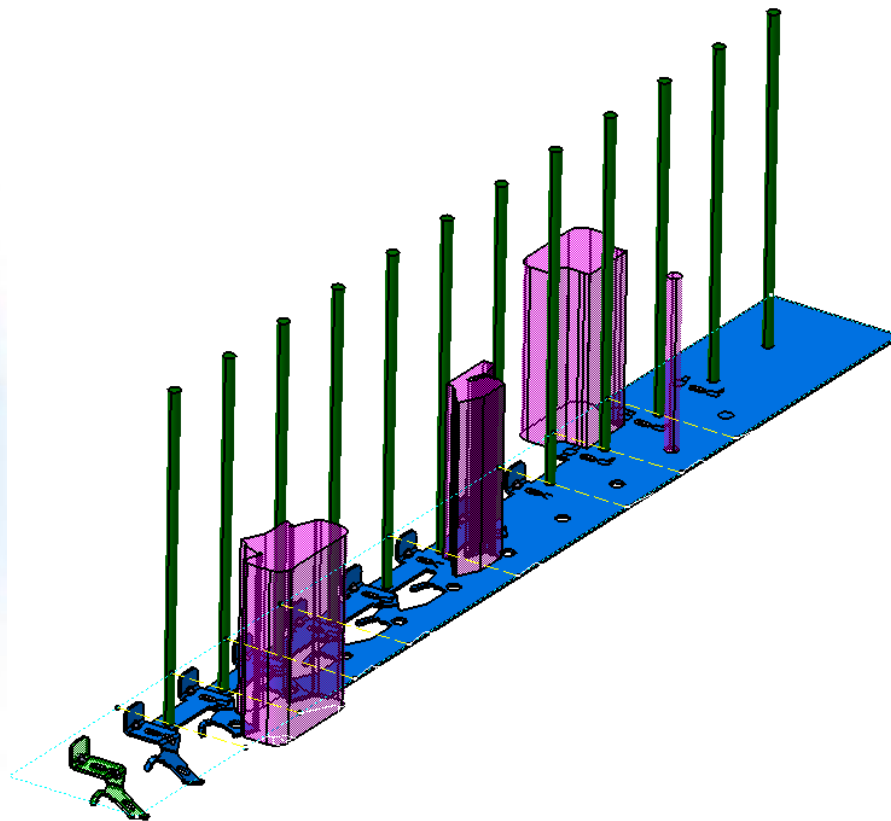


# Progressive Die Strip Layout Design (SL9)

## BPA Delivery 7 for V5R19 (V5.7)




### Implementation Guide



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# Introduction

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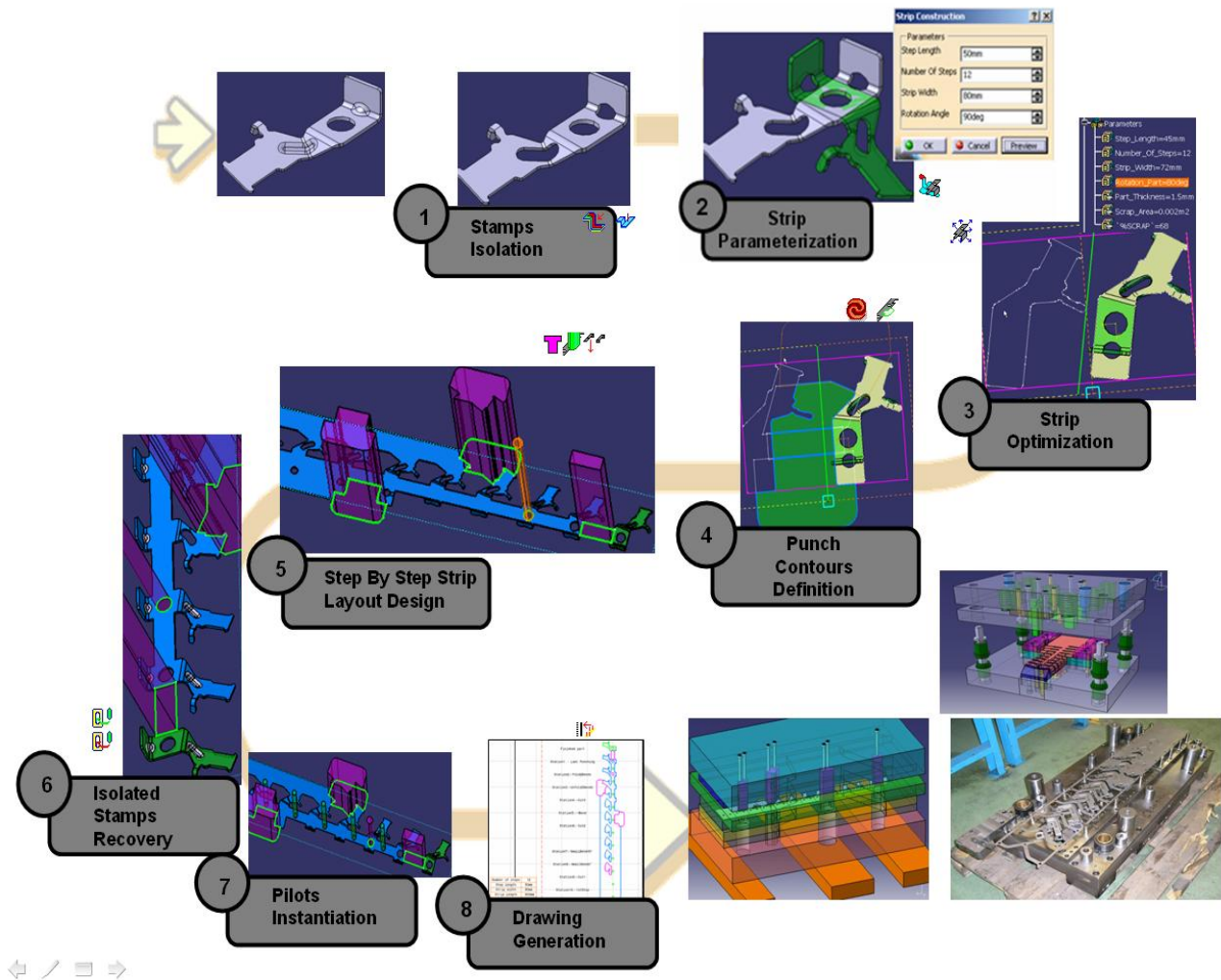
- 🔥 This document is part of a set of documents that will develop a CATIA V5 methodology to define a Strip Layout for Progressive Dies business.
- 🔥 This document will describe
  - 👤 All the specific functions developed for the Strip Layout methodology,
  - 👤 A scenario that will be used for demo or a step by step exercise for training.
  - 👤 Some hints and tips.
  - 👤 Please check that all these documents are available
- 🔥 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***

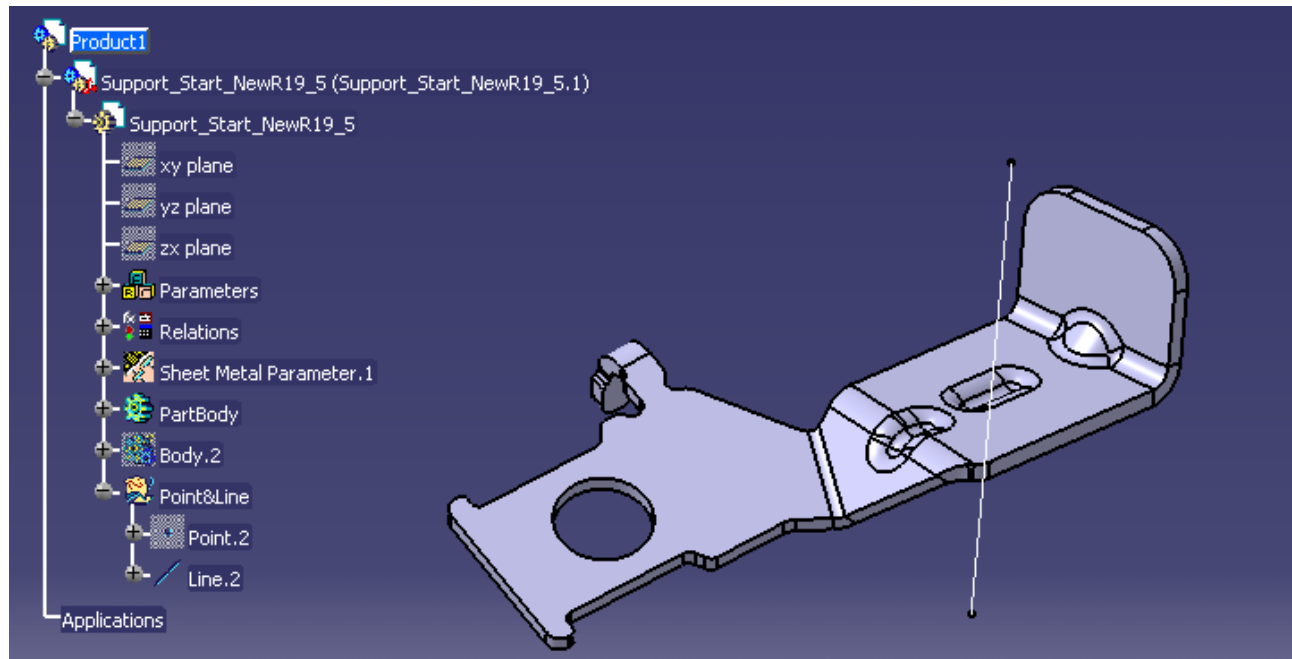
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- The aim of this methodology is to build the strip layout design of a metal part. Starting from an imported IGES part or from a V5 part, this methodology, based on dedicated specific functions defined on a BPA, will enable the user to design the strip 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 Sheet metal Design features:

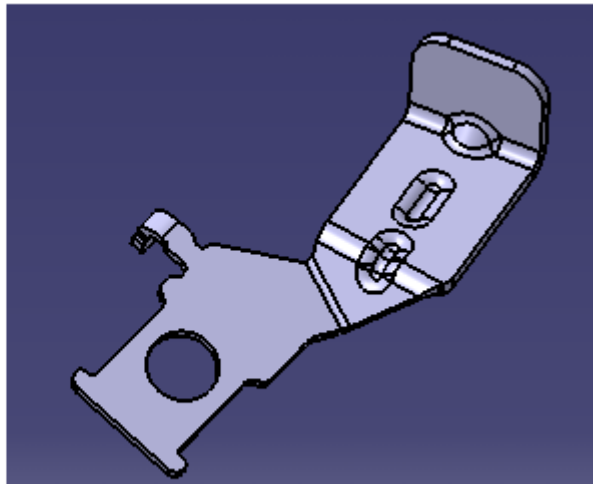
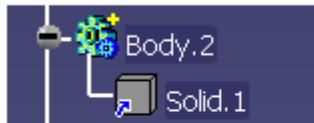
-  In the Generative Sheet metal Design workbench:
  -  Use Generative Sheet metal tools:
    -  Wall, Wall on Edge, Extrusion...
    -  Flanges, Stamps...
-  Once Part is created using Sheet Metal Features, to Proceed with Strip design, Copy the created body and Paste Special "As a Result with Link All Views". This will Create Sheetmetal Solid.





Or

### Import this part from an IGES or V4:

-  The imported solid is a datum solid.

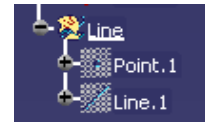
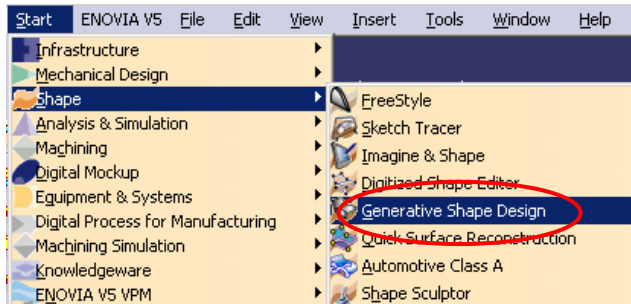


### MANDATORY:

-  If you want to use a part with more than one body, do not use "Isolated" as body name. This name is used by the automation functions, using it may interfere with the complex geometries treatment.
-  The upper face of the final strip will be coincident with the XY plane. That means the upper face of one of the faces of the part that will not move after unfolding operations within the strip must be coincident with XY plane.

## Methodology starting point creation

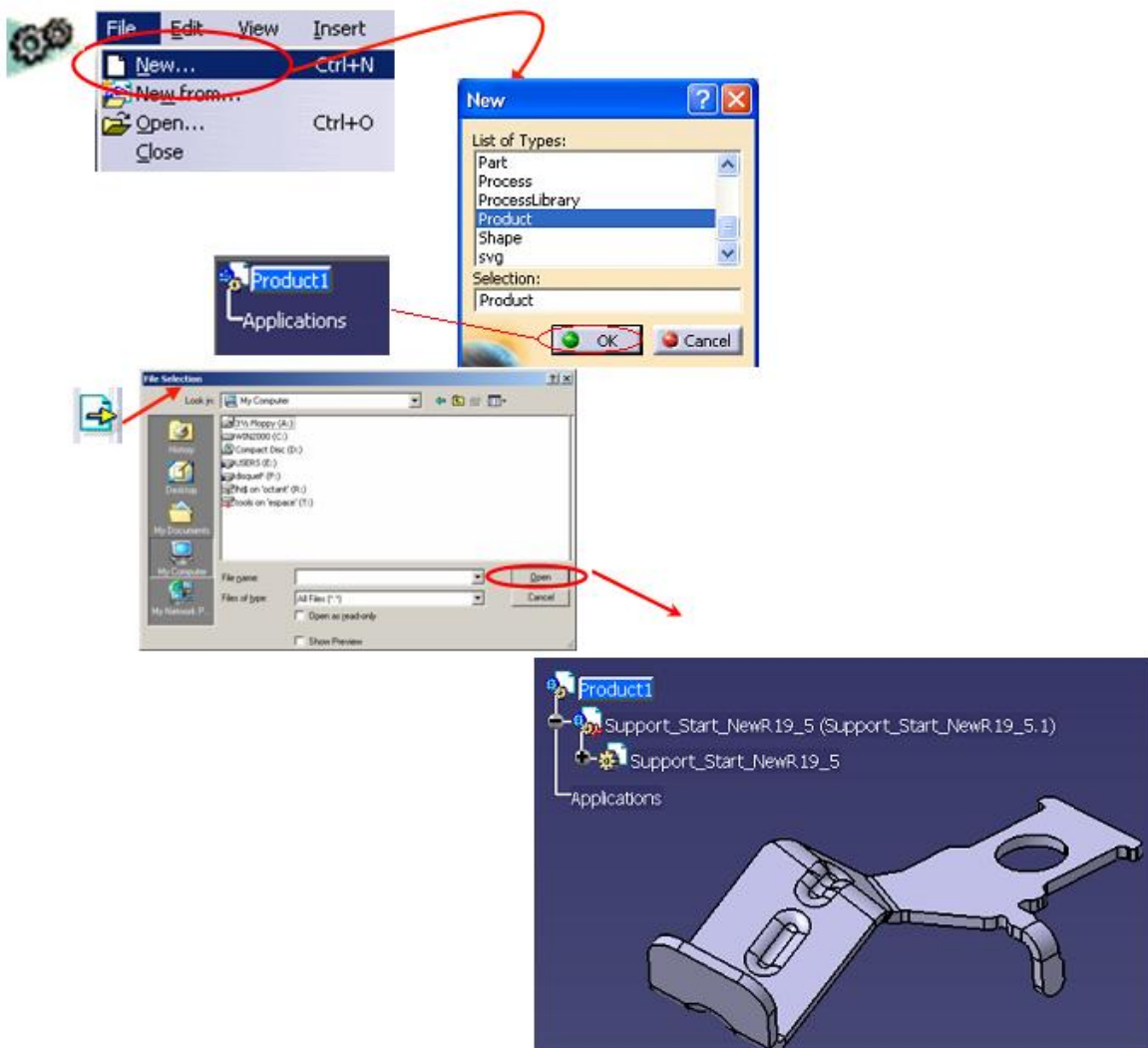
- In order to use some automatisms, the Part must have some specific publications (A solid and a Line). So, whatever it is, a IGES, a V4 or a Sheet metal Design Part, we must retrieve these data.  
 Be sure that the part is in the correct orientation,
- In Generative Shape Design workbench create a line 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

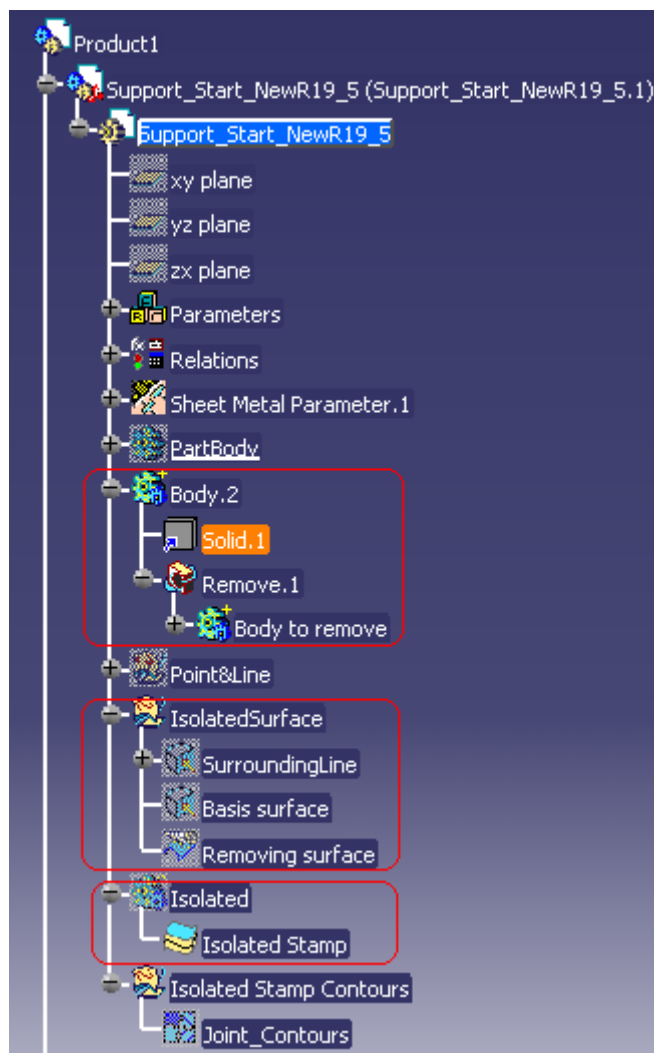
## Preparation of the structure (1 / 2)

- 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.
  - Invert Orientation
    - This is helpful in inverting the Orientation of Isolation of the Stamp
- 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 "Isolated" containing the isolated area. In a case where we have a part containing the Sheet Metal features we do not need this function.

