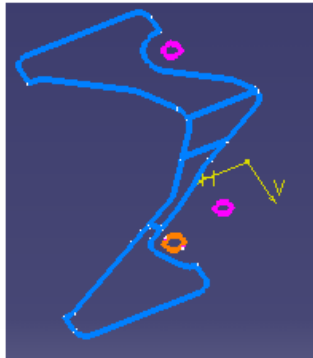
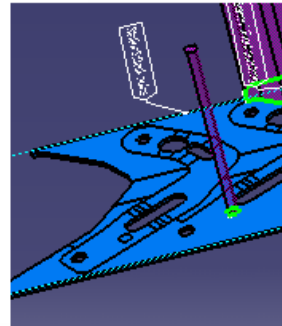
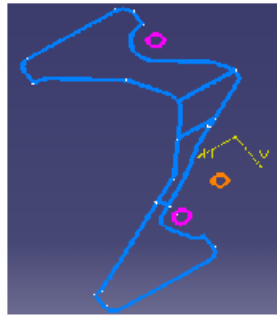
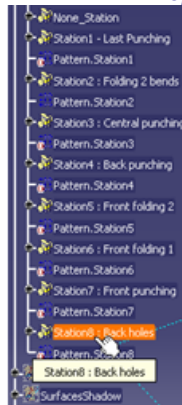
- **Add the step which will receive the punching operation:**
 -  Select the **Add Step** function by selecting the dedicated icon and follow the same process as in the previous steps.
 -  Enter the name of the station (i.e.: "Front hole")
 -  Validate with the **OK** button.
 -  The "Station8: Back holes" is created in the "PartBody" body.
 -  A parameter "Station8_FreeStep" has been created and set to 0.
- **Create the punch tool and the material removed by the tool:**
 -  Select the **Punch Trim Operation** function by selecting the dedicated icon.
 -  Select in the tree, the **Station8: Back holes** assembly feature which has just been created.

 Select the **blue curve** as shown (or in the “References_From_Worked_Part” open body in the tree).

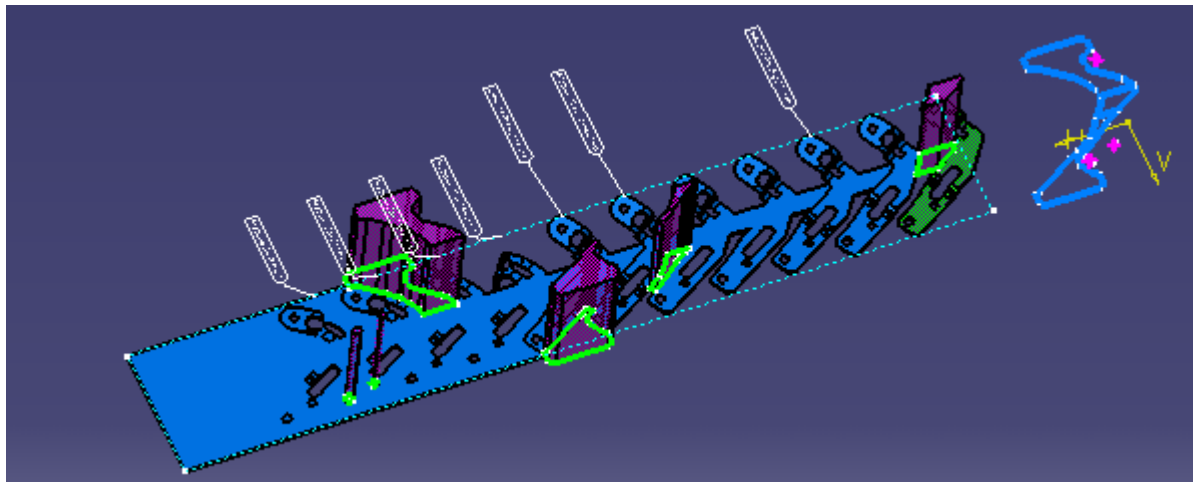
 Select the punch contour.

 The material has been added, the punch tool is displayed in purple and these new features have been created in the “Punch5” and “PositionningPunches” open bodies and in the “Step8 (Result_Of_Step7)” body.

 Repeat the action for other contour at station 8.






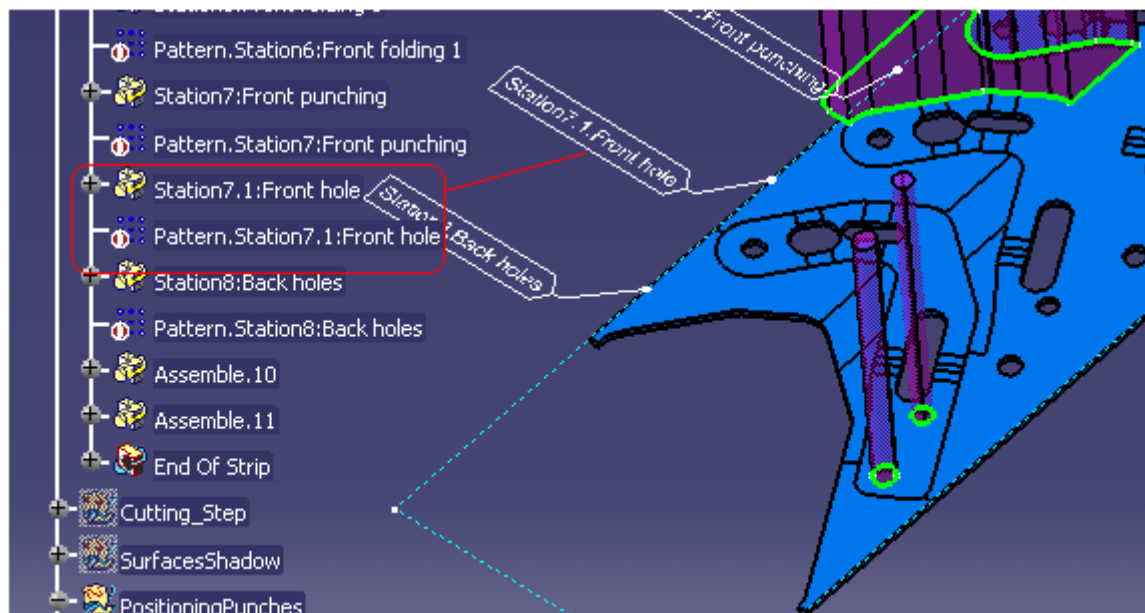
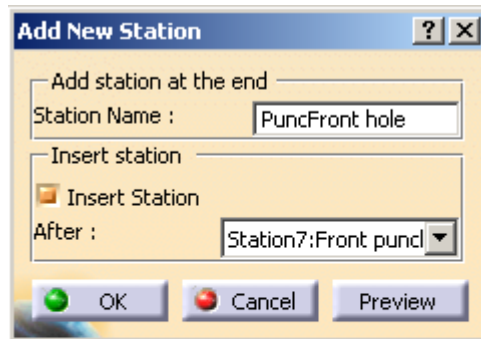
 Click Trim Strip function. The strip will be completed.



Creation of the ninth step: the methodology finishing step (1 / 2)

Add the step which will receive the starting operation:

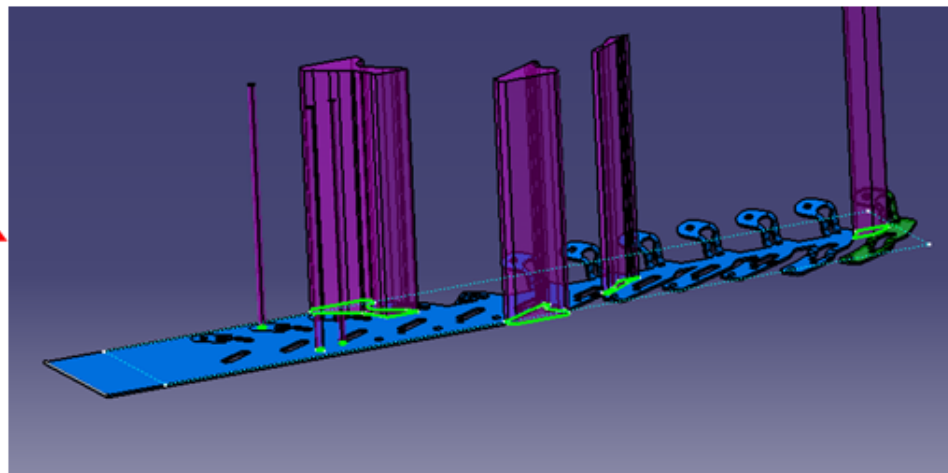
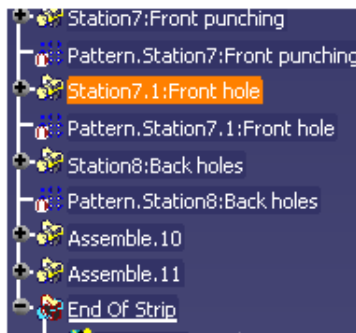
-  Select the **AddStep** function by selecting the dedicated icon. Select Insert Station.
-  Enter the name of the station (i.e: "Front hole").
-  Select station 7 as after which the station has to be insert. Validate with the **OK** button.