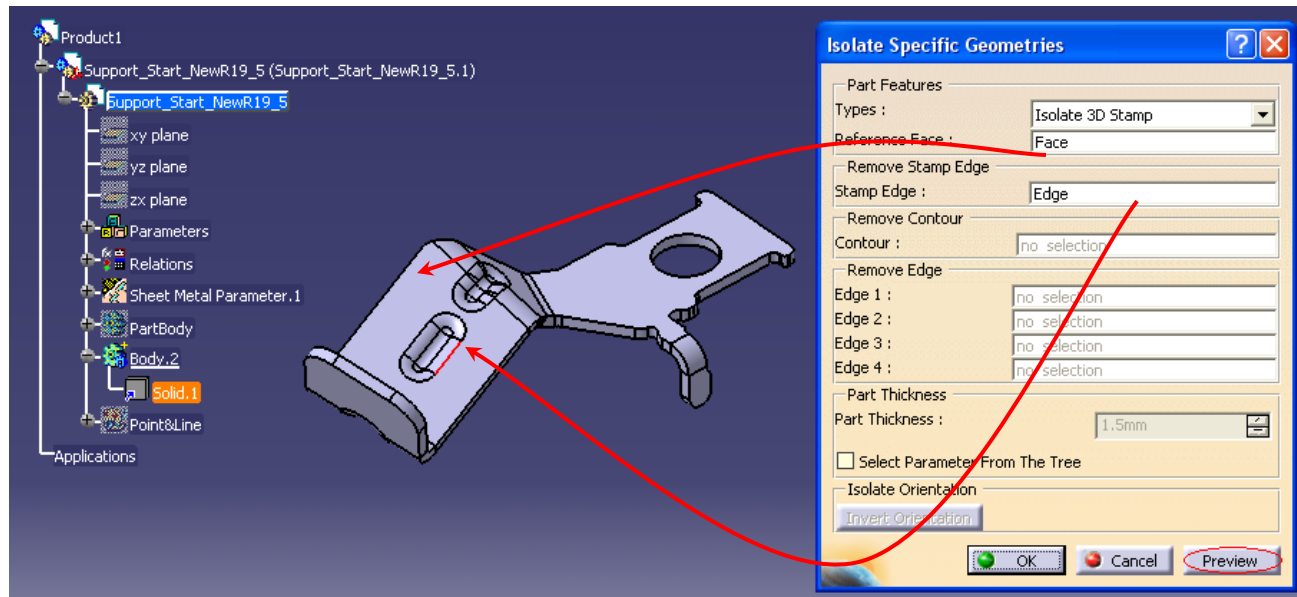


- Mandatory:** You need to work in a product structure for all the functions to work. If Isolation is not being done in part Body user will have to define in work Object the body in which isolation is being done.

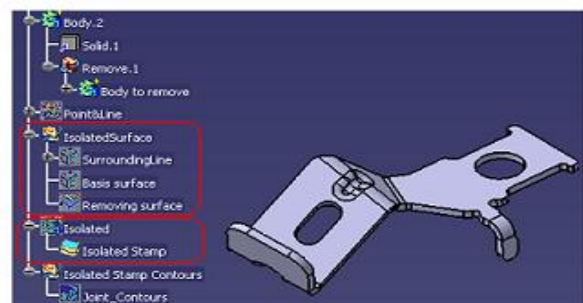
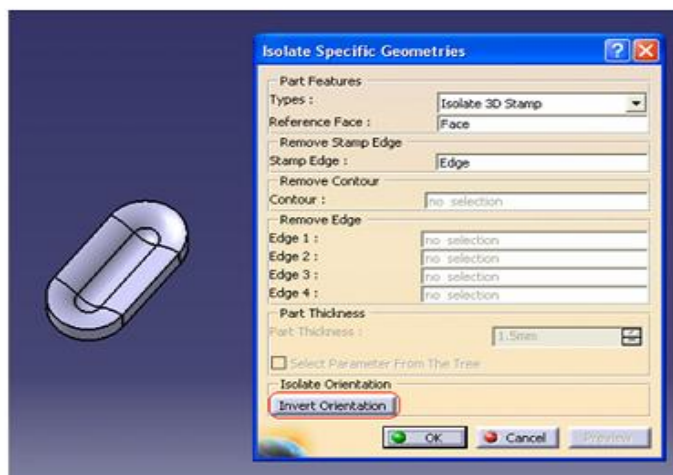


## Isolate specific geometries: Stamp Isolation

- This function is specialized in 2D and 3D stamps isolating.
  - 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.



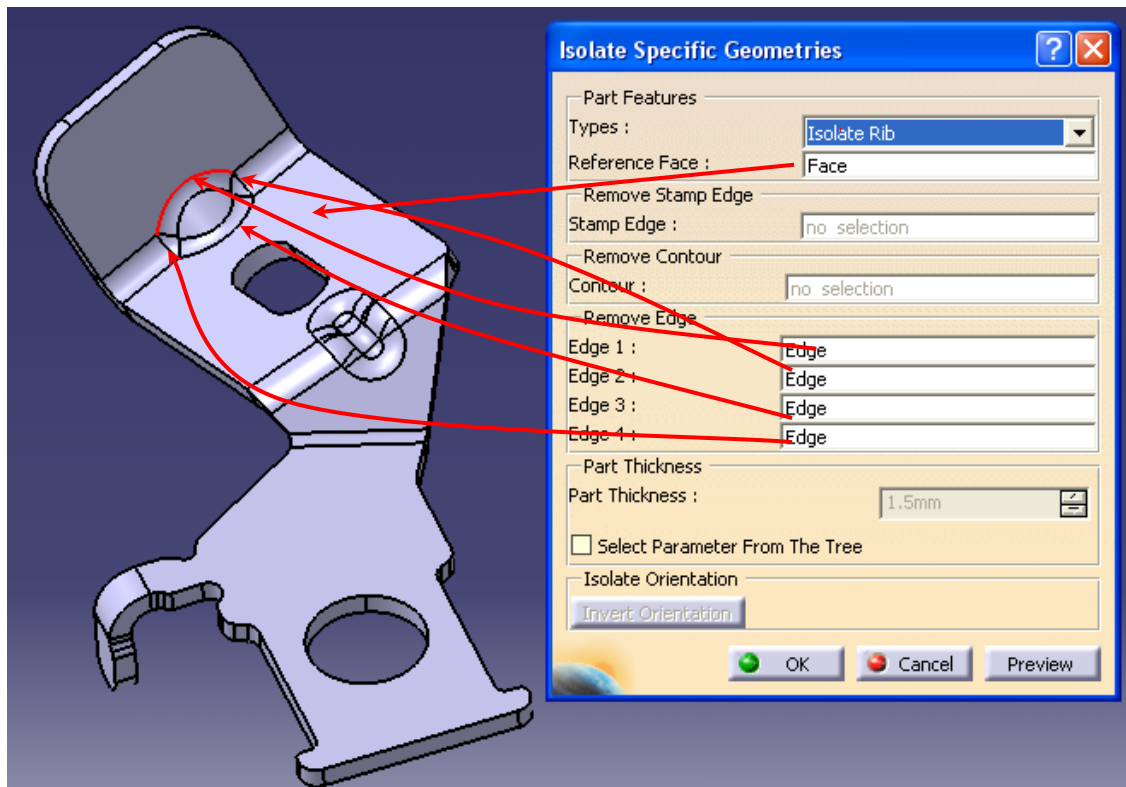
- Press Preview.
  - The stamp is isolated.
  - Press Invert Orientation.



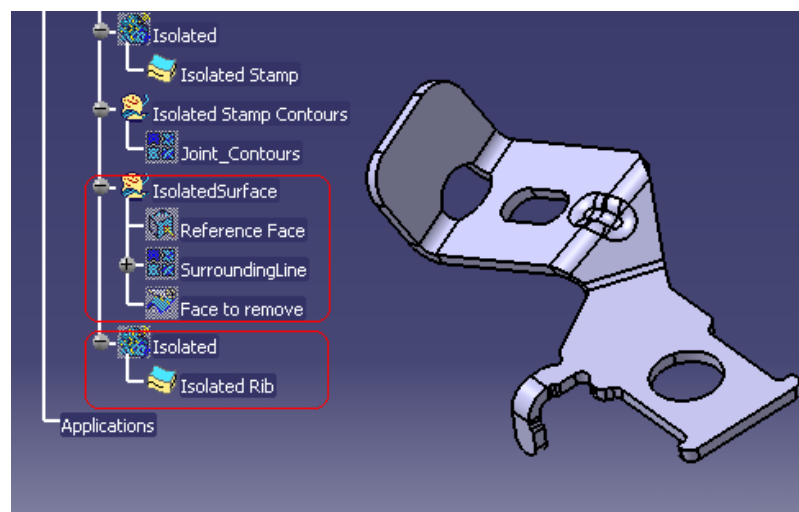
## Isolate specific geometries: Rib Isolation

➤ 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

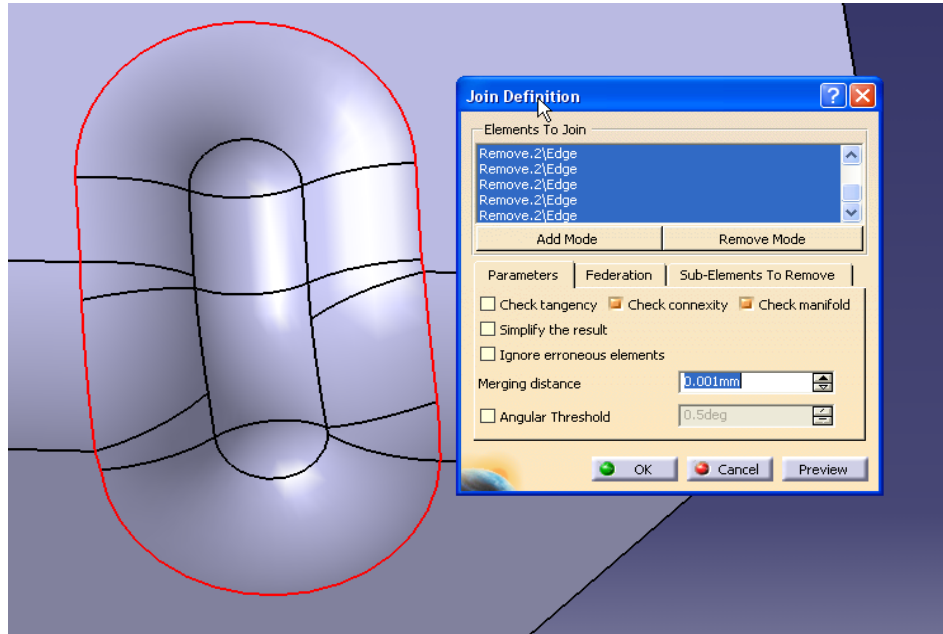




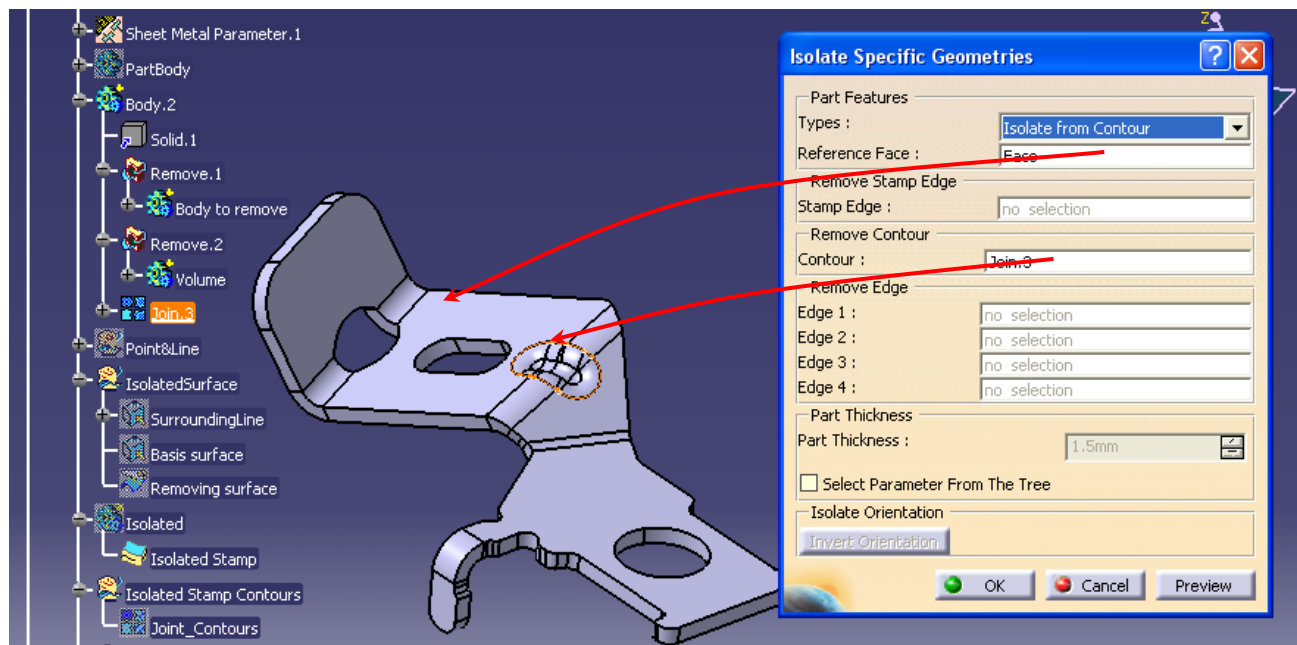
## Isolate specific geometries: Isolation from Contour

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

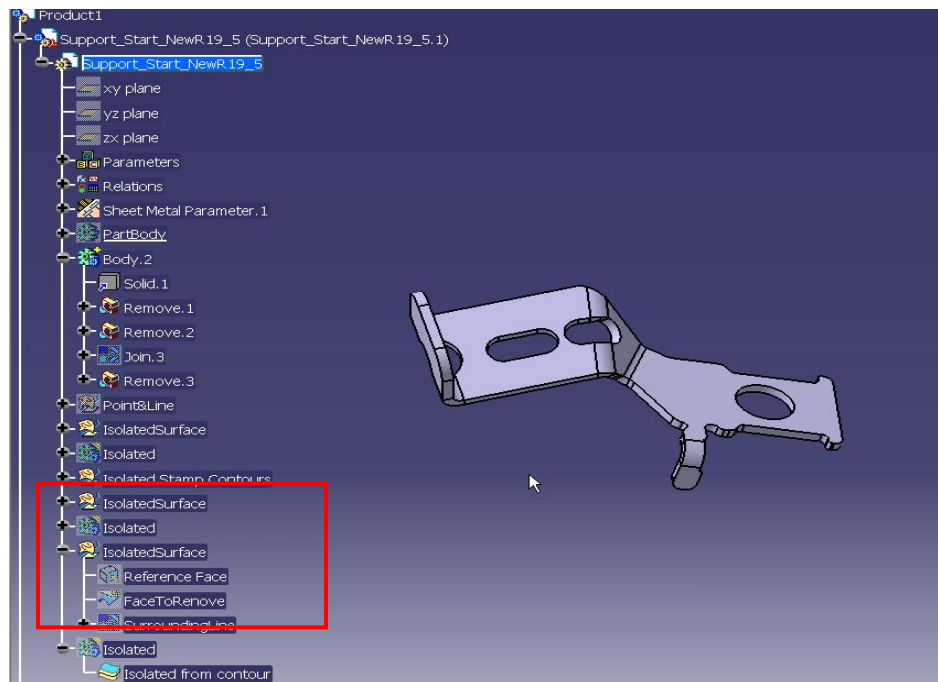
👤 Click on the “Join” icon and pick all the edges of your geometry's contour.



- 👤 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.
- 👤 Join can also be created by Right Clicking in the Contour Edit Field and then pressing Create Join button.



- Click OK
- The geometry is isolated.



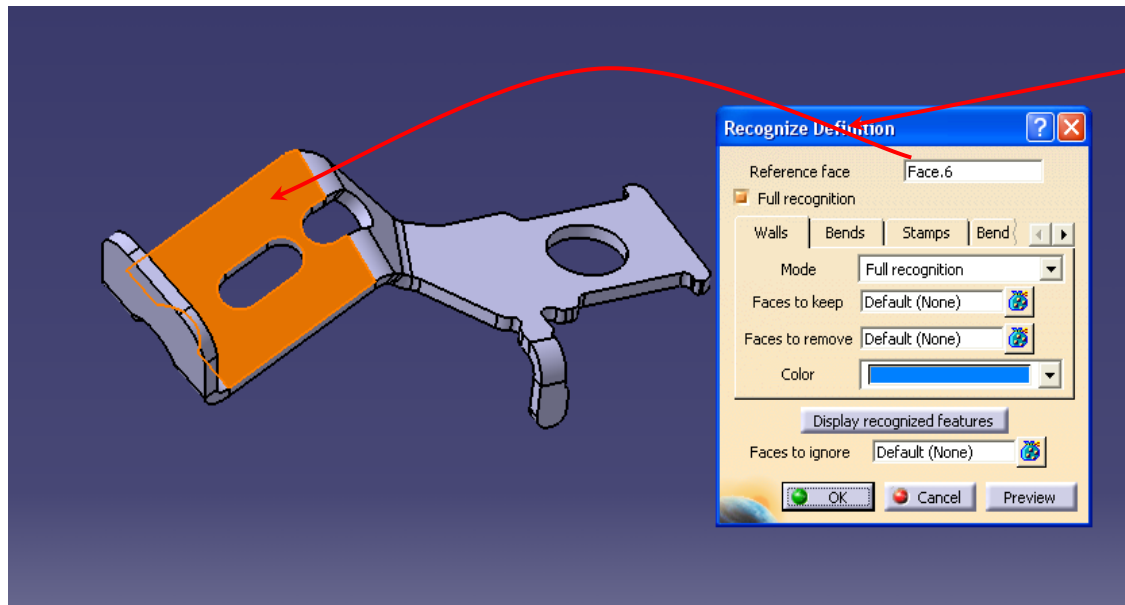
## Invert Orientation

Invert Orientation is required sometimes when the orientation of stamp is not correct. Due to this there may be a problem with Isolation of Geometries. In order to solve this problem Invert Orientation is provided which will give the correct result for Isolation of Complex Geometries. This button is activated only on Preview. This case is already shown in the First Isolation in the guide.

### Mandatory:

- After isolation of complex geometries, Recognize has to be done. If part is sheet metal featured, we do not need isolation.
- If No Isolation is Done (i.e. No Complex Geometries are to be removed) and whole Part Is designed with Sheet-Metal Features, Then a Copy Paste Special As a Result with Link All Views is to be done which will create a Sheet Metal Solid. This Sheet Metal Solid can be used directly for Next Steps of Strip Layout Design.

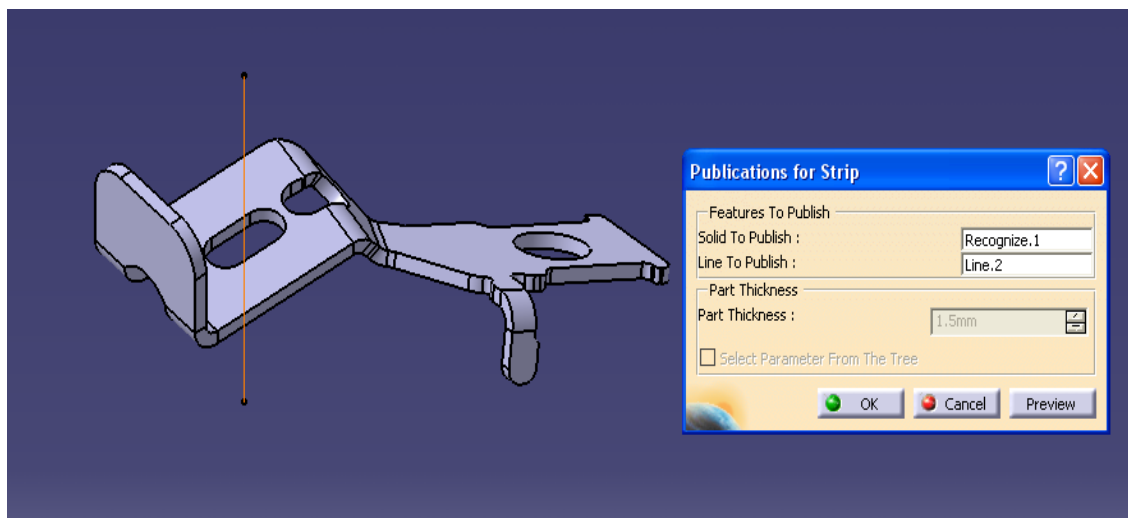




## Publishing the elements



- 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\_Contours and the rotation line are published.



- ▶ **Save the Product**
- ▶ **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***

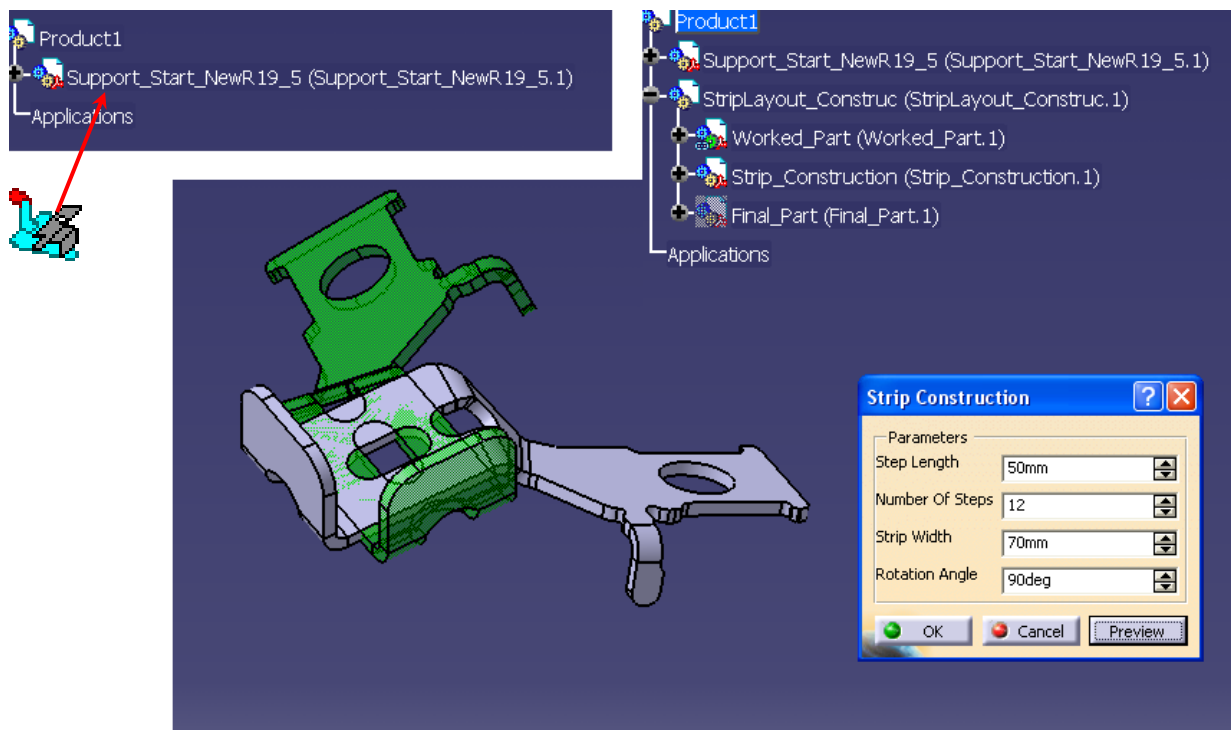
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## ***Introduction***

- **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.
  
- **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)

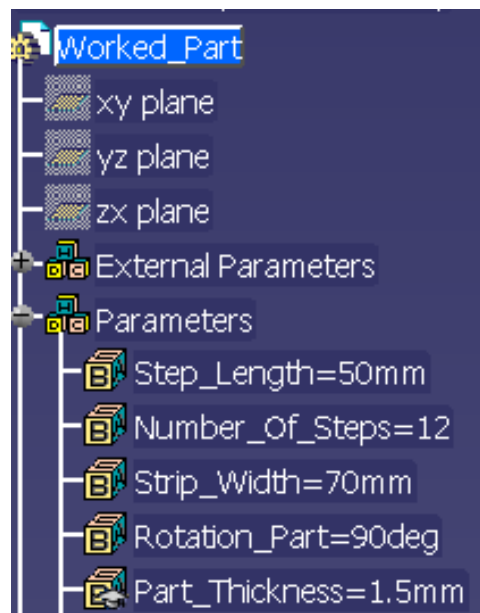
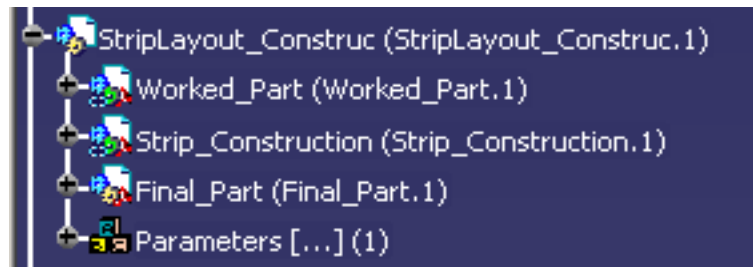
- Click on the Strip Construction dedicated icon and run this function to Prepare a Structure for Strip layout Design.
- ✎ Edit the new product by double-clicking on it in the tree.
  - ✎ The new product is highlighted in blue in the tree
- ✎ Select the **StripLayout Construction** function
- ✎ Set the **Step\_Length** parameter to **50mm**.
- ✎ Set the **Number\_Of\_Steps** parameter to **12**.
- ✎ Set the **Strip\_Width** parameter to **70mm**.
- ✎ To orientate the part along X-Axis set the **Rotation\_Part** parameter to **90°**
- ✎ Check with the **Preview** button
  - ✎ A new Structure is created which has a Strip\_Layout Construction product and three parts namely "Worked\_Part", "Strip\_Construction" and "Final\_Part".
  - ✎ The instantiated "Worked\_Part" is displayed (in transparency) and the strip layout tree is built.
  - ✎ Try Changing the Rotation angle and Press Preview again. We can modify the parameters after Preview.







## Structure description

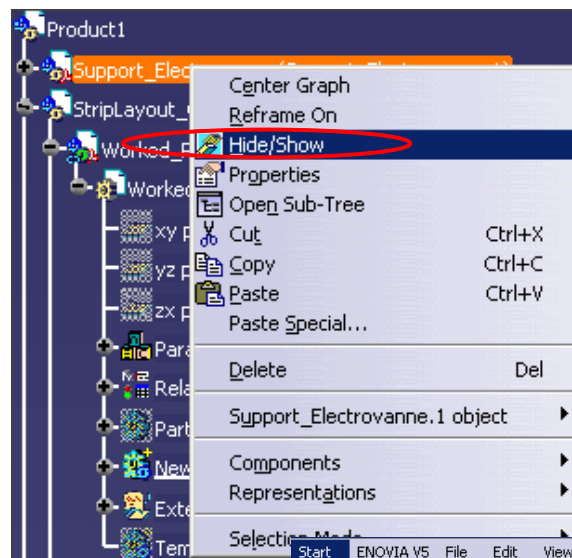
- The "StripLayout Construction" document template instantiates a product document "StripLayout\_Construc.CATProduct" made of two parts documents "Worked\_Part.CATPart" and "Strip\_Construction.CATPart"
- "Worked\_Part.CATPart" contains the "Worked\_Part" part on which the operations of retrieving punches 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

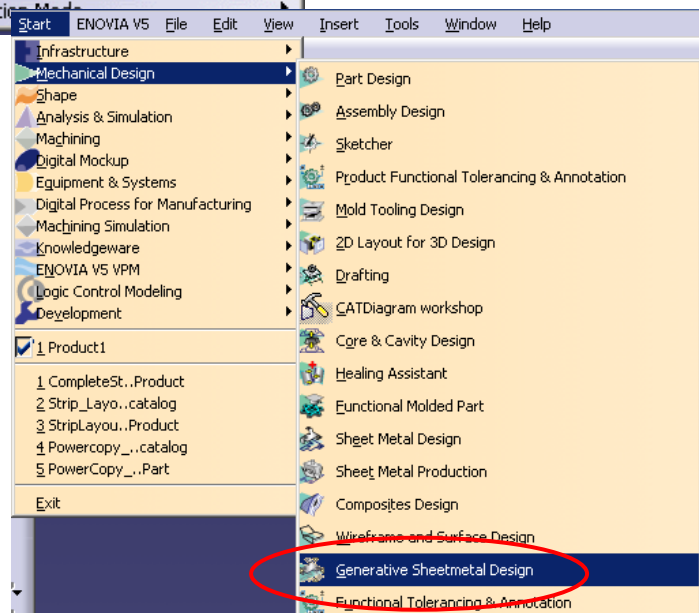
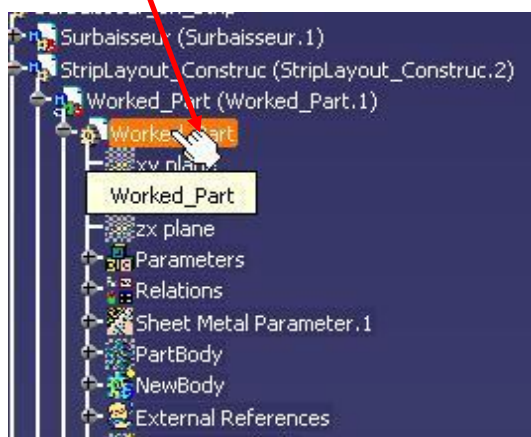


## Parameters of the strip layout (1 / 4)




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

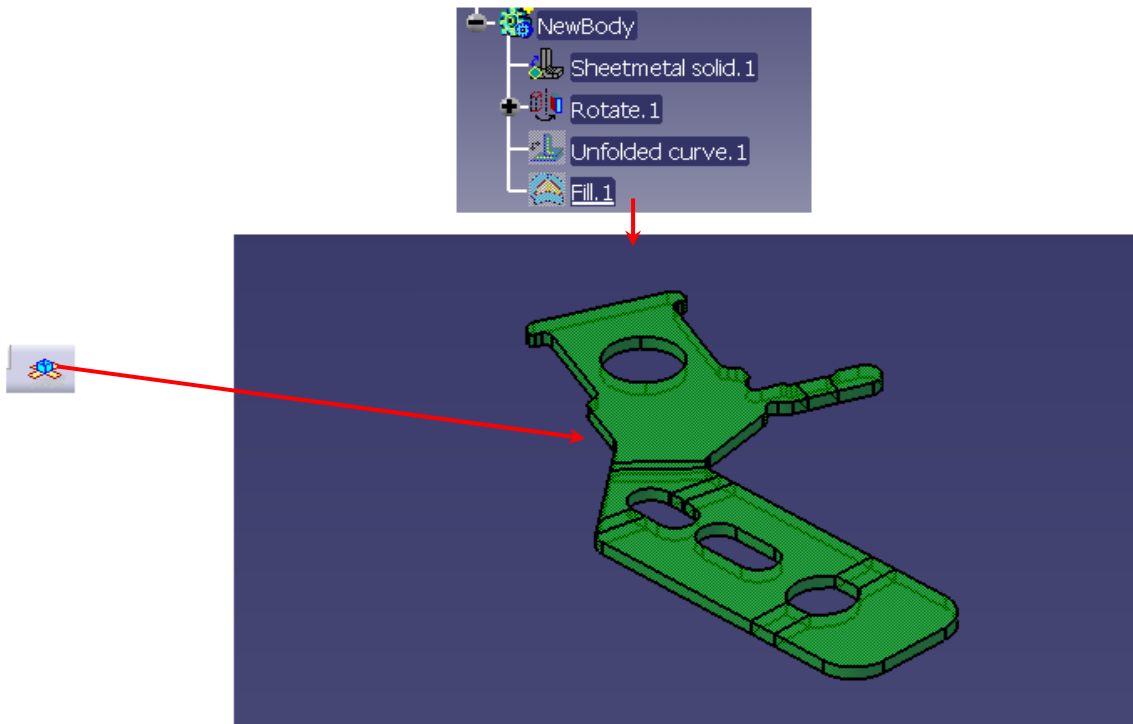


**Double Click**














## Parameters of the strip layout (2 / 4)

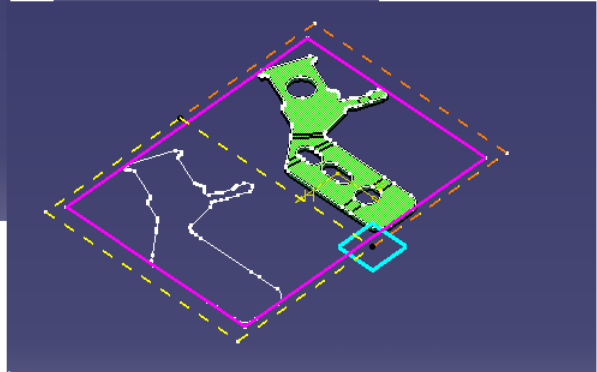
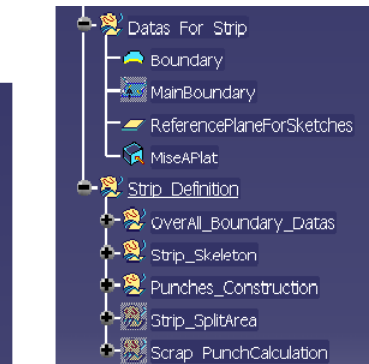
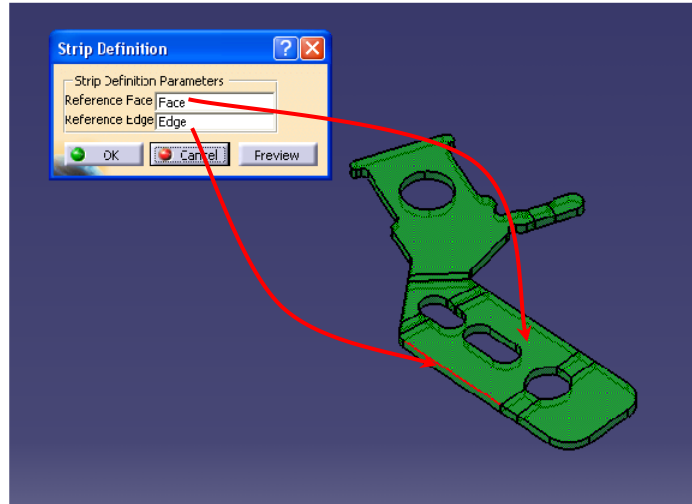
- ▶ Verify that the body “NewBody” is in work object (underlined), if not right-click on it in the tree and select “Define in Work Object”.
  -  The body “NewBody” is underlined.
- ▶ Unfold the part to get the unfolded result:
  -  Select the **Fold/Unfold** icon
  -  The unfolded result is displayed.