Creation of the ninth step: the methodology finishing step (2 / 2)

➤ **Create the punch tool and the material removed by the tool:**

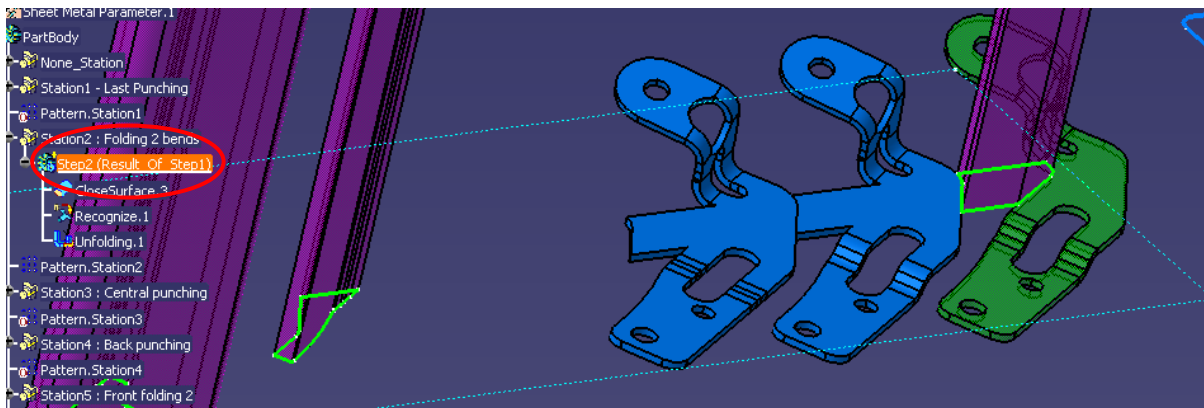
- 👤 Select the **Punch Trim Operation** function by selecting the dedicated icon.
- 👤 Select in the tree, the **Station9 : Front hole** assembly feature which has just been created.
- 👤 Select the **blue curve** as shown (or in the "References_From_Worked_Part" open body in the tree). As the punch for those two holes has been designed in the same sketch, the two elements of the punch are instantiated in the same operation.
- 👤 *The material has been added, the punch tool is displayed in purple and these new features have been created in the "Punch6" and "PositionningPunches" open bodies and in the "Step9 (Result_Of_Step8)" body.*





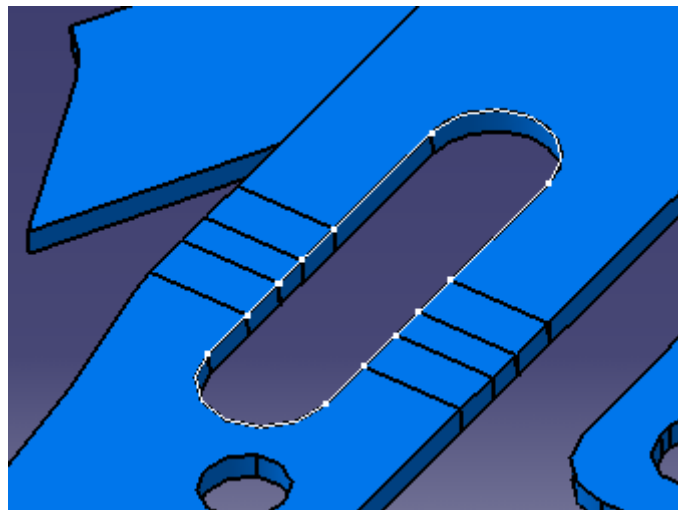
Removed geometry holes filling

- As you can see in your strip, there are lots of holes to fill. The following operation can be done whenever you want during the strip construction. We have decided to group them at the end only for clarity.
- We will illustrate the two main possibilities to fill the holes, but if you are familiar with another, you can use it as you want.
- The associative construction works also for the holes, so you will just have to fill the holes once.
- **Filling a hole using the wall function.**
 - 👤 Define the "Step2" body as in work object.

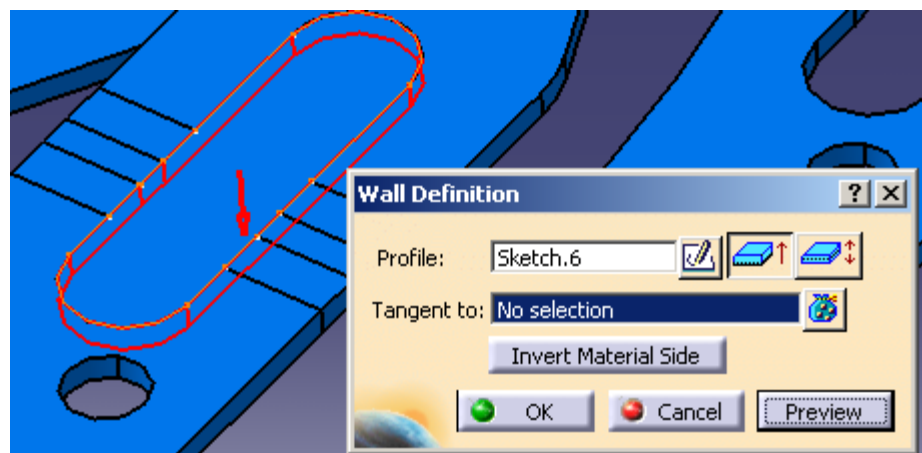
 The 3 first steps of the strip are displayed.

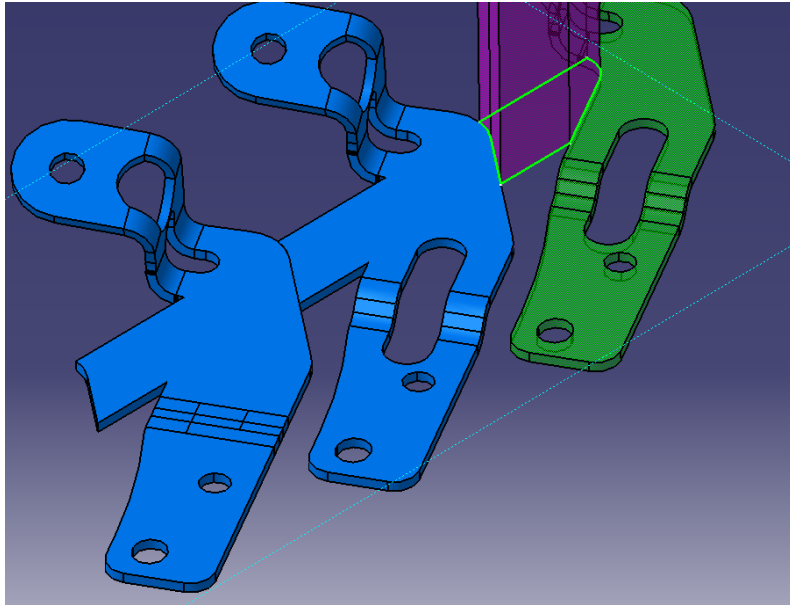


-  Launch the “wall” function.
-  Create the sketch for profile.

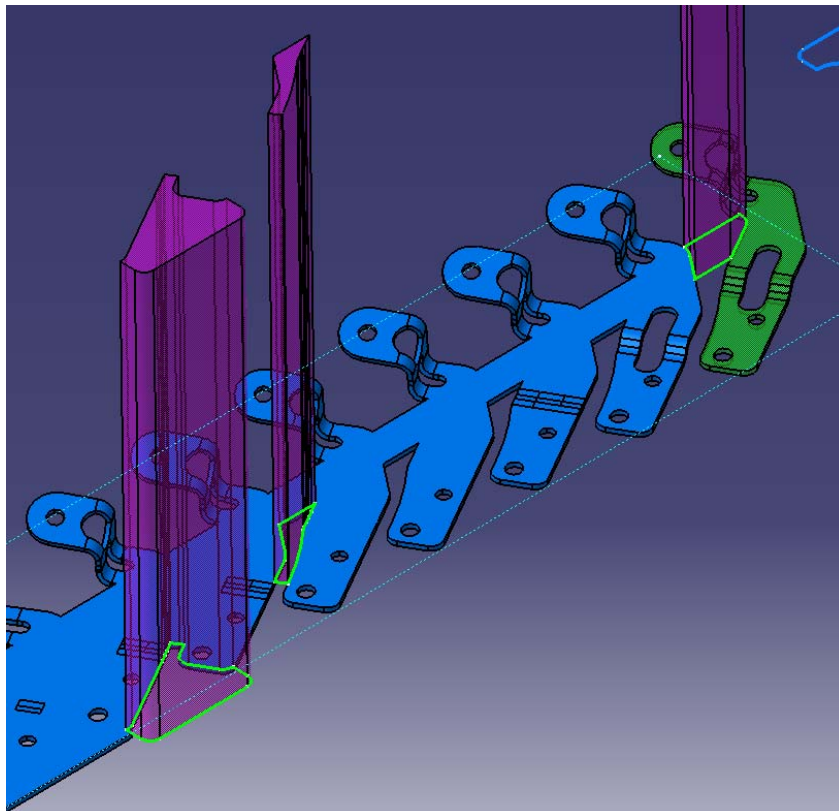


-  Click OK to validate.
-  The hole is filled.



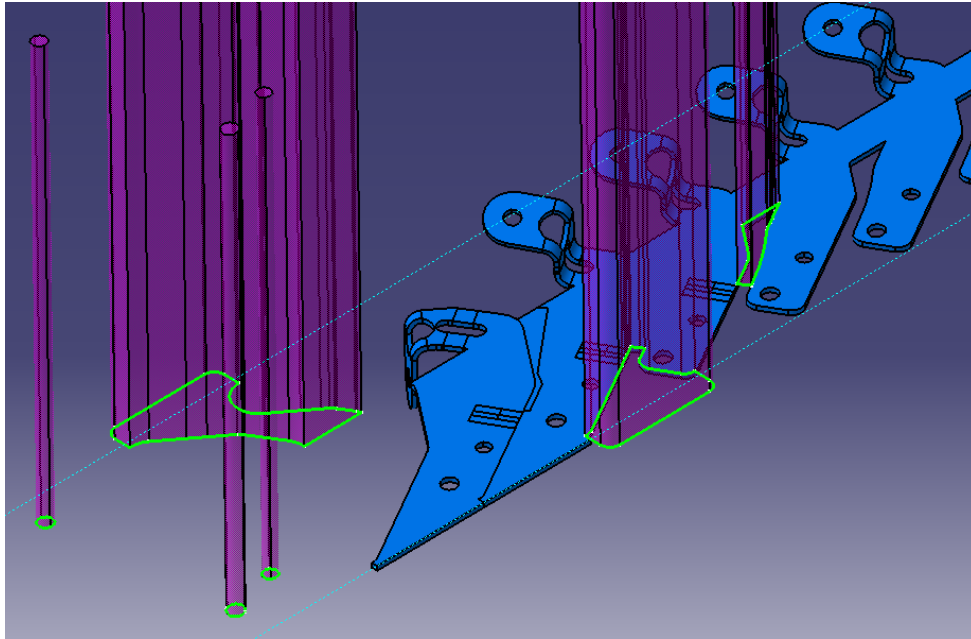



- Define the PartBody as in work object
- The fill is propagated to all the following steps.

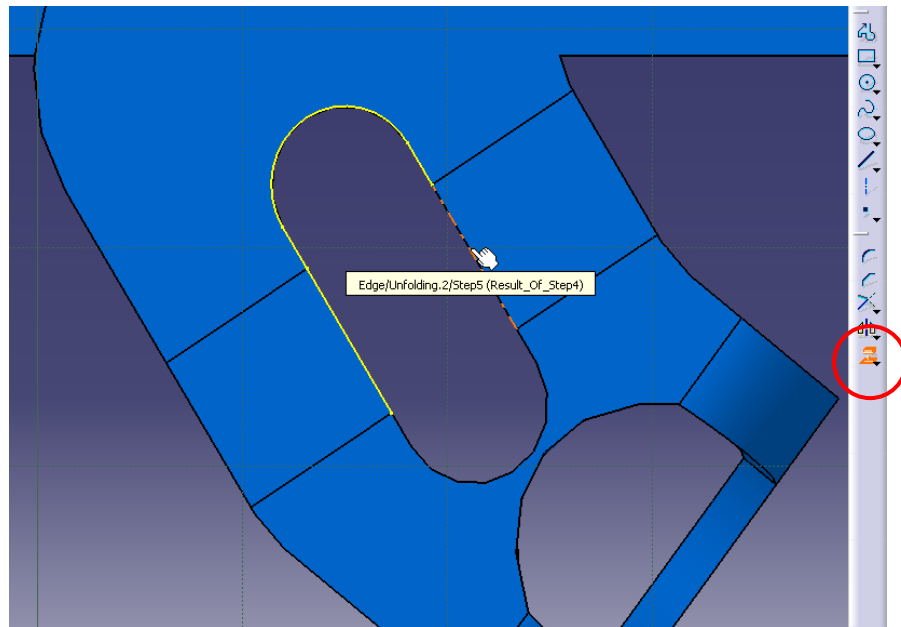


Filling a hole using wall

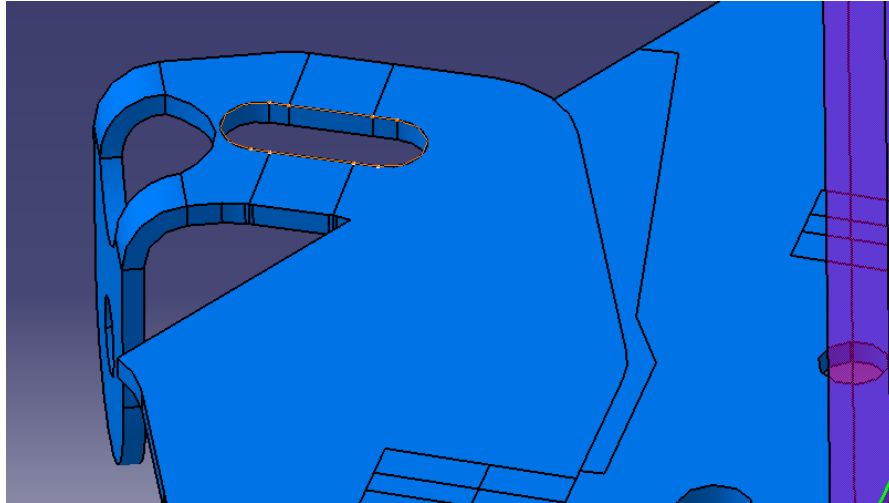
- Define the "Step5" body as in work object.
- The following steps are hidden.



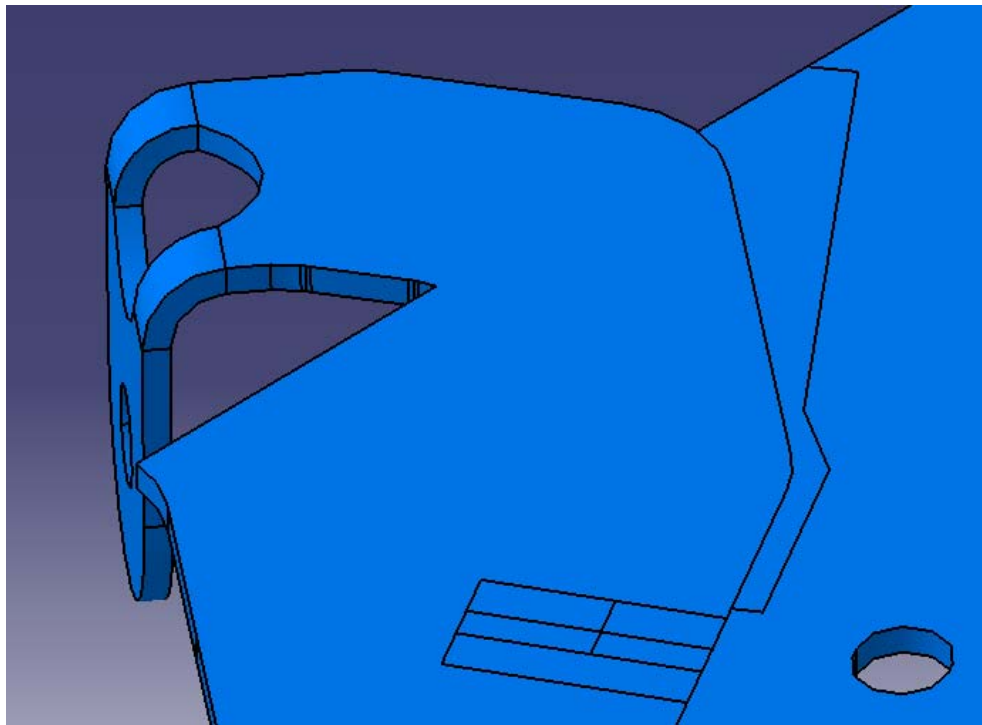
- 👤 Create a sketch on the upper face of the strip and double-click on the “Project 3D Elements” icon ().
- 👤 Pick, one by one, all the edges of the contour of the hole you want to fill.
 - 👉 The edges are projected in yellow on the sketch.