## 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 “Reference Face” in the list.
  -  Select the **edge** as shown on the picture on the right
    -  The selected edge is displayed as “Outer Edge” 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.

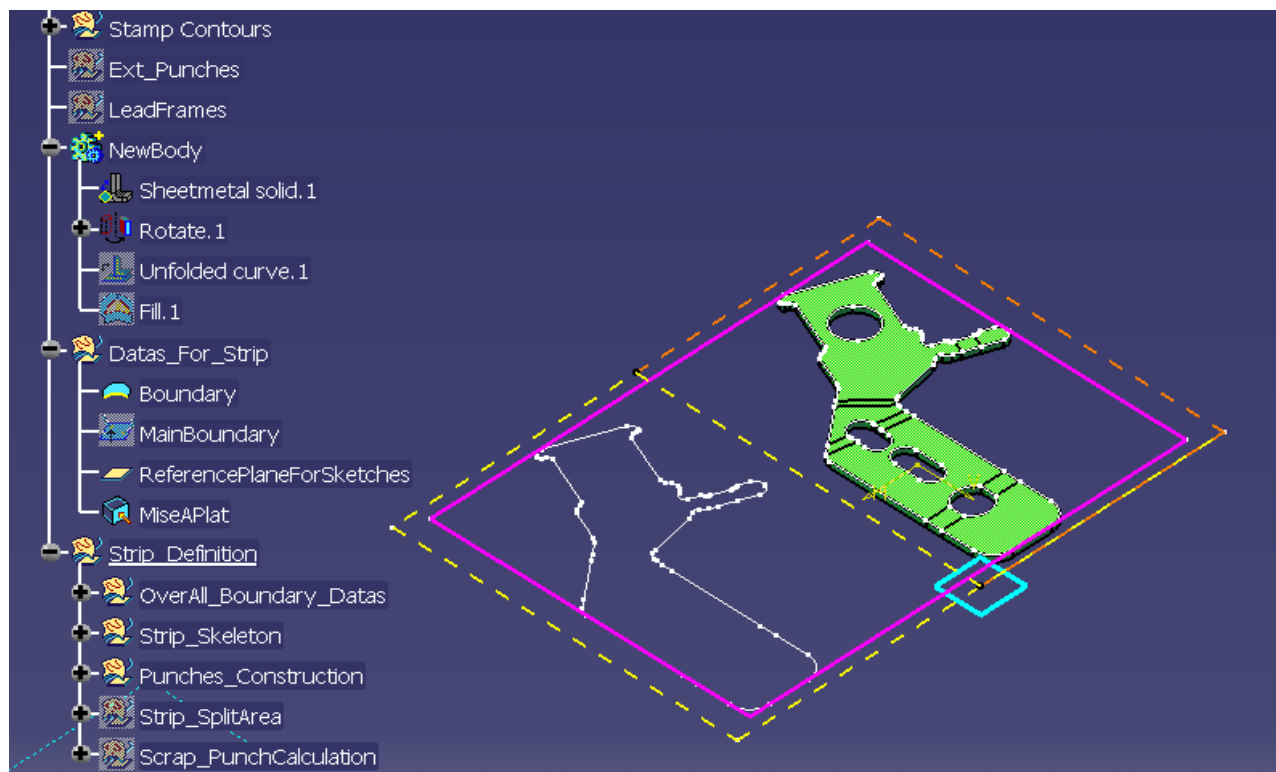


## Parameters of the strip layout (4 / 4)

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 Strip\_SplitArea which has the External Scrap Area based on the input parameters.
  - 👤 Scrap\_PunchCalculation which has the Scrap Area fill.





## Creation of punch sketches (1 / 2)



### Mandatory:

- ✎ Define the “Sketch\_Punches” open body as in work object to create the sketches in it.
- ✎ In Order to obtain a correct result for End of Strip it has to be ensured that the Punches remove the whole of scrap Area which can be seen through the Show Scrap Area Function.
- ✎ Punches cannot be created for sketches with Multiple Domains.

- ✎ Click on **Sketch Contour icon**.

↳ The “Sketch\_Punches” open body is underlined.

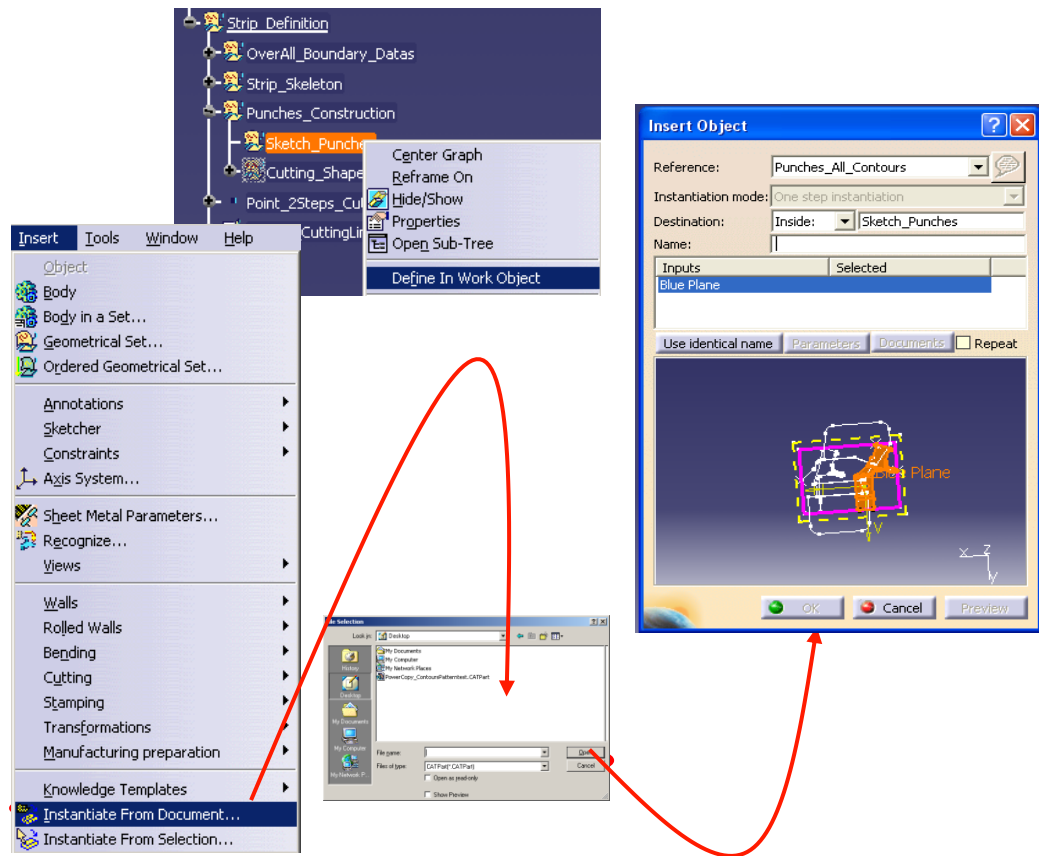
- ✎ To create all the sketches, it is possible to manually use the sketcher tools and the “ReferencePlaneForSketches” as support (blue plane). As the aim of this methodology is not to build sketches, the time of sketches creation will be saved by instantiating a powercopy containing the punch sketches.

- ✎ Select **Insert / Instantiate from Document...**





↳ The “File Selection” window is displayed.

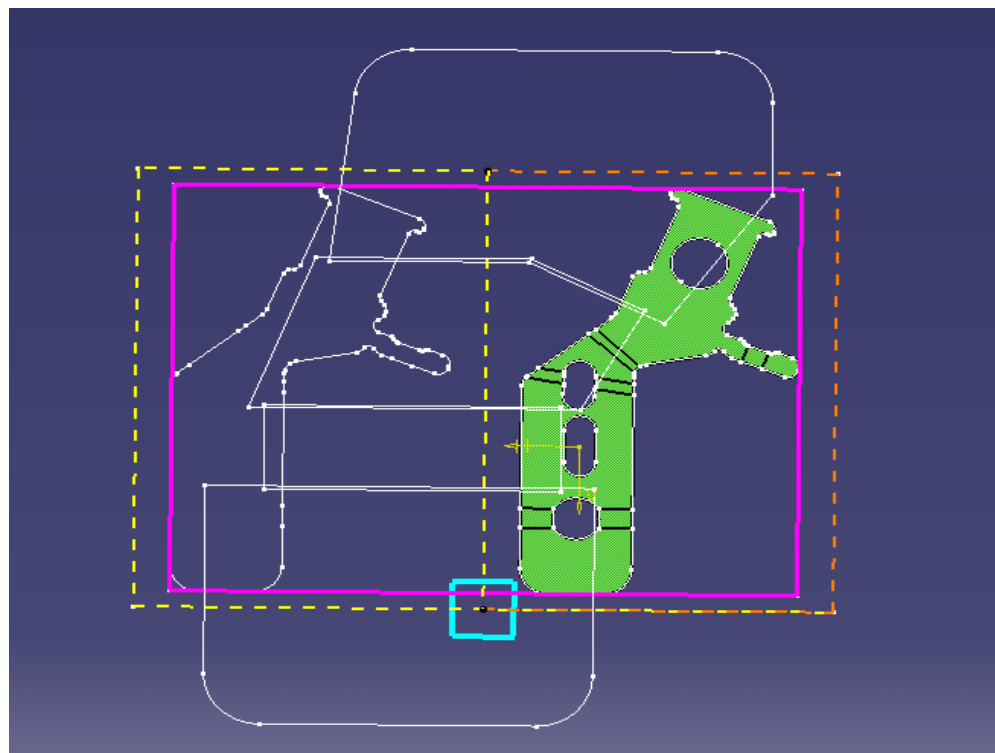
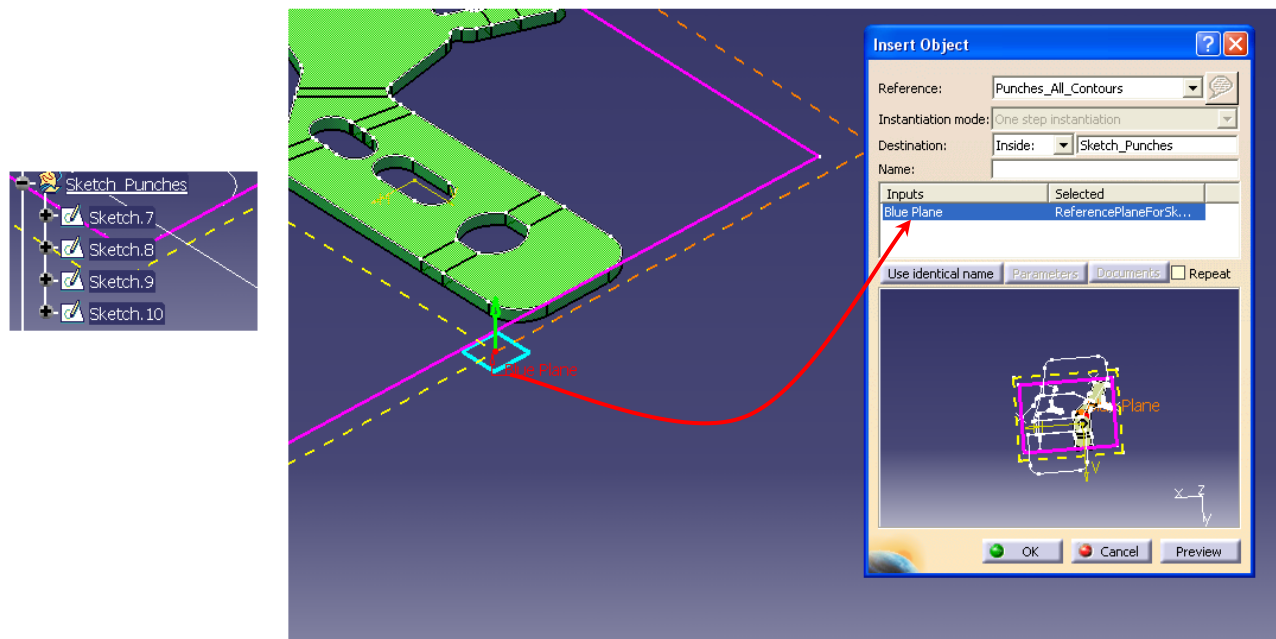
- ✎ Open the “...\Support\_Punches\_Contours.CATPart” file

↳ The “Insert Object” window is displayed

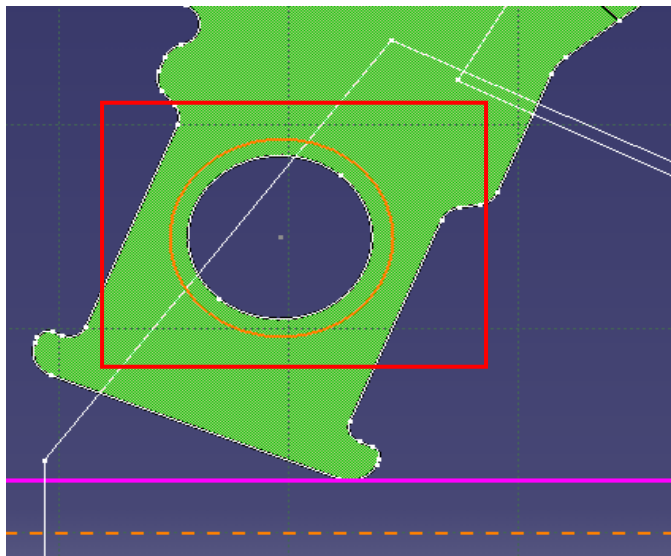


## Creation of punch sketches (2 / 2)

-  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.
- The sketch is directly created under the Sketch Punches Geometric Set.



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.

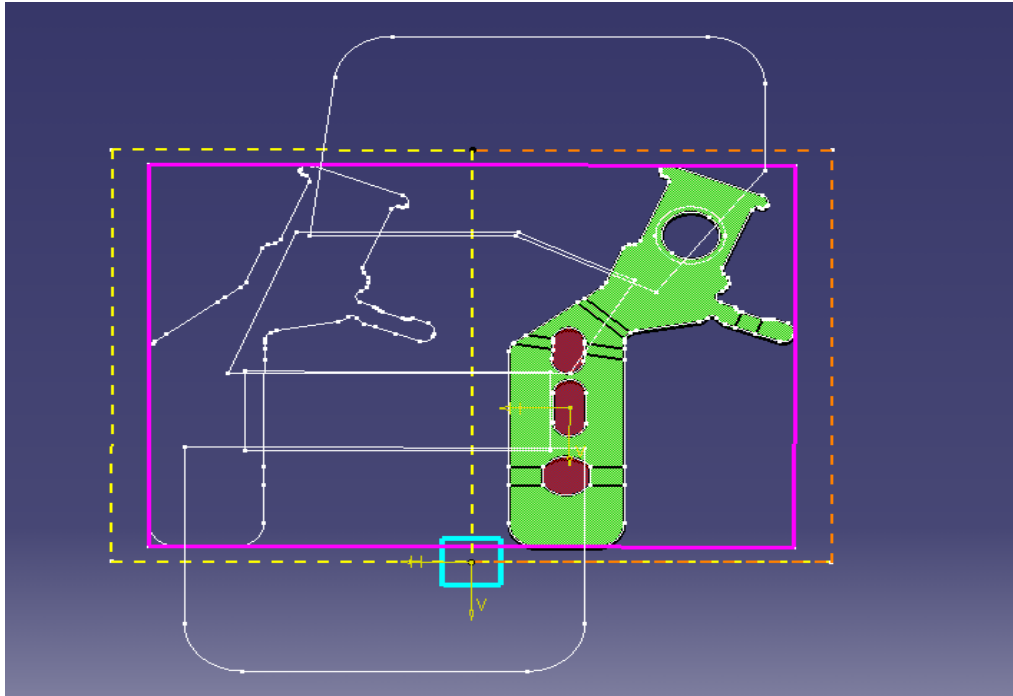
## ***Show Isolated area***



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

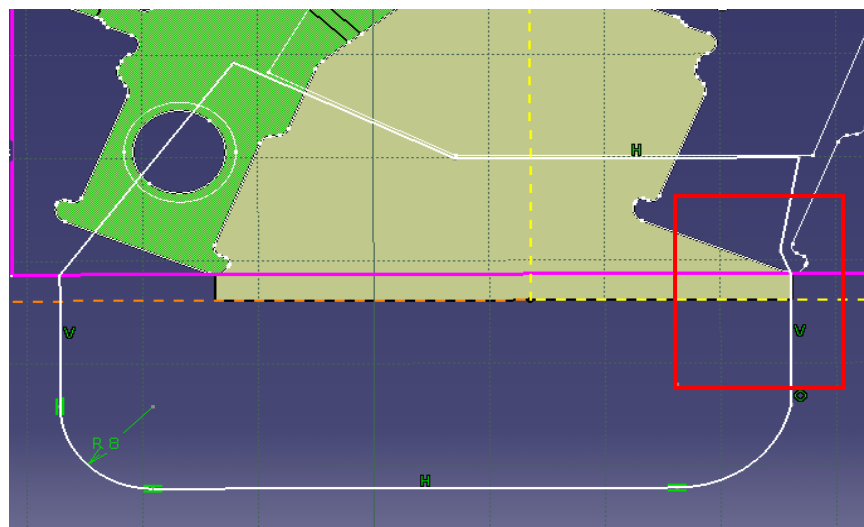


## Show Scrap Area



This Command shows the external scrap Area to the user. This scrap Area is created during Strip Definition Command. While Defining the Punch Contours it has to be ensured that the whole of External Scrap Area is removed by the punches. This is extremely necessary for the correct design of the Strip. If whole of the scrap Area is not removed by Punches or Lead Frames then it will affect the End of Strip Command. In the picture below you can see that whole of the scrap area is not covered by punch sketches. If strip is designed using these punch contours the punches will not remove the total scrap area and the resultant strip is not correct. For solving this problem:

-  Double click on the top Sketch (Edit the Top Sketch).
-  Edit the Top Sketch in such a way that the whole of scrap Area is covered by all of Punch sketches.
-  This will ensure that total Scrap area will be removed by Punches.