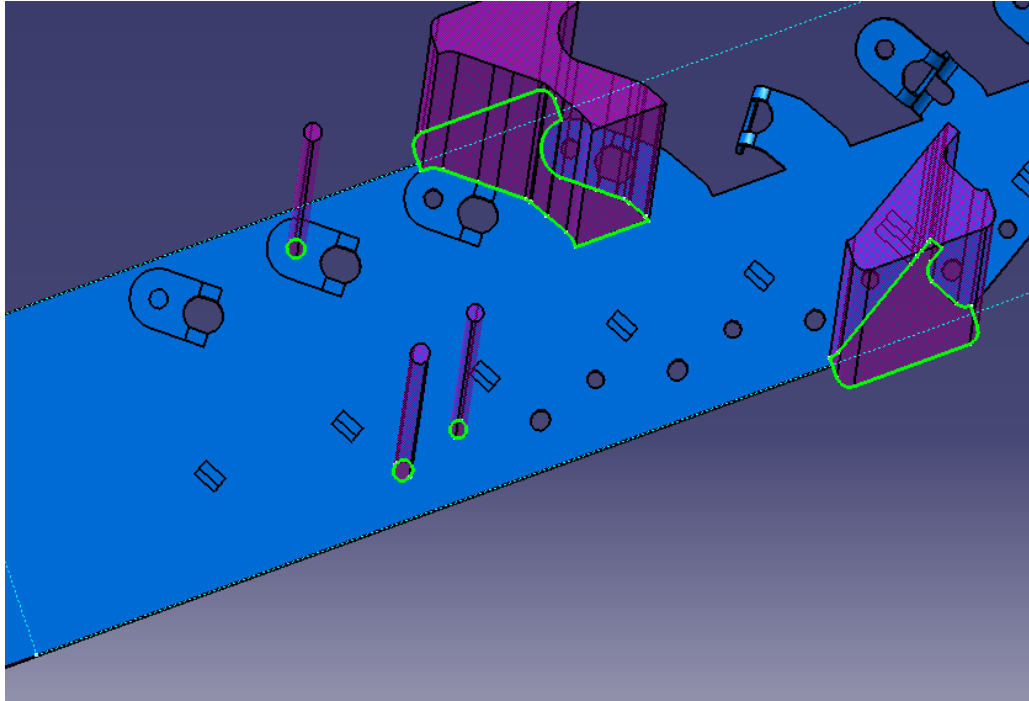
- 👤 Leave the sketcher.
 - 👉 The sketch is visible with the hole's contour.



- Launch the wall Definition.
- Pick the previously created sketch, check the extrusion direction.
- Click OK to create the wall.
- The wall is created

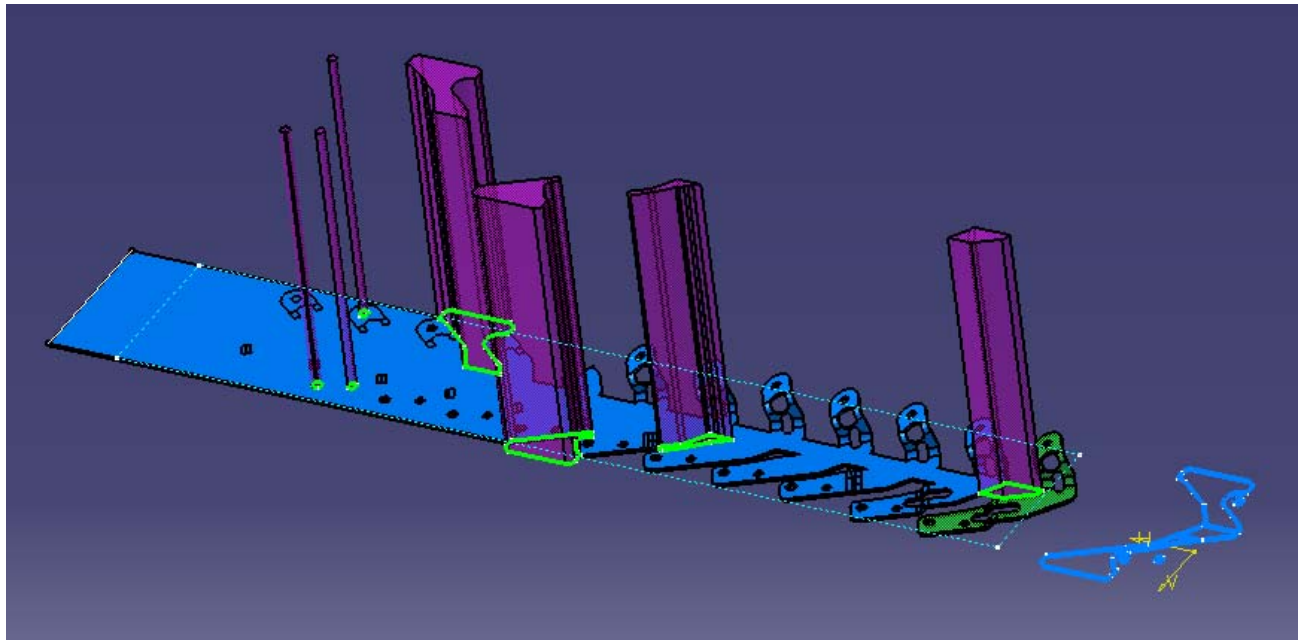


- Define the PartBody as in work object
- The wall is repeated to all of the following stations.



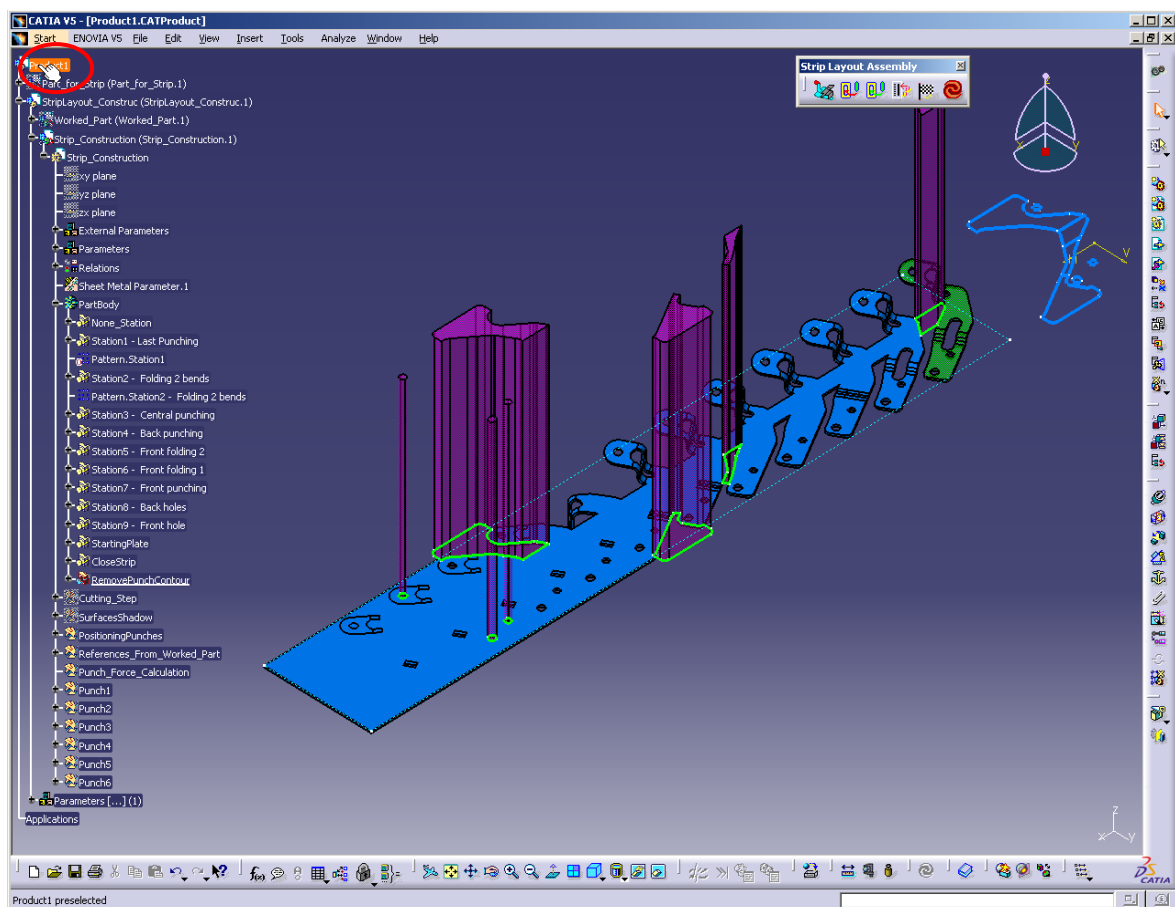
 Repeat this method of projection / padding the following hole on Step6.

 Your strip layout is now filled. Only the removed geometries are missing. We will give you all the method after the description of the “Independent steps” construction method.

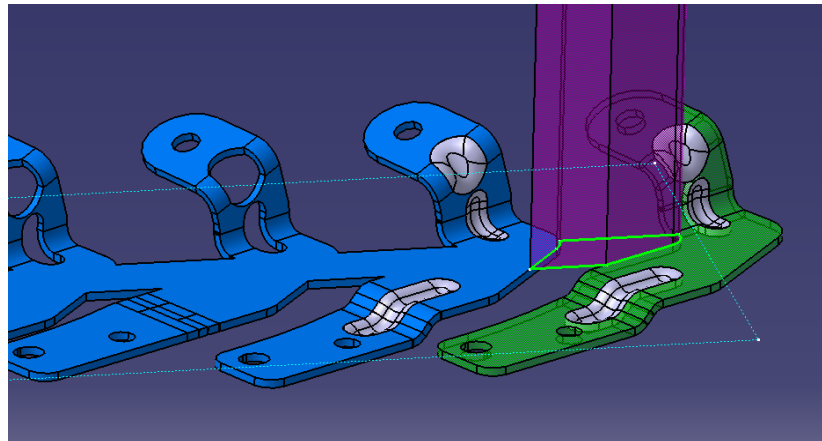
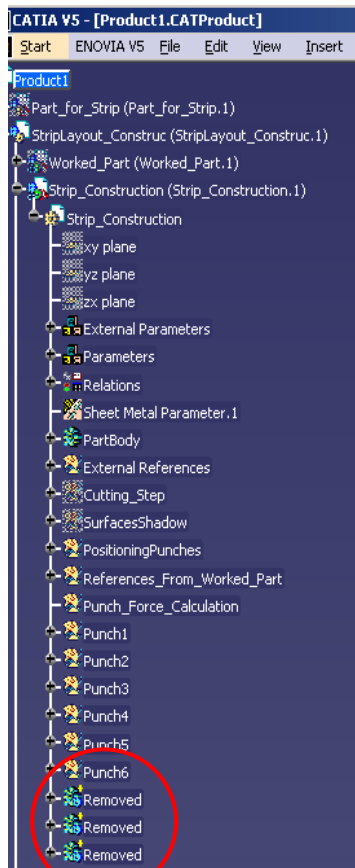


Stamps, ribs and complex geometries recovery

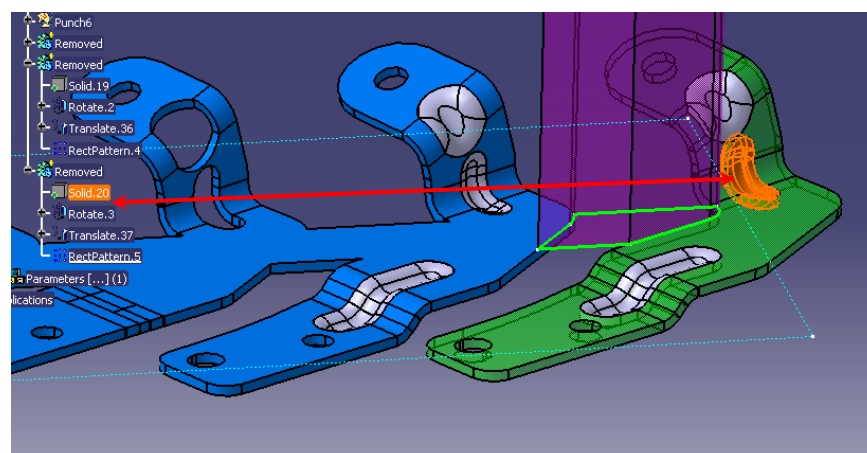
- The following explanation and screenshots have been done on the “Independent Steps” version of the strip, but are also valid for the “Associative Steps” mode.
- All the isolated geometries have to be recovered for the trip to be complete. We will use two functions for that.
 - 👤 The recovery to the two first steps will be done automatically with the “Automatic_Recovery” function (🔧).
 - 👤 The recovery to the other steps will be done semi-automatically with the “Manual_Recovery” function (🔧).
- Recovering geometries to the two first steps.
 - 👤 Activate the root product.



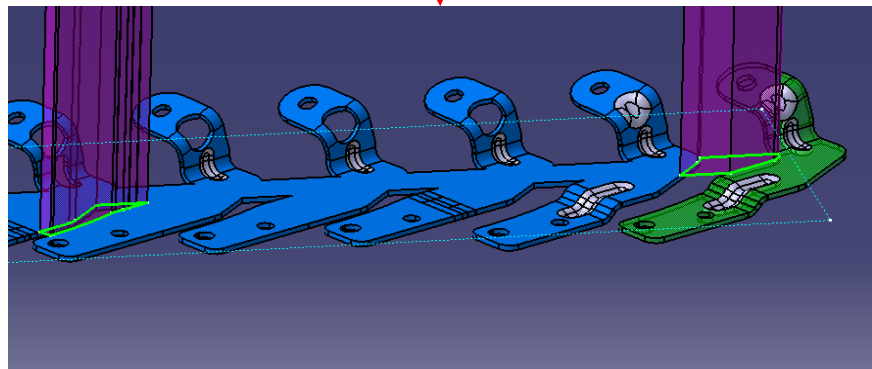
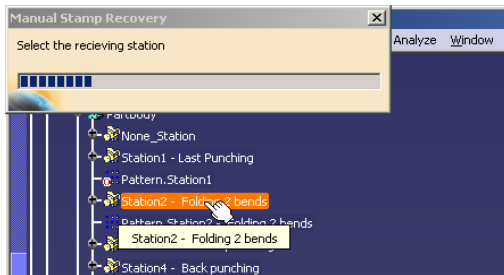
- 👤 Launch the “Automatic Recovery” function.
 - 👤 The isolated geometries are copied to the None_Station and the Station1 – Last punching.
 - 👤 Three “Removed” bodies are added to the “Strip_Construction” part.



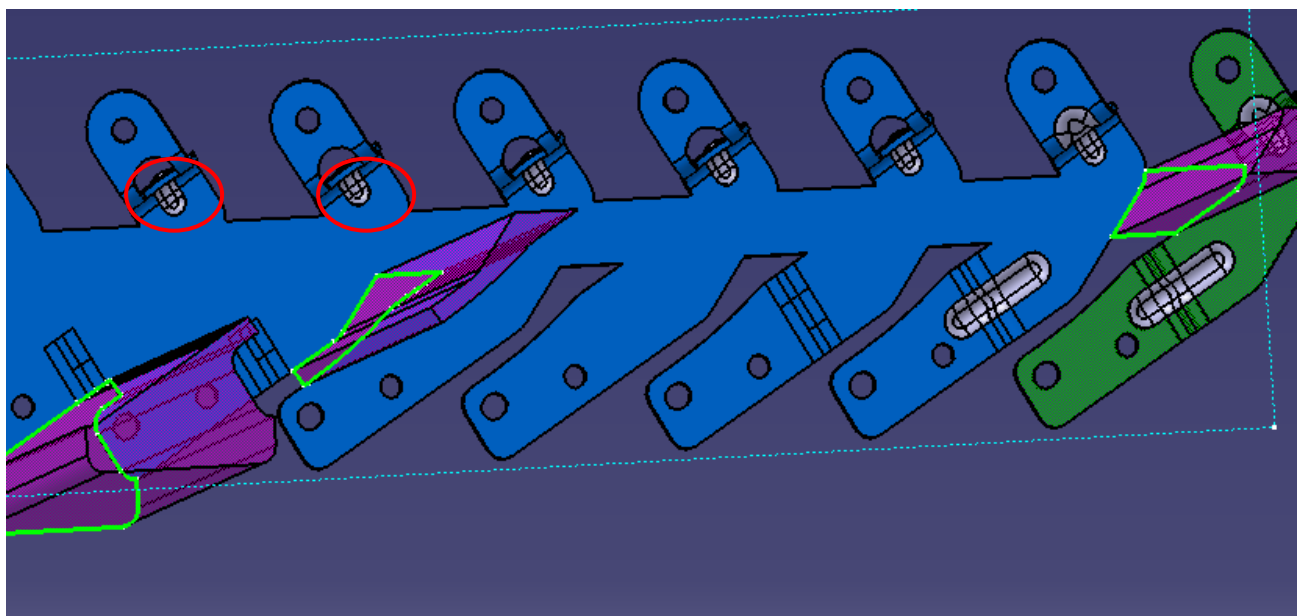
- 👤 Translating geometries to the other steps.
 - 👤 Expand the PartBody and all the removed bodies.
 - 👤 Launch the “Manual_Recovery” function.
 - 👤 Pick the solid corresponding to the second front folding in the tree.
 - 👤 **Mandatory :** Pick only the first solid of the body, not the translate, rotate or pattern features.