 Save the Product after Sketches are done.



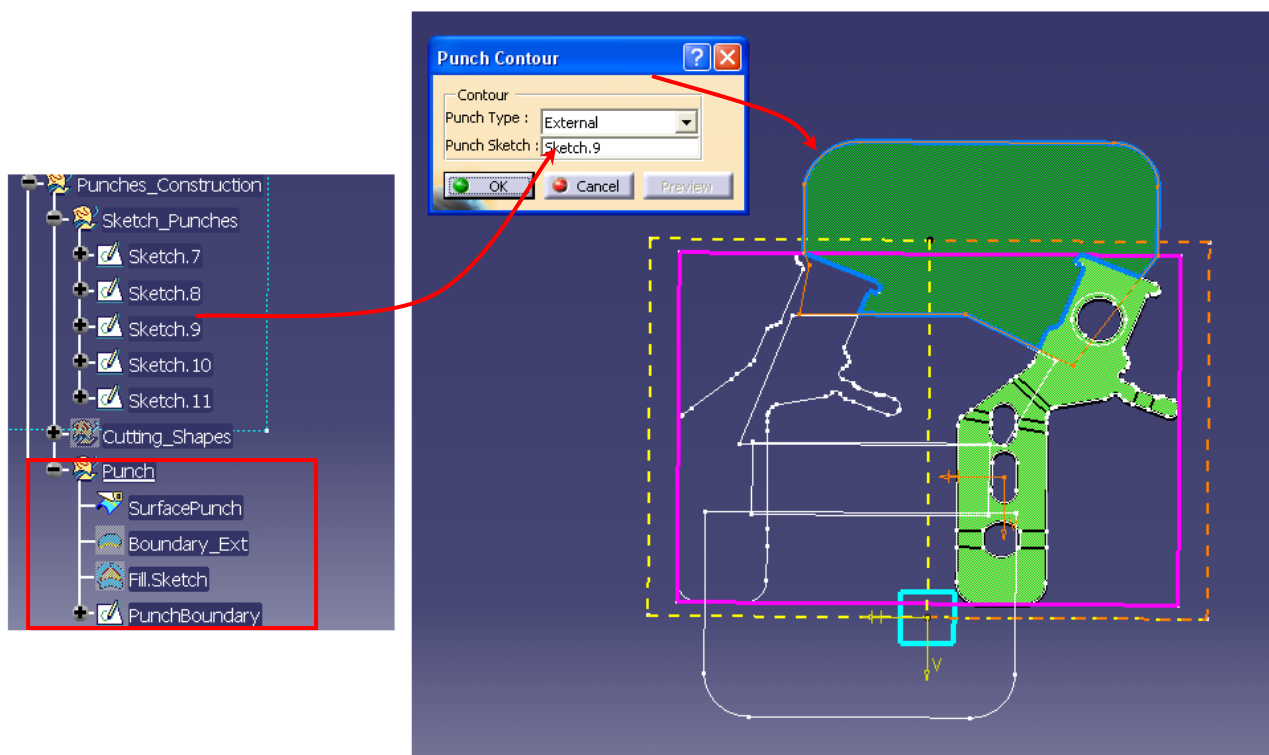
## Definition of the punch contours (1 / 4)

### Mandatory:

- ⚙ Define the “Sketch\_Punches” open body as in work object to create the sketches in it.
- ⚙ In Order to obtain a correct result for End of Strip it has to be ensured that the Punches and Lead Frame remove the whole of scrap Area which can be seen through the Show Scrap Area Function.
- ⚙ Punches and Lead Frames cannot be created for sketches with Multiple Domains.
- ⚙ Multiple Lead Frames can be defined.

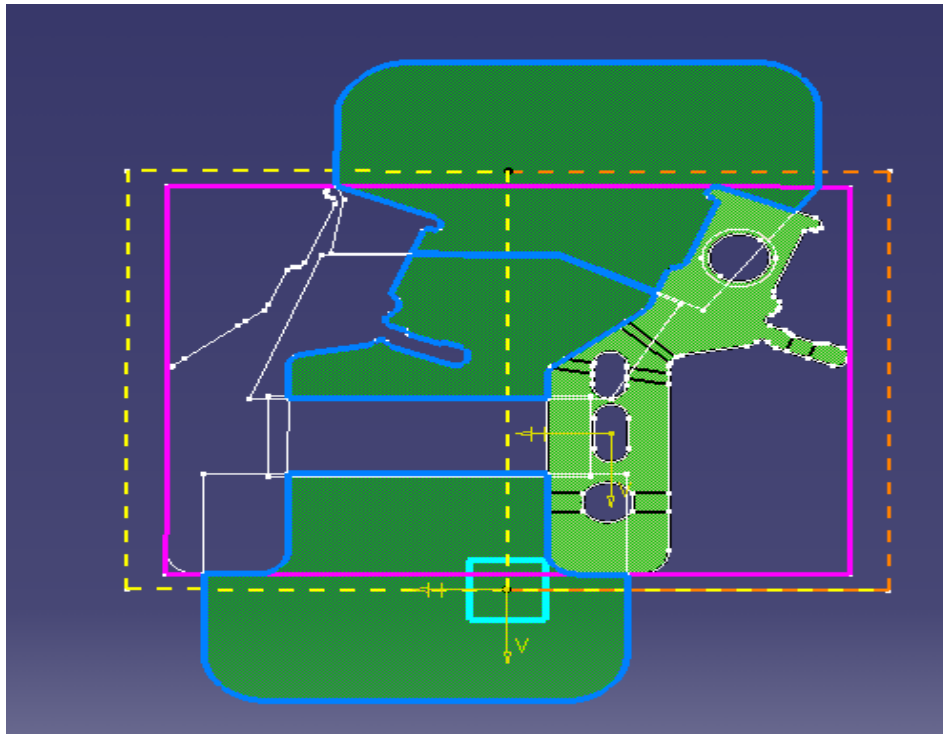
### Create the external Punch Contours:

- ⚙ 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 Lead Frames (3 / 4)

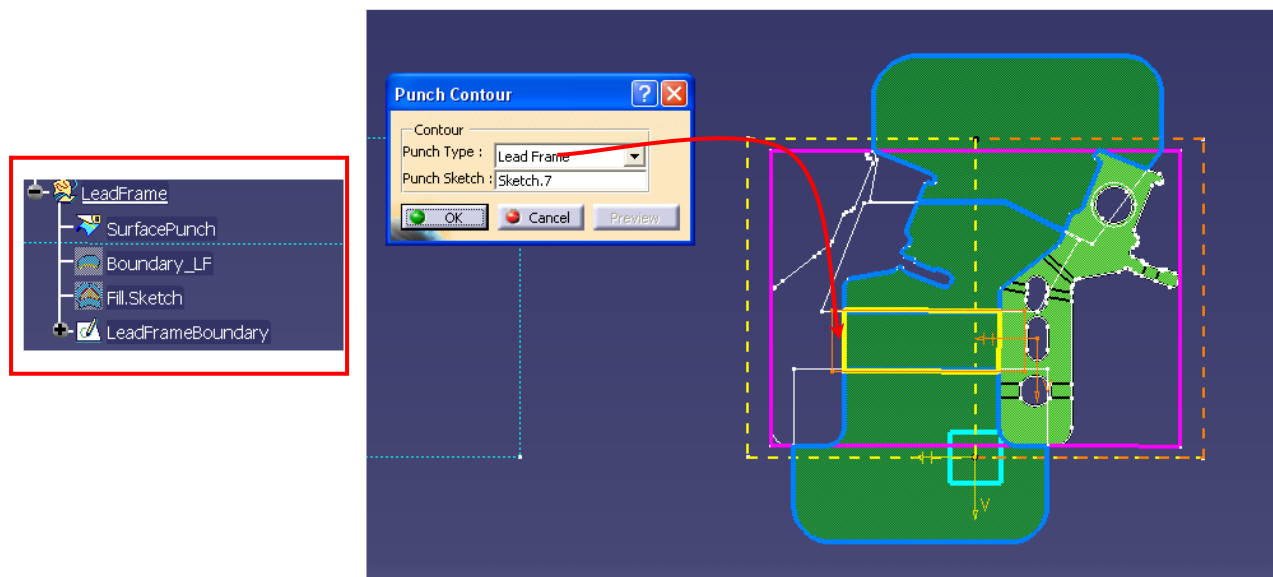


This is not a Mandatory Step. This is dependent on the designer, Instead of Lead Frame you can create External Punch contour also.

### ✎ Create the Lead Frame Contours:

- ✎ Select the **Punch contours** function.
- ✎ Select the type (Lead Frame).
- ✎ 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
- ✎ Lead Frame Boundary is created with Yellow Color, In order to distinguish between Punch Contour and Lead Frame Contour.



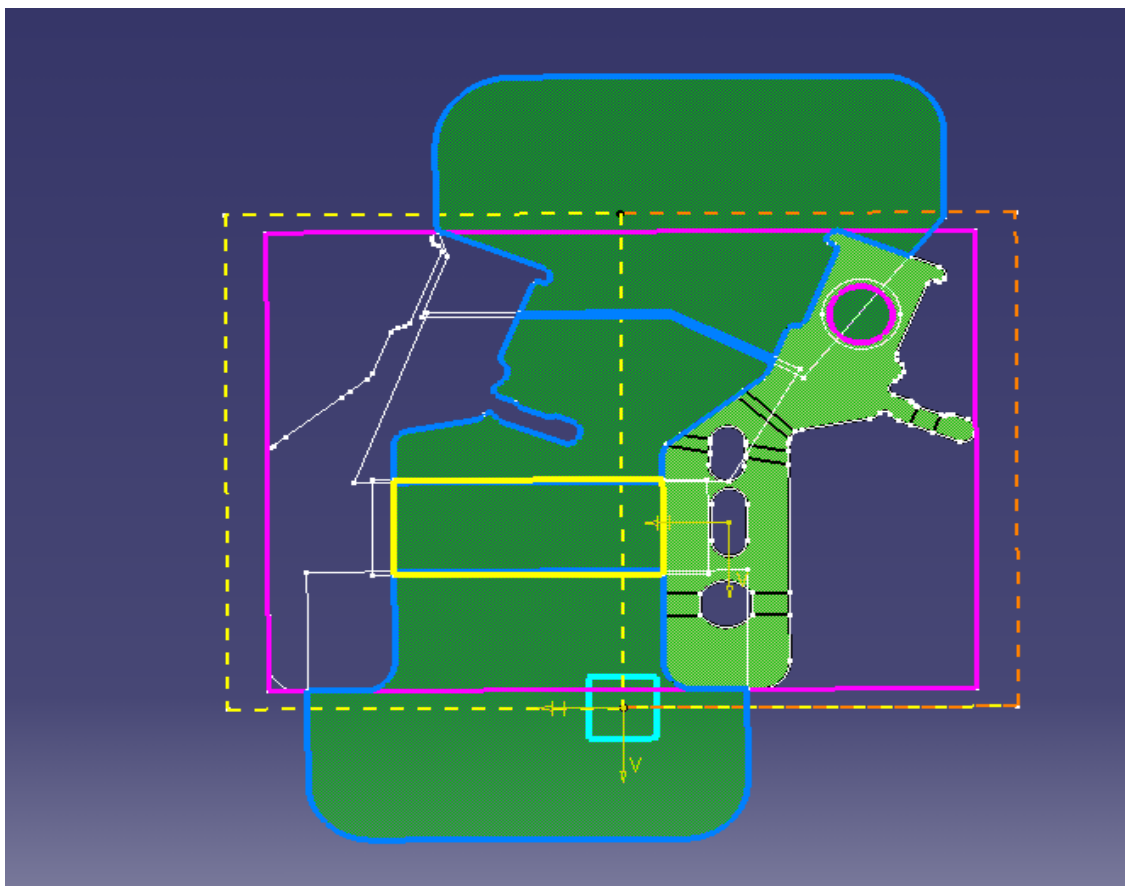
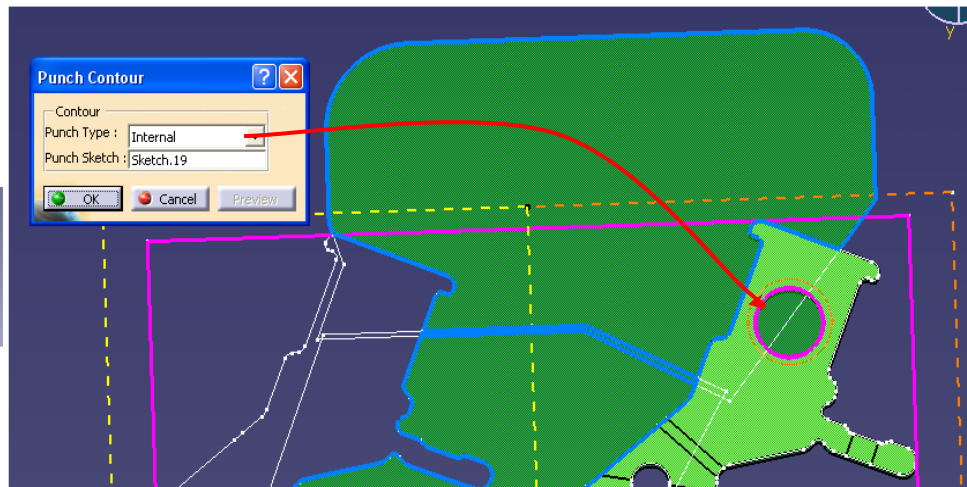


## Definition of the punch contours (3 / 4)



### 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.
- 👤 Internal Punch Contour is created with Pink Color, In order to distinguish between External Punch Contours and Lead Frame Contours and Internal Punch Contours.
- 👤 Validate with the **OK** button



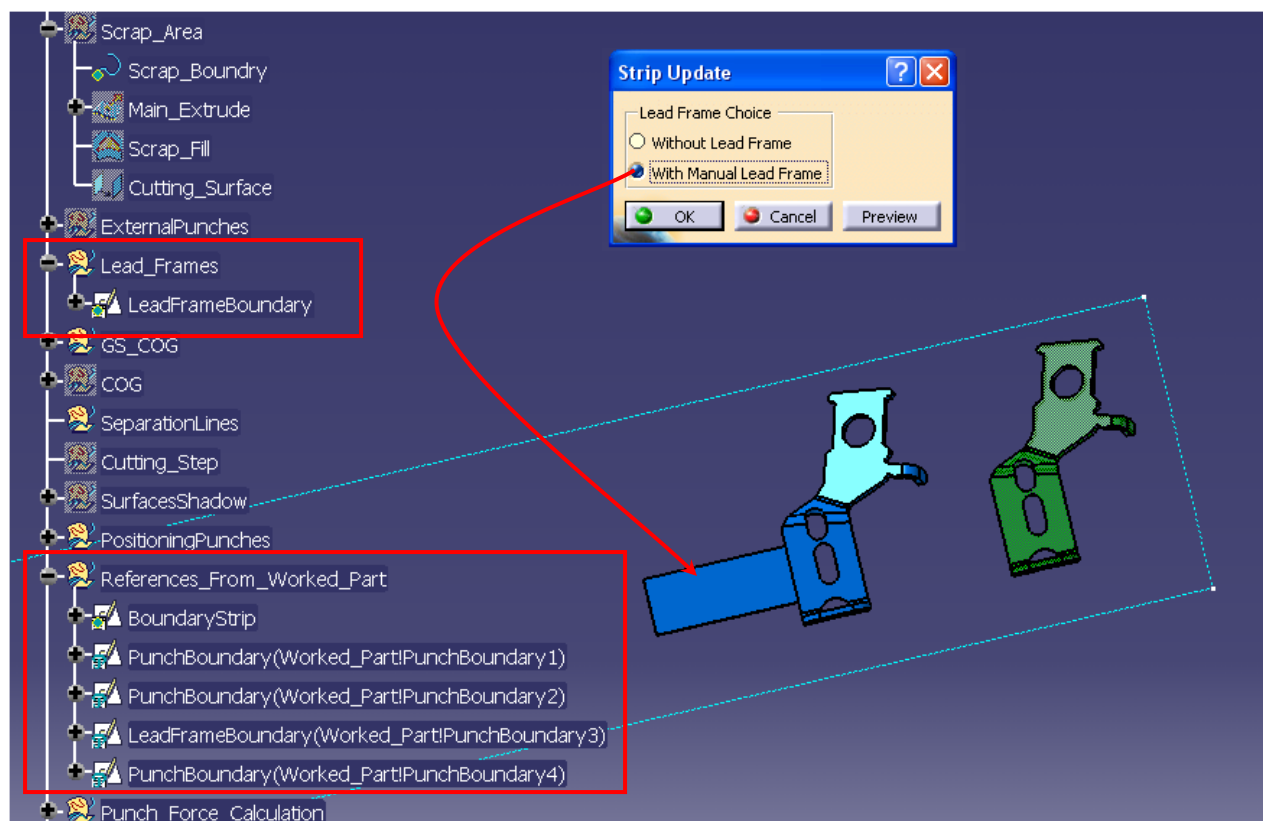
## Strip Update

### Click on Strip Update.

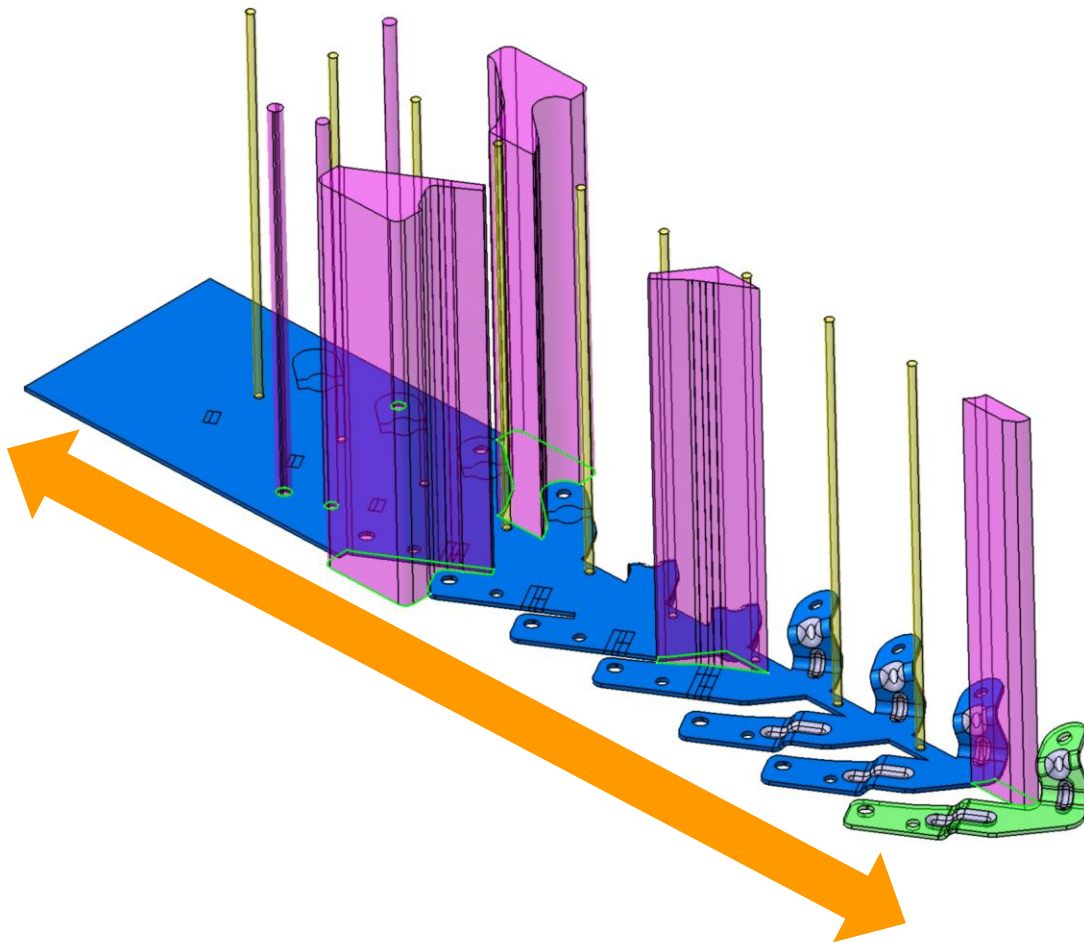
- ✚ Select With Manual Lead Frame (If Lead Frame is designed) or Select Without Lead Frame.
- ✚ All the Punch Contours and Lead frame Contours are copy pasted from Worked Part to Strip Construction part in this function and first two stations are created in this command.
- ✚ Once Strip Update is done it can only be used to update the punch Contours and Stations are not recreated

### Mandatory:

- ✚ In cases of Lead Frames Strip Update with Lead Frames is done only once.
- ✚ Hence it has to be ensured that all the Lead Frames are designed before running this command.
- ✚ Once strip Update is done it can only be used to update the punch contours.



# Strip Layout Design

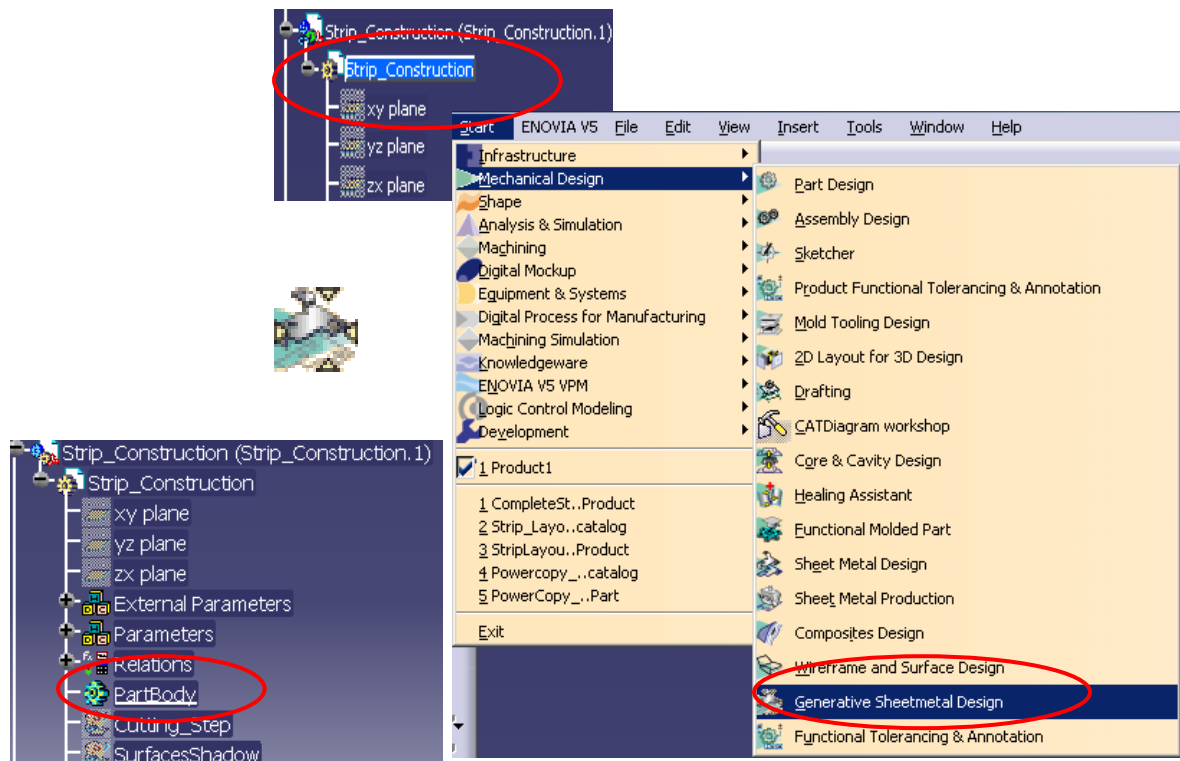


## Introduction

- 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 through the dedicated functions developed for the Strip Layout methodology.  
Of course it is possible to define a different strip

- Edit the “Strip\_Construction” part by double-clicking on it:  
 The *Strip\_Construction* part is highlighted in blue.
- Verify to be in “Generative Sheet metal 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 second step: Unfolding Step (1 / 2)

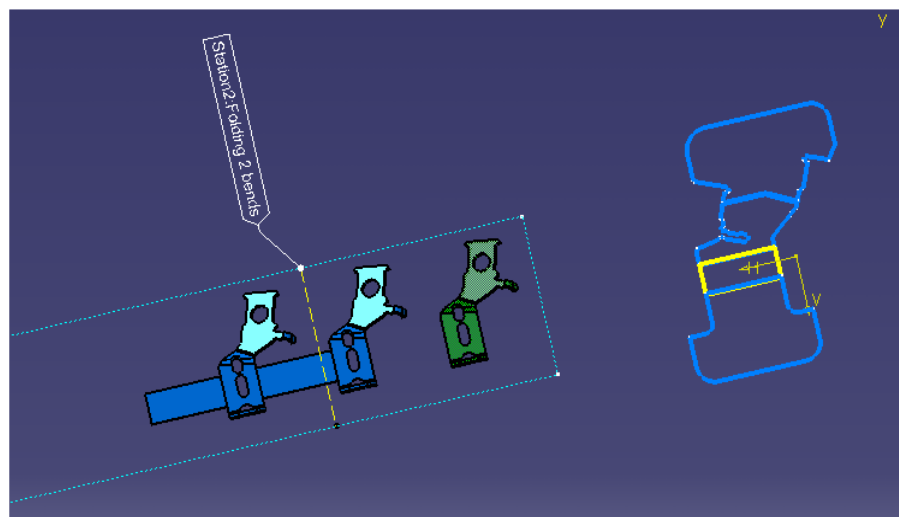
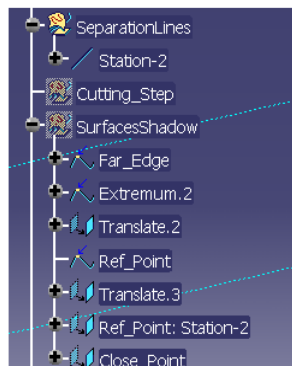
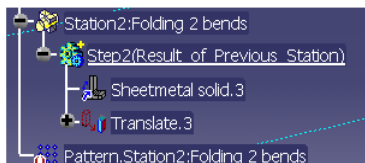


The First Step is already created in the Strip Update. As we are not doing any operation on the first Step we proceed with creation of next Step.

- **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.
  - 👤 A separation Line is created in the Separation Lines GS in the Strip Construction Part. If user doesn't want to see it user can hide the Separation Lines GS.

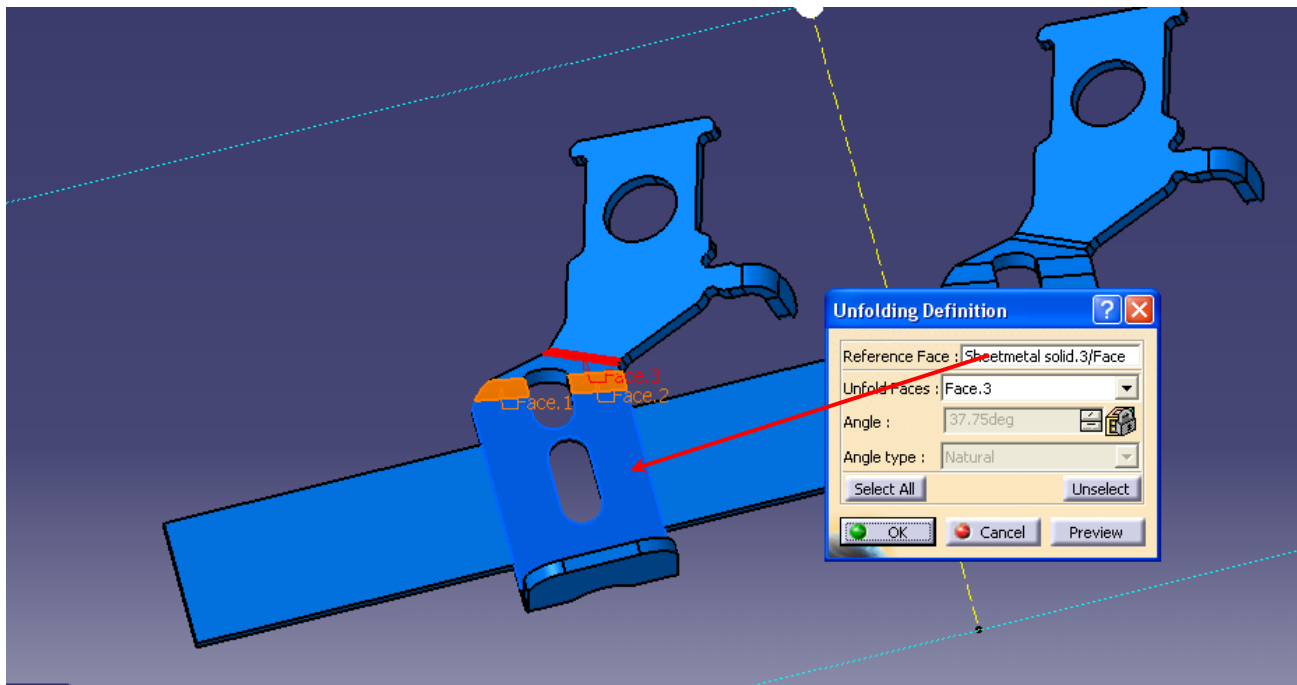




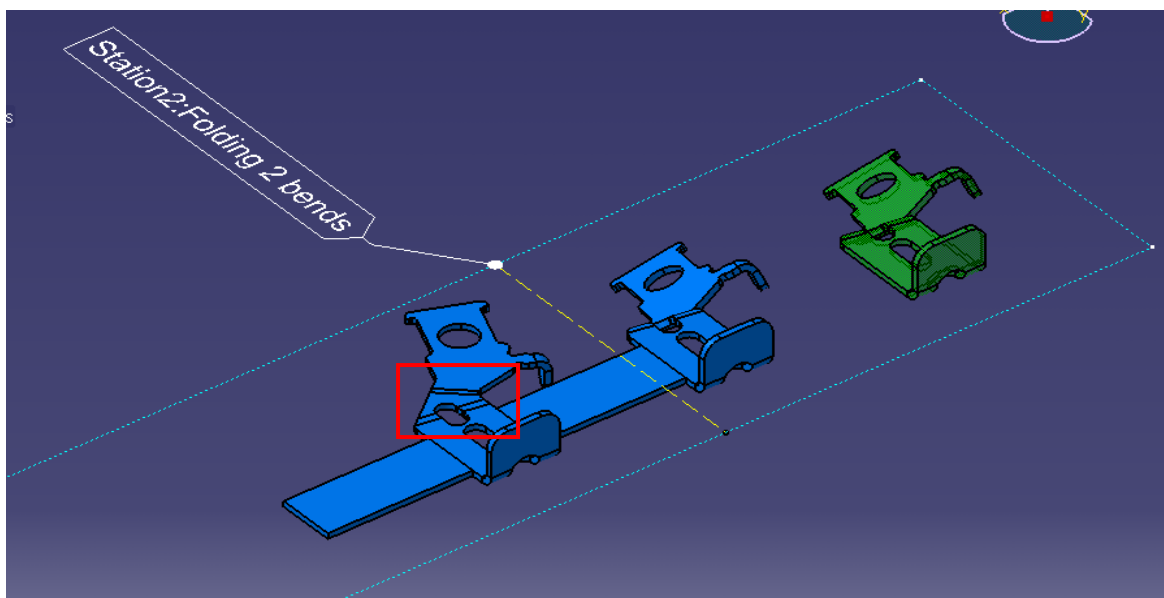
## Creation of the second step: Unfolding two bends (2 / 2)

### 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)








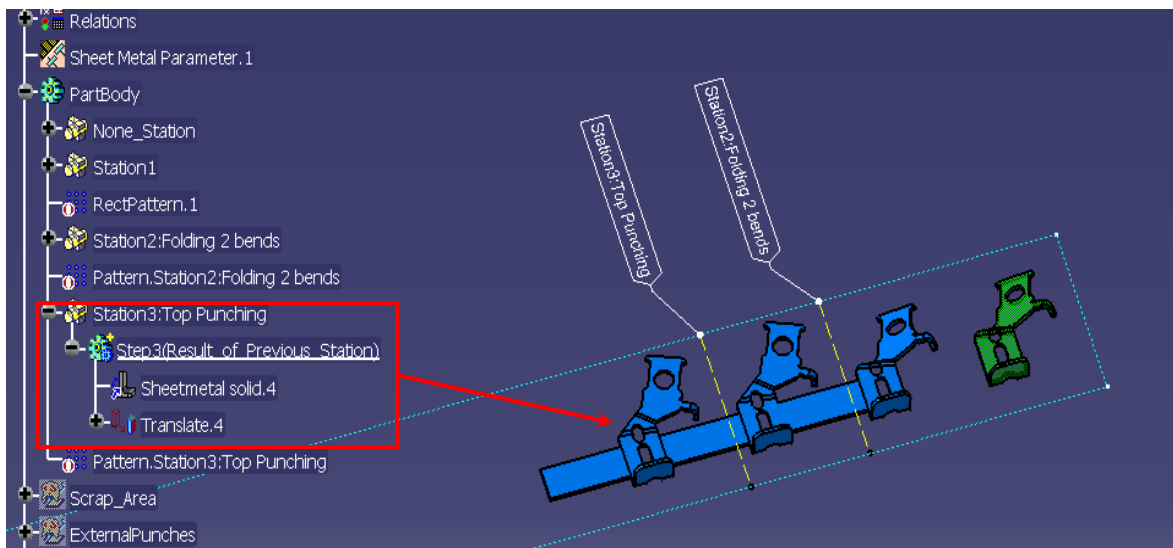
-  Select the **OK** button to validate
  -  The bend is unfolded



## Creation of the third step: A punching step (1/2)



### 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.: "Top punching")
-  Validate with the **OK** button.
  -  The "Station3: Top punching" is created in the "PartBody" body.
  -  A parameter "Station3\_FreeStep" has been created and set to 0.