- 👤 Pick the Station2 : Folding two bends in the tree.
 - 👤 A new "Removed" body is created.
 - 👤 The geometry is translated to the Station2.
 - 👤 As the station has 2 free steps, the geometry is patterned to the free steps.
- 👤 As we do not need more transformations yet, press ESCAPE to end the function.

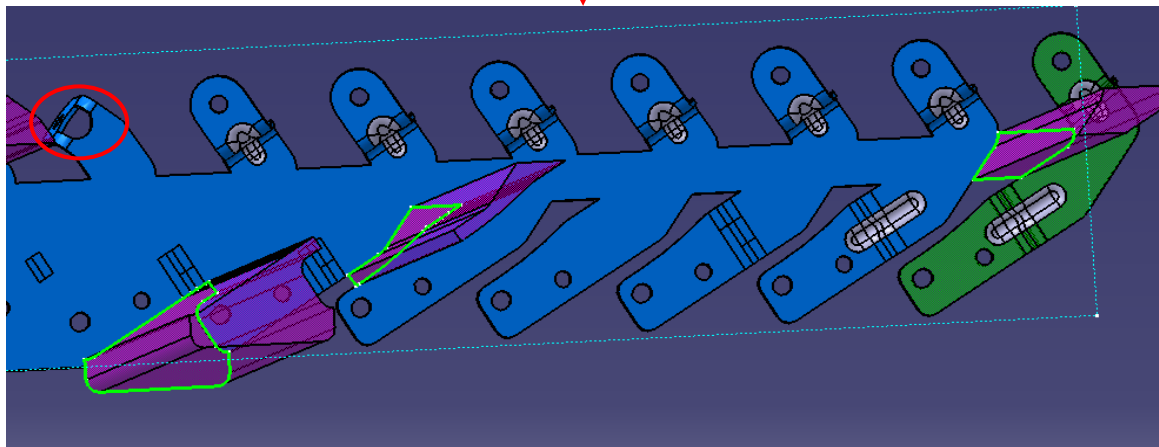
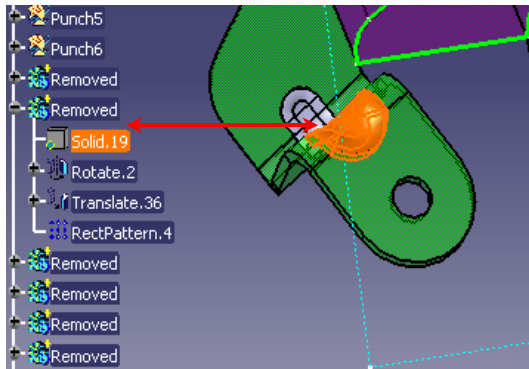


- 👤 Repeat this operation for Station3 and Station4.
 - 👤 All the geometries corresponding to the second front folding are in place.

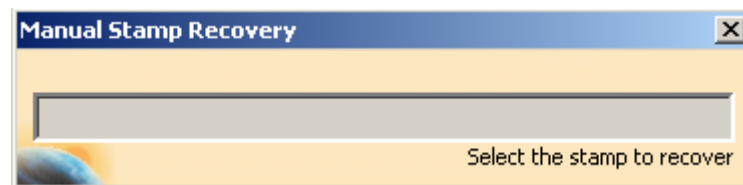


Repeat this operation for the geometry corresponding to the first front folding, on Station2, Station3 and Station4.

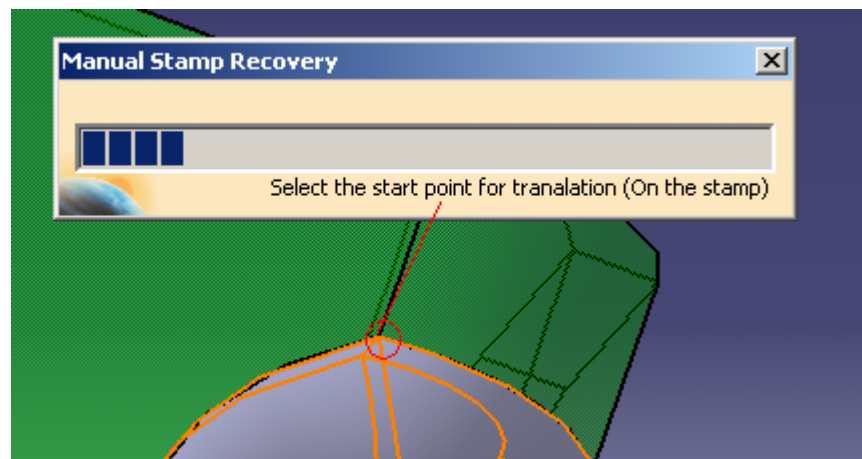
The geometries are in place, only the Station5's one is missing.



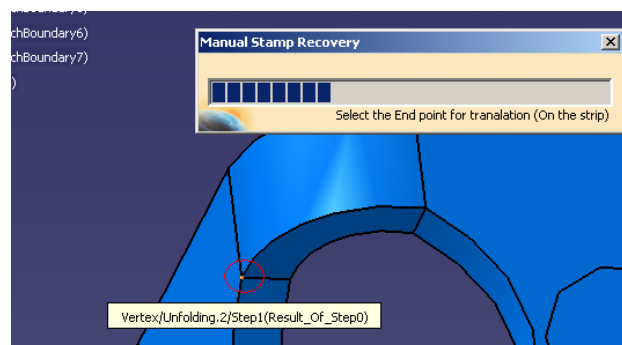
- 👤 Launch the “Manual_Recovery” function to translate the missing geometry to Station5.
 - 👤 *The geometry is close to the Station5.*
 - 👤 *It is asked to select solid.*



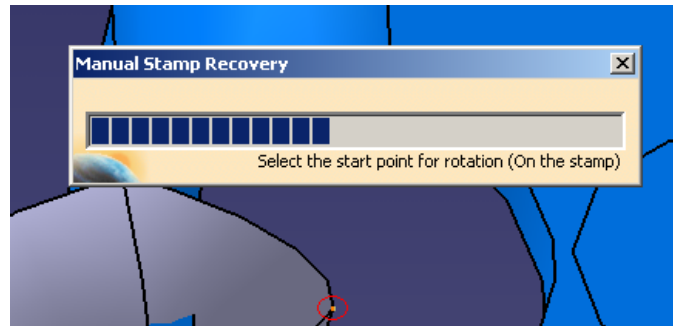
- 👤 *It is asked to select start point for translation on the stamp*



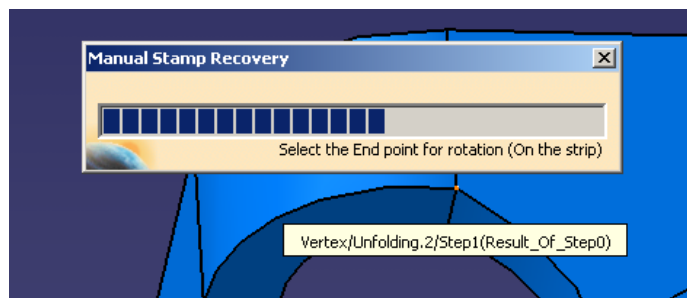
- 👤 *It is asked to select the end point on the strip*



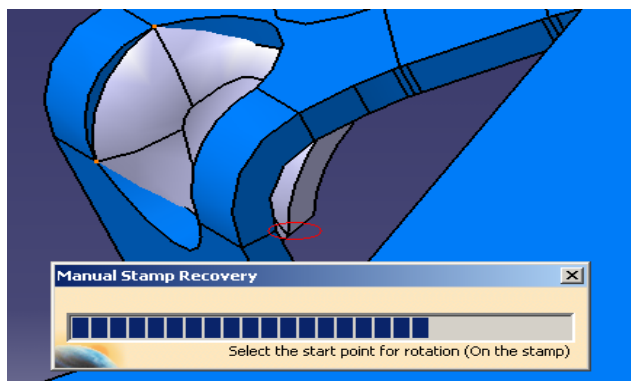
- 👤 The geometry is translated.
- 👤 It is asked to select the rotation axis.



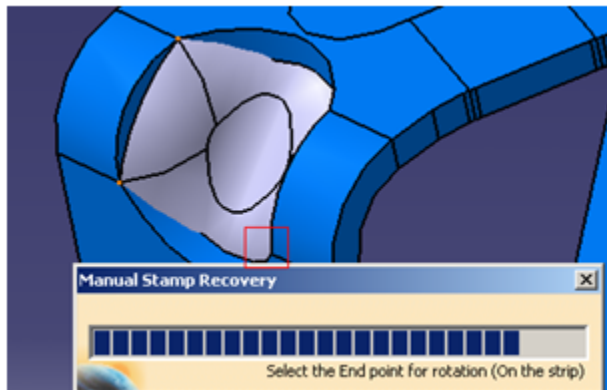
- 👤 It is asked to select the End point for rotate the geometry.