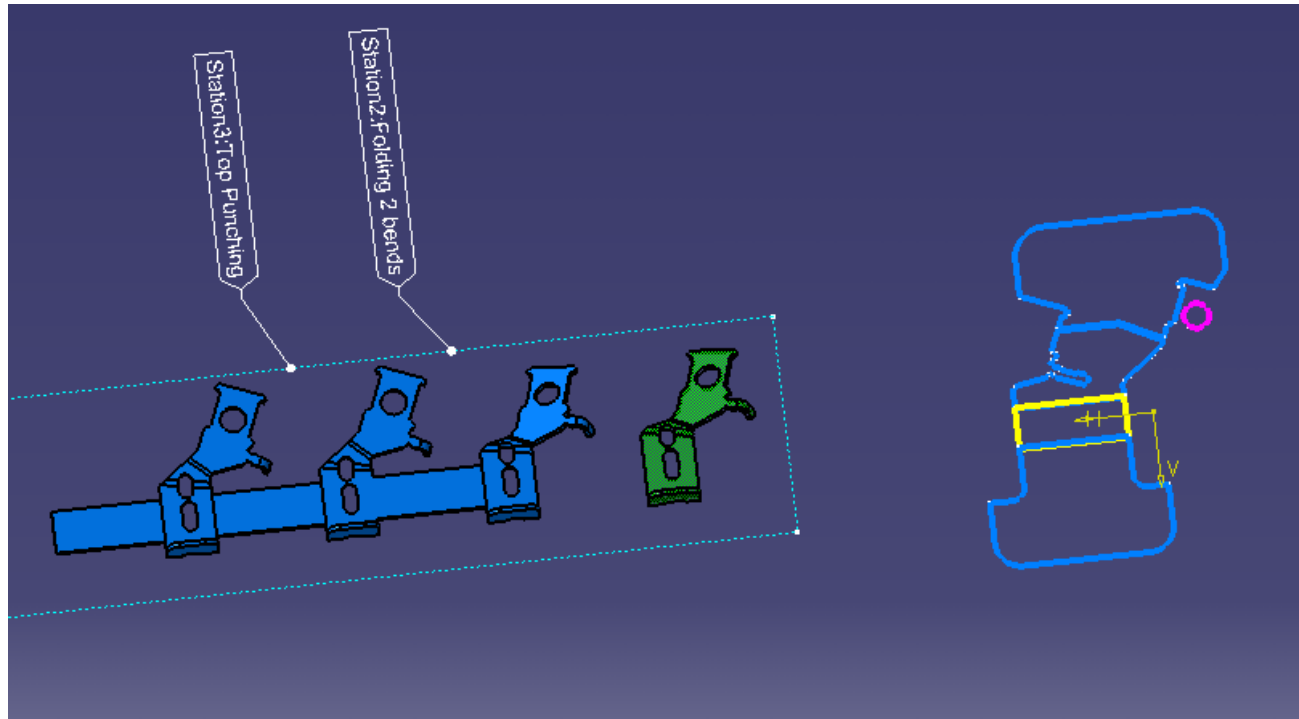


## Creation of the third step: A punching step (2/2)

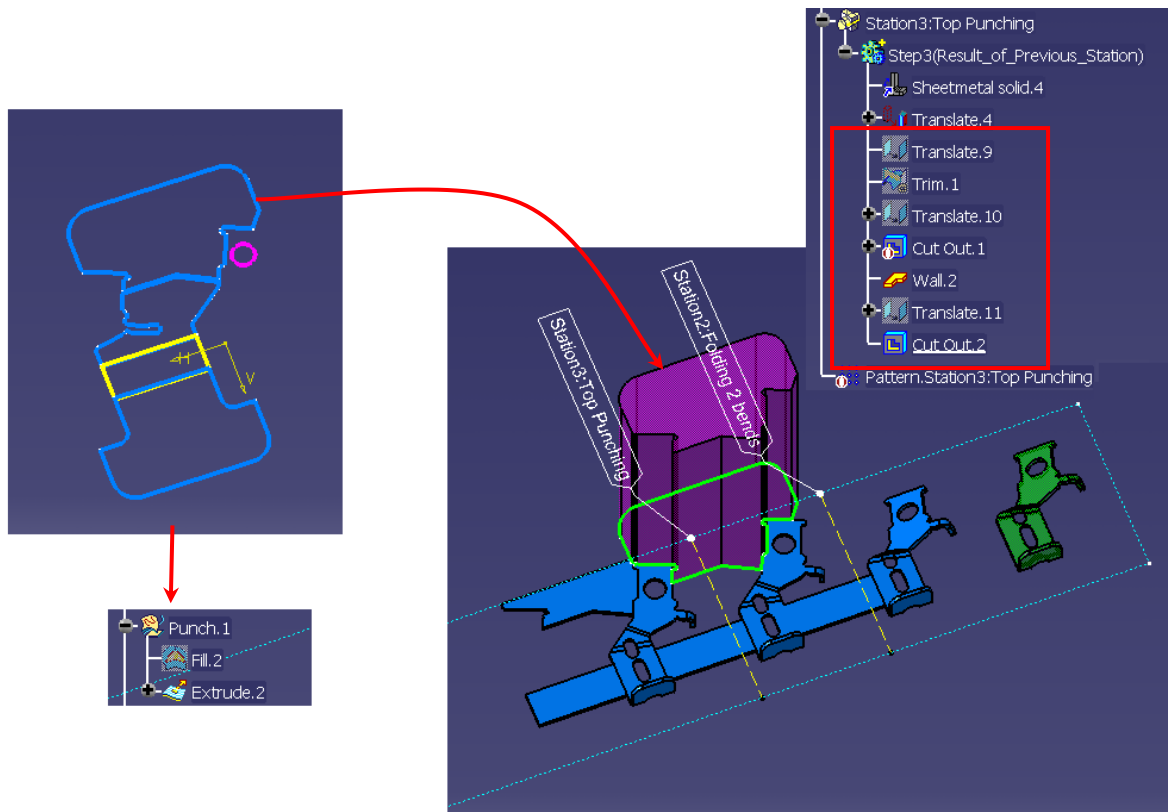
### Create the punch tool and the material removed by the tool:

-  Use the Cutting Punch function by selecting the dedicated icon.
-  The "Create Cutting Punch" window will be displayed in top left corner of the screen.





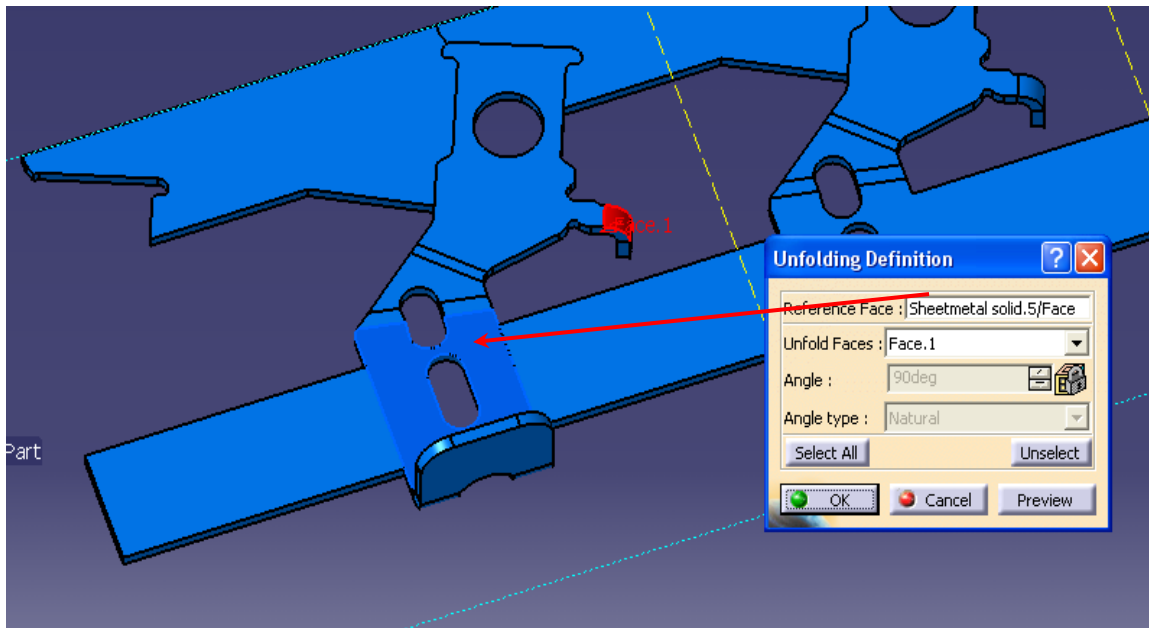
- 👤 Select the “Station3 – Top Punching” in the combo box in dialog.
- 👤 Select the right top **blue punch contour** that will instantiate the punch to separate the finished part from the strip
  - 👤 *The material removed by the tool has been added, the punch tool is displayed in purple and these new features have been created in “Punch1” and “PositionningPunches” open bodies and in the Station As well.*











## Creation of the fourth step: A folding step

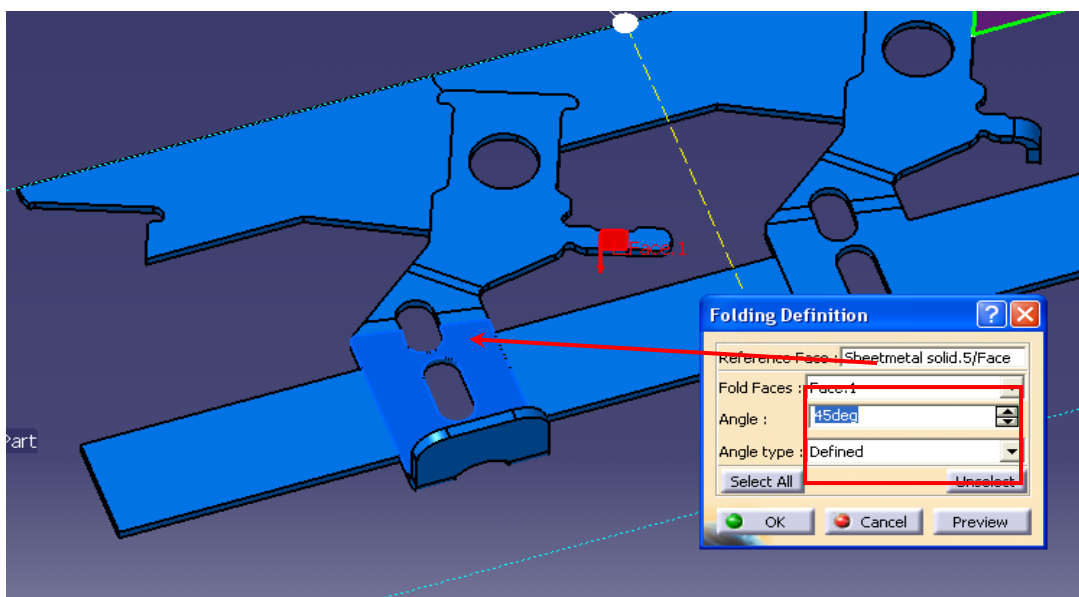


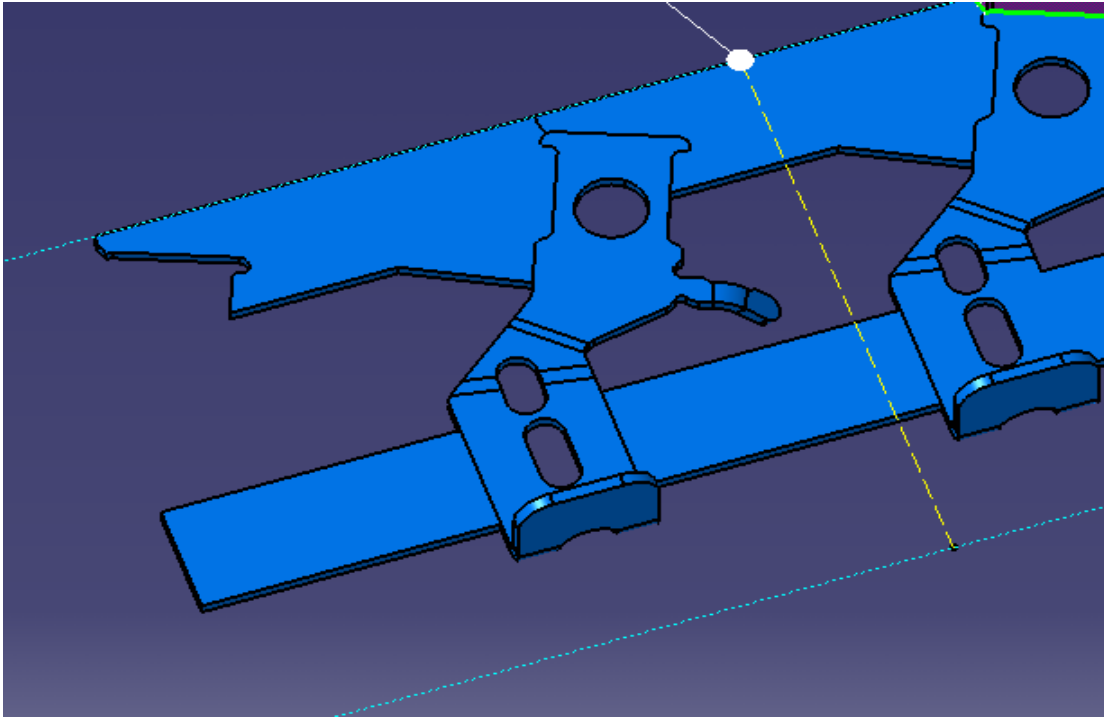
- **Add the step which will receive the folding operation:**
  - 👤 Select the **Add Step** function by selecting the dedicated icon and follow the same process as in the previous steps.
  - 👤 Enter the name of the station (i.e.: "Central Folding 1")
  - 👤 Validate with the **OK** button.
    - 👤 The "Station4: Central Folding 1" is created in the "PartBody" body.
    - 👤 A parameter "Station4\_FreeStep" has been created and set to 0.
- **Unfold the Central 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.



#### Refold the Central bend:

-  Select the **Folding** icon (  )
  -  The “Folding 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 **Fold Faces** (either the internal or the external face of the bend can be chosen)
  -  Select the mode as **Defined Mode** and valuate the Folding angle to 45 degrees.
-  Select the **OK** button to validate
  -  The bend is folded.






## Creation of the fifth step: A folding step

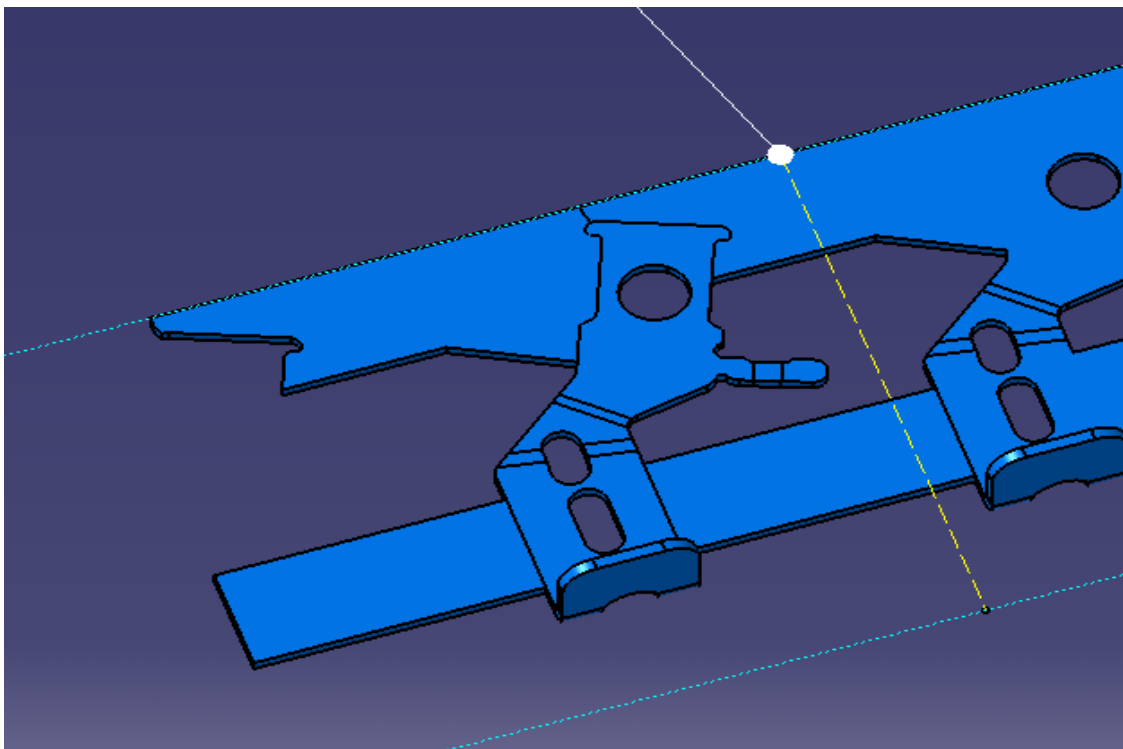
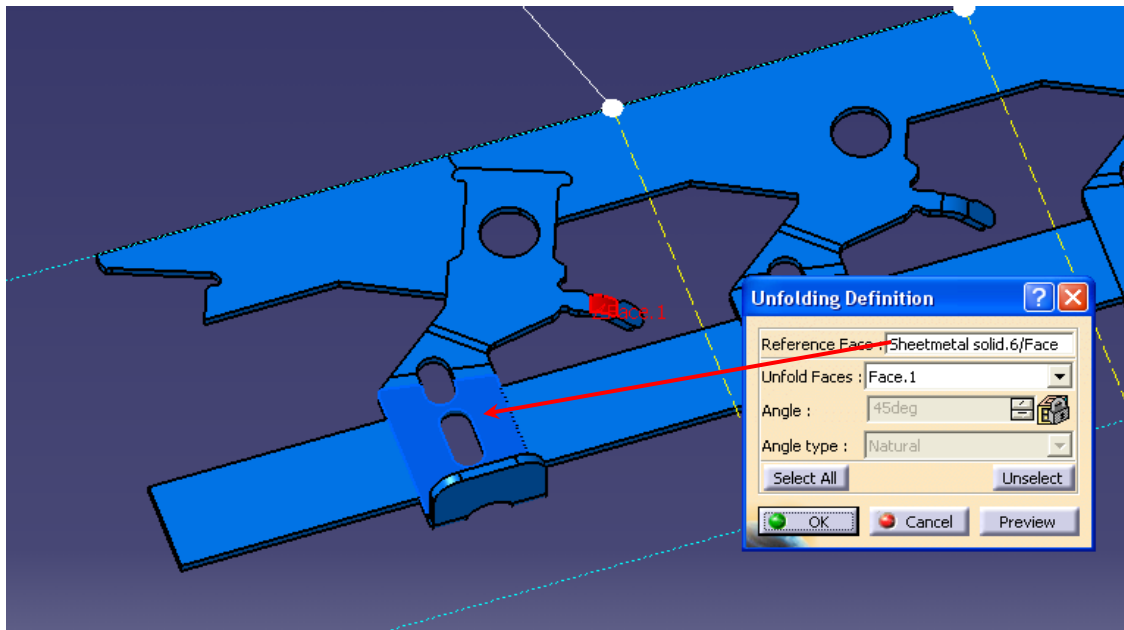


### ➤ Add the step which will receive the folding operation:










- 👤 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.: "Central Folding 2")
- 👤 Validate with the **OK** button.
  - 🔗 The "Station5: Central Folding 2" is created in the "PartBody" body.
  - 🔗 A parameter "Station5\_FreeStep" has been created and set to 0.