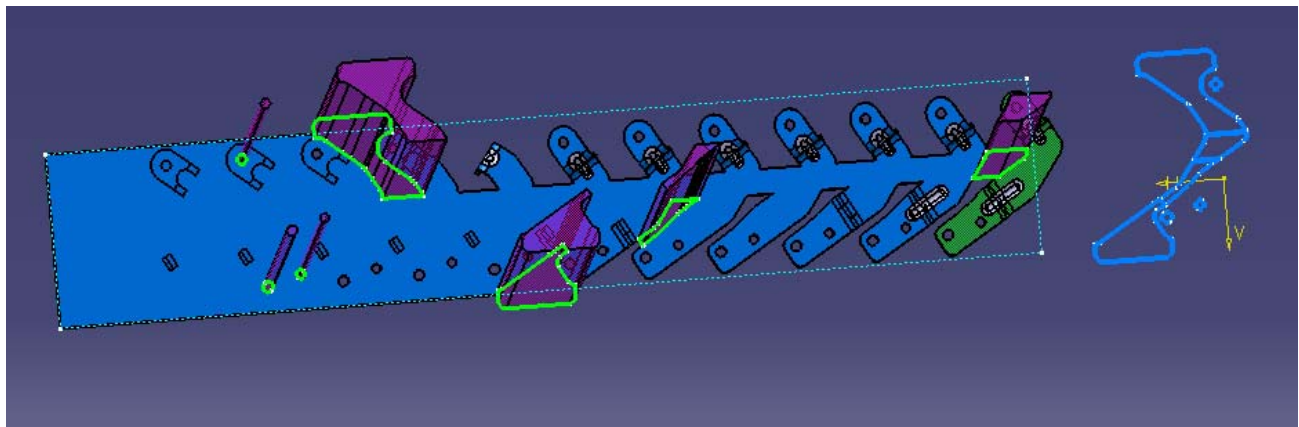
- 👤 Select a point on the geometry.
- 👤 It is asked to select the rotation start point on stamp.



- 👤 Select the corresponding point on the strip.
- 👤 The geometry is rotated.



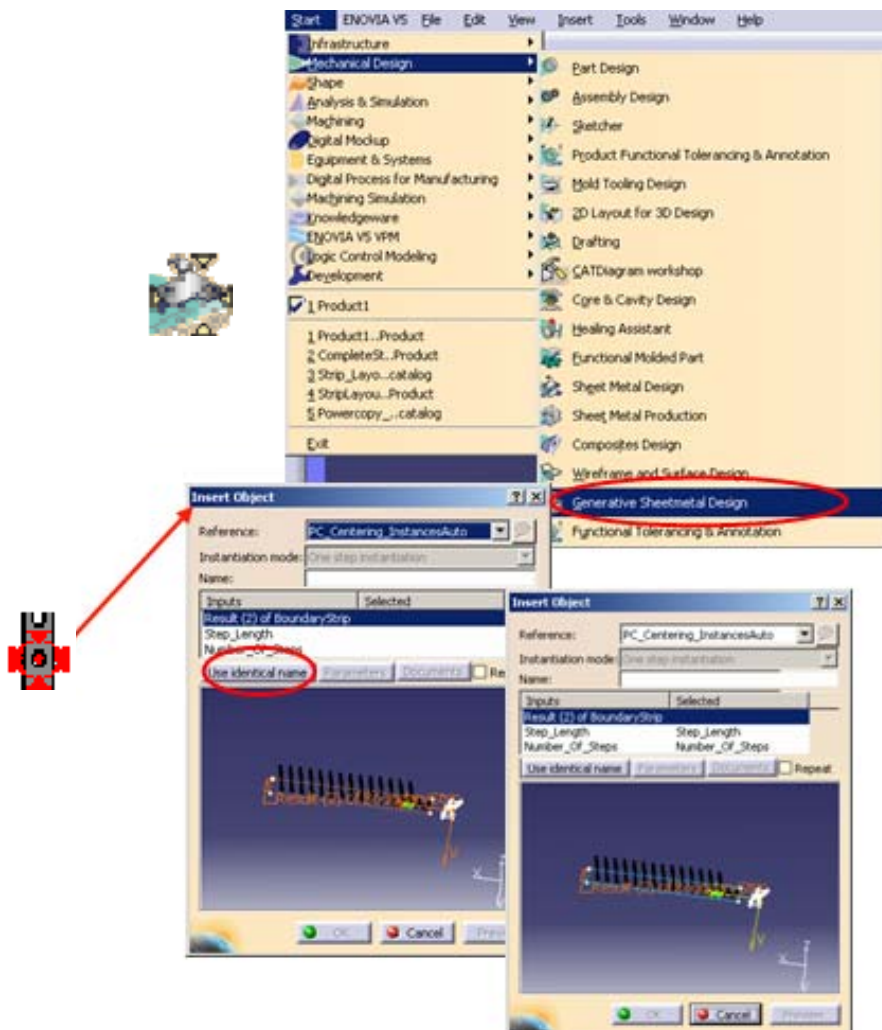
🔸 Your strip is now ready to receive the centring tools (optional)















Instantiation of the centering tools (pilots)

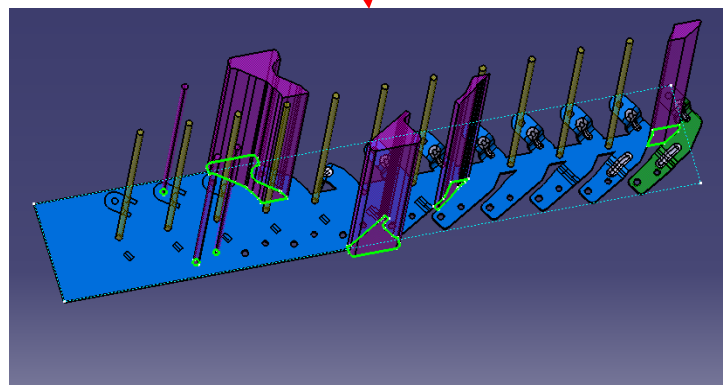
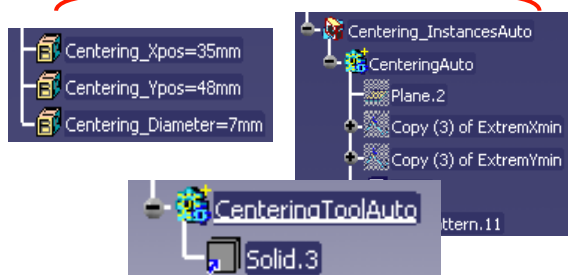
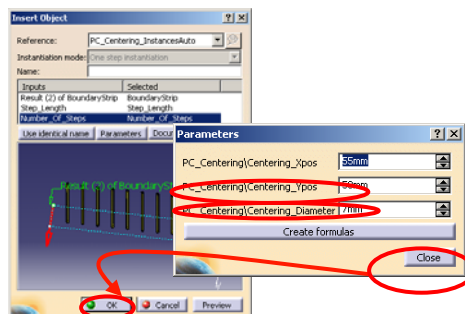


- In the methodology, centering tools may be designed in order to maintain the strip layout in the correct position
- The centering tools can be instantiated automatically by computing the number of steps:
 - 👤 Verify to be in **Generative Sheetmetal Design** workbench. If it is not the case switch to “Generative Sheetmetal Design” workbench
 - 👤 Verify that the **PartBody** body is in Work Object (underlined)
 - 👤 Select **Centering_Tools_Auto** function
 - 👤 The “Insert Object” window is displayed
 - 👤 Select **Use identical name** button
 - 👤 Inputs have been correctly selected and displayed in the “Inputs” list.




















Instantiation of the centering tools (pilots) (2 / 3)

-  Select the **Parameters** button
 -  The “Parameters” window is displayed.
-  Change the values:
 -  Set **Centering_Xpos= 79 mm**
 -  Set **Centering_Ypos= 57 mm**
 -  Set **Centering_Diameter= 7 mm**
-  **Close** the “parameters” window
-  Check with **preview** that the pilots are in the right position, either, modify the previous parameters
-  Validate with the **OK** button
 -  The centering tools have been all instantiated and displayed
 -  The features used to remove the material of the strip layout and schematise the tools are displayed in the tree
 -  The parameters are displayed in the tree



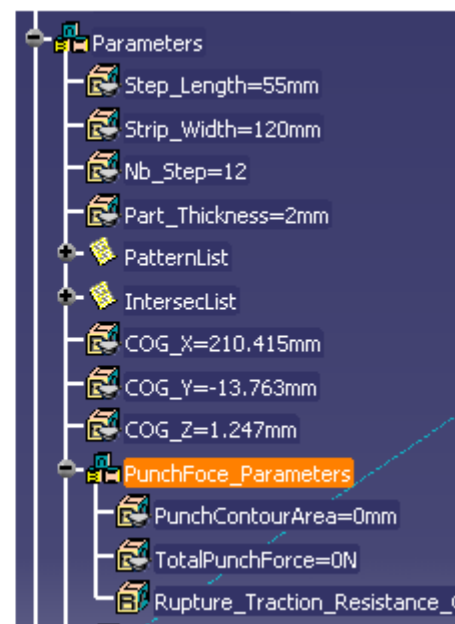
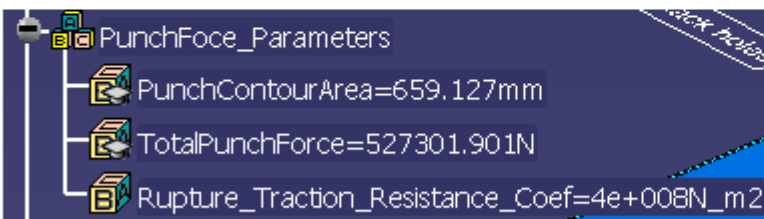
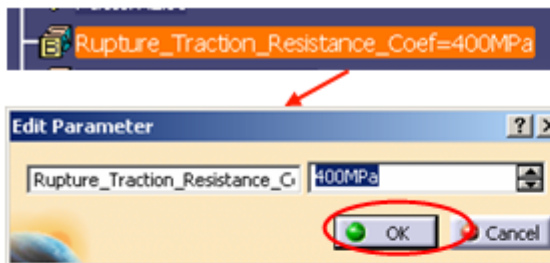
Instantiation of the centering tools (pilots) (3 / 3)

- ▶ The centering tools can also be instantiated manually if the user wants to instantiate his own number of tools:
 -  Verify to be in **Generative Sheetmetal Design** workbench. If it is not the case switch to "Generative Sheetmetal Design" workbench
 -  Select **CenteringTools_Manu** function
 -  *The "Insert Object" window is displayed*
 -  Select **Use identical name** button
 -  *The "Step_Length" input have been correctly selected and displayed in the "Inputs" list.*
 -  Select the "parameters" button.
 -  *The "Parameters" window is displayed.*
 -  Change the values:
 -  Set **Centering_Xpos= 79 mm**
 -  Set **Centering_Ypos= 57 mm**
 -  Set **Centering_Diameter= 7 mm**
 -  Set **Instances= 11**
 -  **Close** the parameters window
 -  Validate with the **OK** button
 -  *The centering tools have been all instantiated and displayed*
 -  *The features used to remove the material of the strip layout and schematise the tools are displayed in the tree*
 -  *The parameters are displayed in the tree*

Computation of the punching force

➤ If the user wants to know the force necessary to perform the punching operation,

- ✎ Edit "Rupture_Traction_Resistance_Coeff" by double-clicking on it and put an appropriate value.
- ✎ Total Punch Force will automatically calculate and will be shown in parameter "TotalPunchForce".

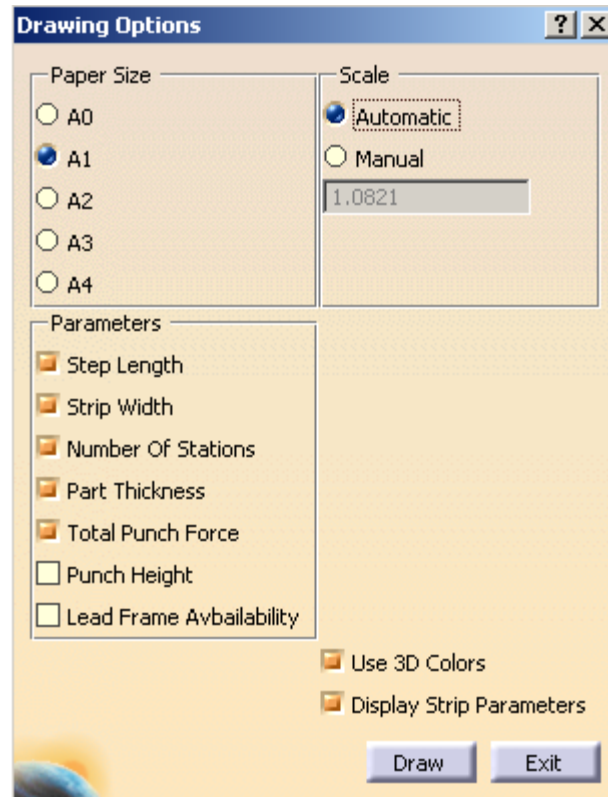




Drawing generation

➤ This methodology offers you the possibility to compute the top view of your strip.

- 👤 Activate the root product.
- 👤 Launch the “Automatic Drawing” function.
- 👤 The Drawing options window is displayed.



- 👤 Select your paper size in the “Paper Size” area.
 - 👤 *The paper will always be in portrait orientation.*
- 👤 Select the scale computation method in the “Scale” area
 - 👤 *If you have selected the automatic scale, it will be recomputed at each paper size modification.*
 - 👤 *If you have chosen the manual scale, you have to enter it manually.*
- 👤 If you want your drawing to be in colors, select the “Use 3D colors” option.
- 👤 If you want to have a table with the strip parameters in the bottom left corner of the drawing, select the “Display strip parameters” option.
 - 👤 *A table with number of steps, step length, strip width, strip length, strip thickness and wasted metal will be displayed.*
- 👤 Click Draw
 - 👤 *The drawing is generated.*
 - 👤 *The name of each station is written in front of them. Free steps do not have name.*

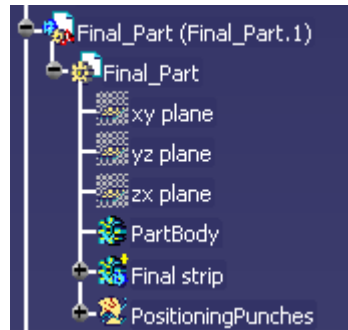
➤ Let’s have a look to a colour drawing in next page.



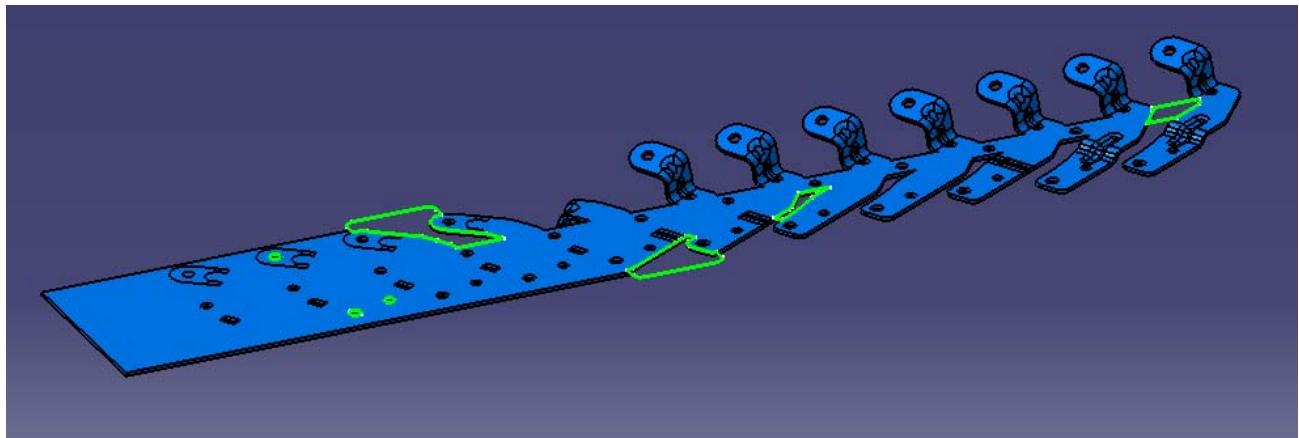


Final part computation

- At the end of the strip layout process, you are able to generate a final part to give to the tool designer.
- This part will be as simple as possible (one body and one geometrical set), without any link to other parts and will contain the geometry of the strip and the punch contours.



- Now, let's compute the Final part.**
 - Activate the root product.
 - Launch the final part function
 - The final part is computed.*
 - Hide the "Strip_Construction" Part.
 - You can see the final strip.*



Conclusion

- 🔥 **This methodology will help to create the whole strip layout**

- 👤 The first stage is the preparation of the strip

- 👤 *Preparation of the part*

- 👤 *Creation of the punches' contours*

- 👤 *Copy as result with link of the mandatory features in the document on which the strip layout design will be done*

- 👤 The second stage is the design step by step, driven by the user know how, of the strip from the finished part to the starting plate

- 🔥 **A result can be found under**

- ...\StripLayout_End\Final_Strip.CATProduct**