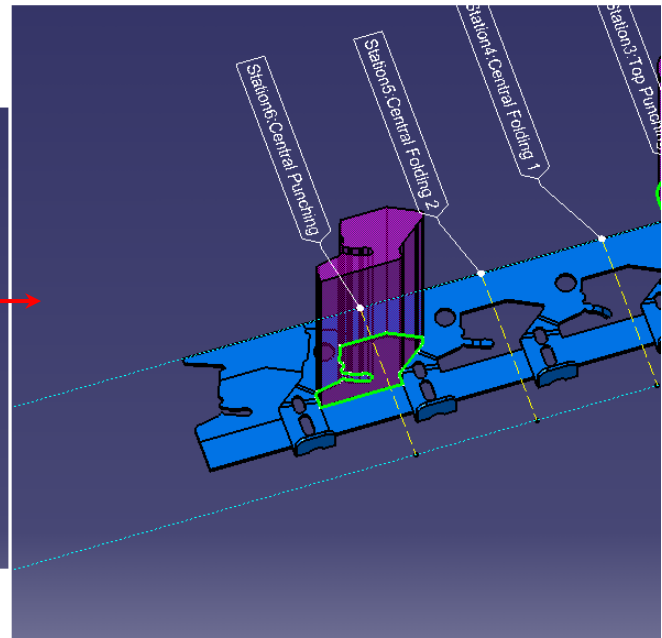
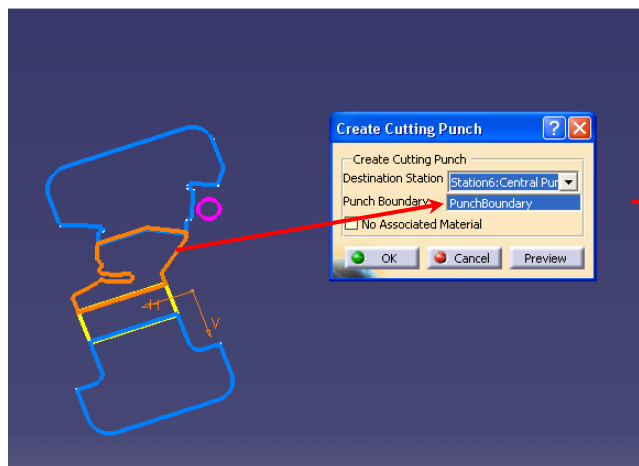
### ➤ Unfold the Central bend:

- 👤 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 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 "Station6: Central punching" is created in the "PartBody" body.*
    -  *A parameter "Station6\_FreeStep" has been created and set to 0.*
- **Create the punch tool and the material removed by the tool:**
  -  Select the **Create Cutting Punch** function by selecting the dedicated icon.
  -  Select in the tree, the **Station6: 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 "PositioningPunches" open bodies and in the Station 6 Assemble.*






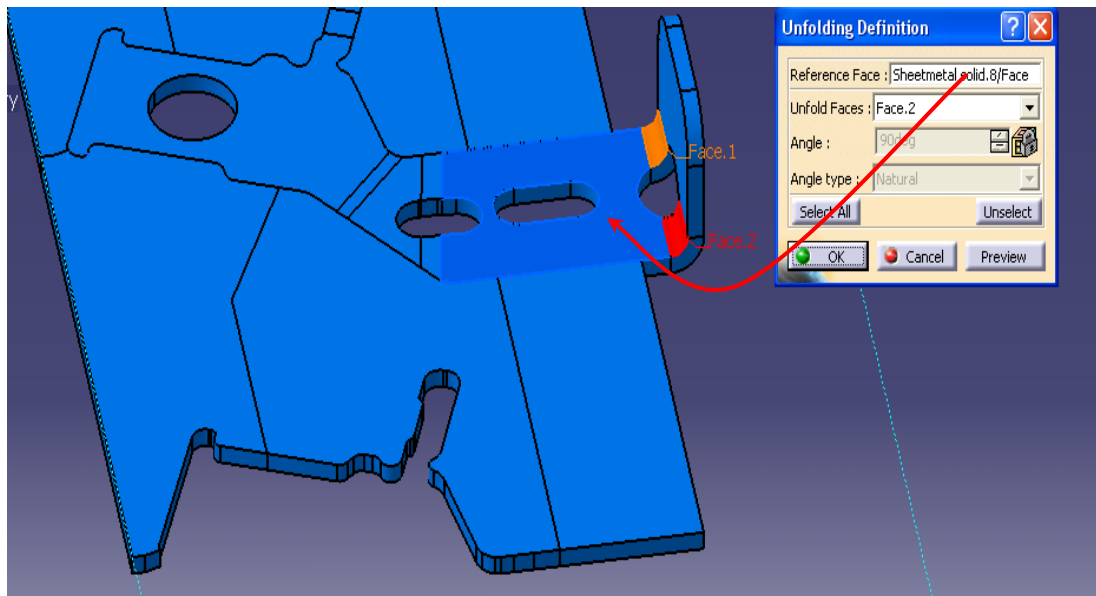
## Creation of the seventh step: A folding step

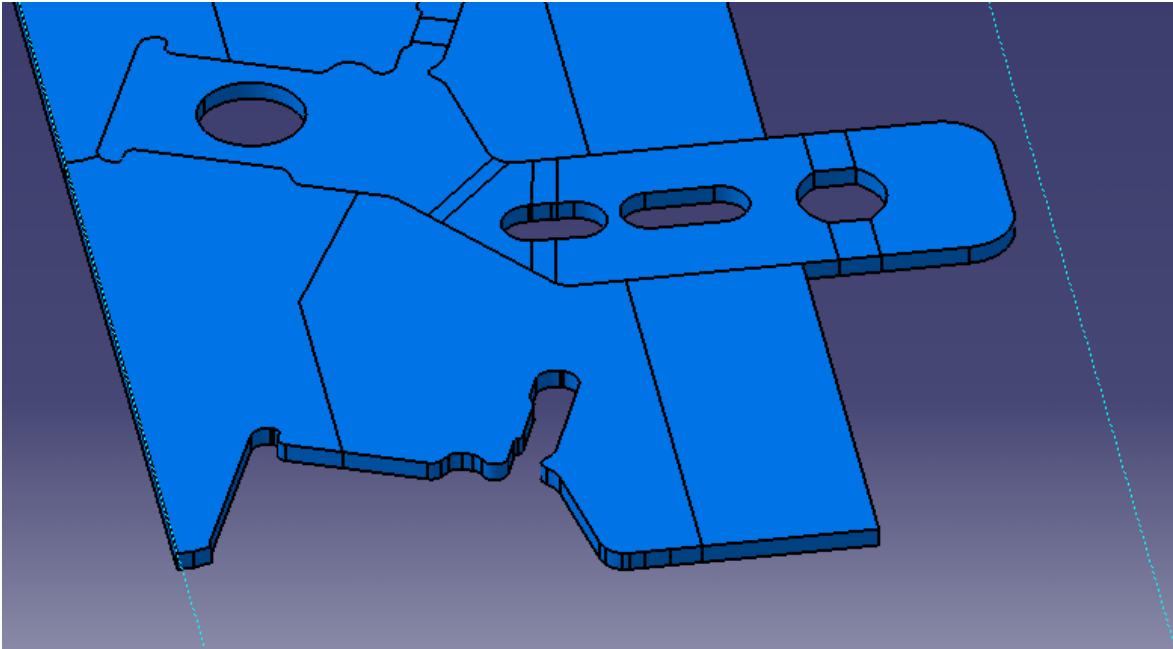
### ➤ Add the step which will receive the folding operation:

- ✎ 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.: "Bottom Folding ")
- ✎ Validate with the **OK** button.
  - ✎ The "Station7: Bottom folding 2" is created in the "PartBody" body.
  - ✎ A parameter "Station7\_FreeStep" has been created and set to 0.

### ➤ Unfold the Bottom 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.



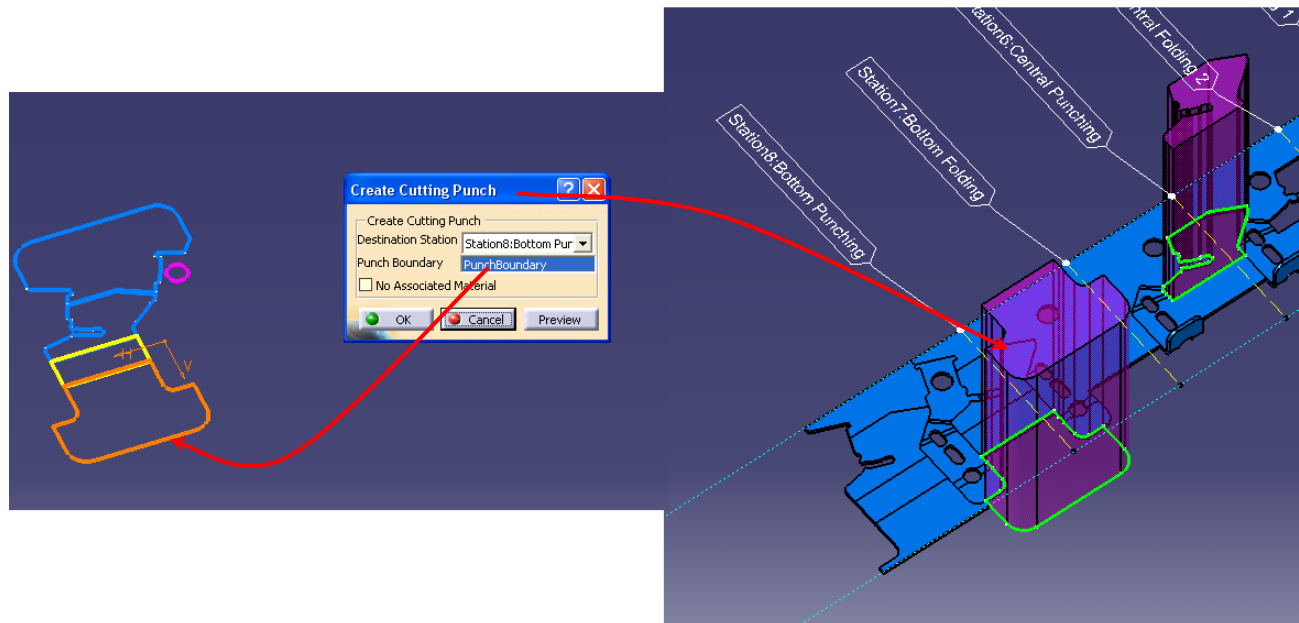


## Creation of the eighth step: A Punching step



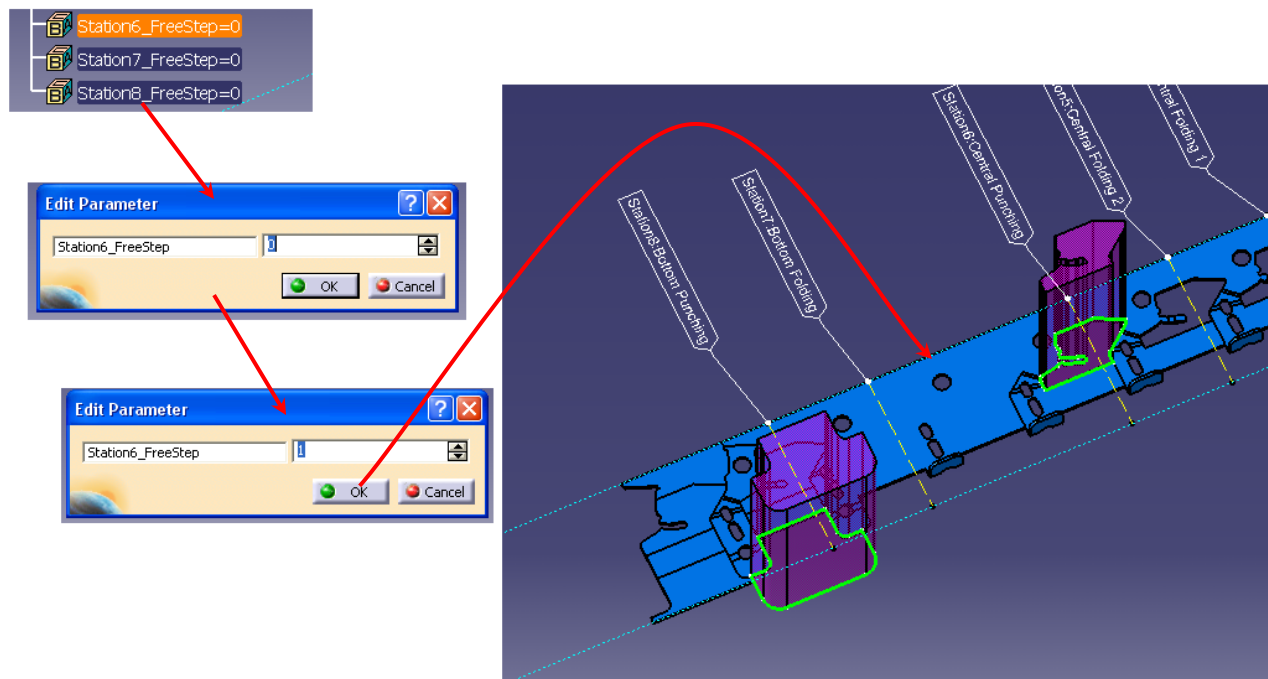
- **Add the step which will receive the punching operation:**
  -  Select the **Add Step** function by selecting the dedicated icon and follow the same process as in the previous steps.
  -  Enter the name of the station (i.e.: "Bottom punching")
  -  Validate with the **OK** button.
    -  *The "Station8: Bottom punching" 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 **Create Cutting Punch** function by selecting the dedicated icon.
  -  Select in the tree, the **Station8: Bottom 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 Station 8 Assemble.*





## Creation of free steps

- To prevent the collision between folding and cutting punches, we will add free steps between station 6 and station 7.
  - 👤 Select the **Station6\_FreeStep** parameter of the "Strip\_Construction" parameters
    - 👉 The "Edit Parameter" window is displayed.
  - 👤 Change the value from "0" to 1.
    - 👉 The "Station6\_FreeStep" parameter is now equal to 1.
    - 👉 The "Pattern.Station6" 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 Managing Free Steps.
  - 👤 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.
    - 👉



## Insertion of a new step for Punching between Existing Steps (1/2)

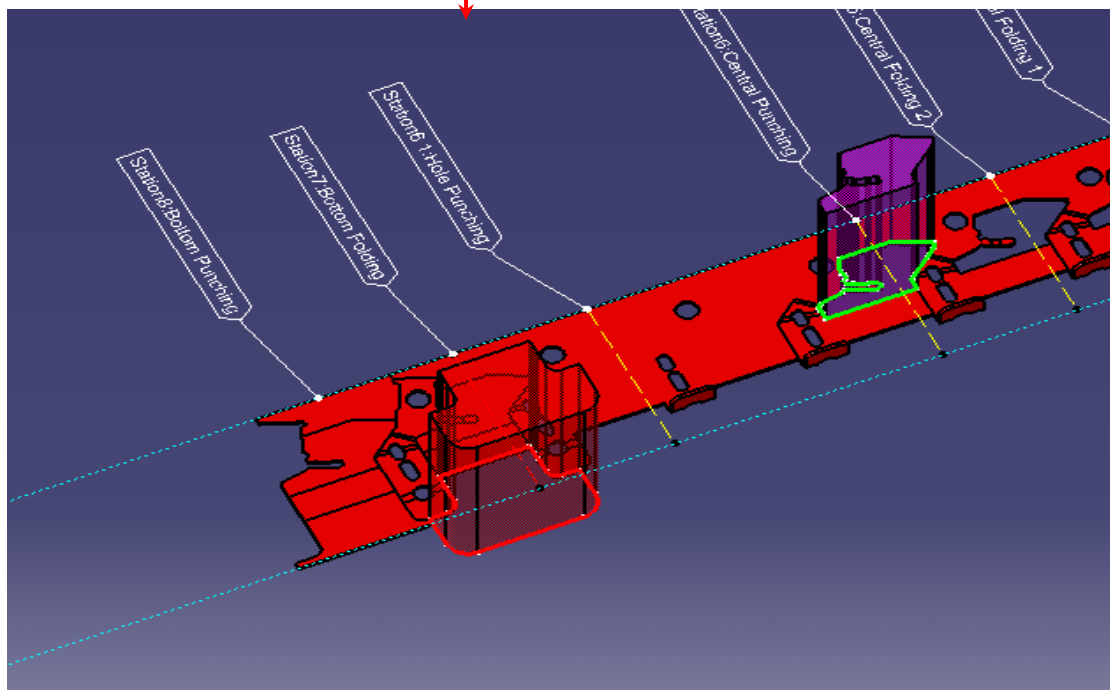
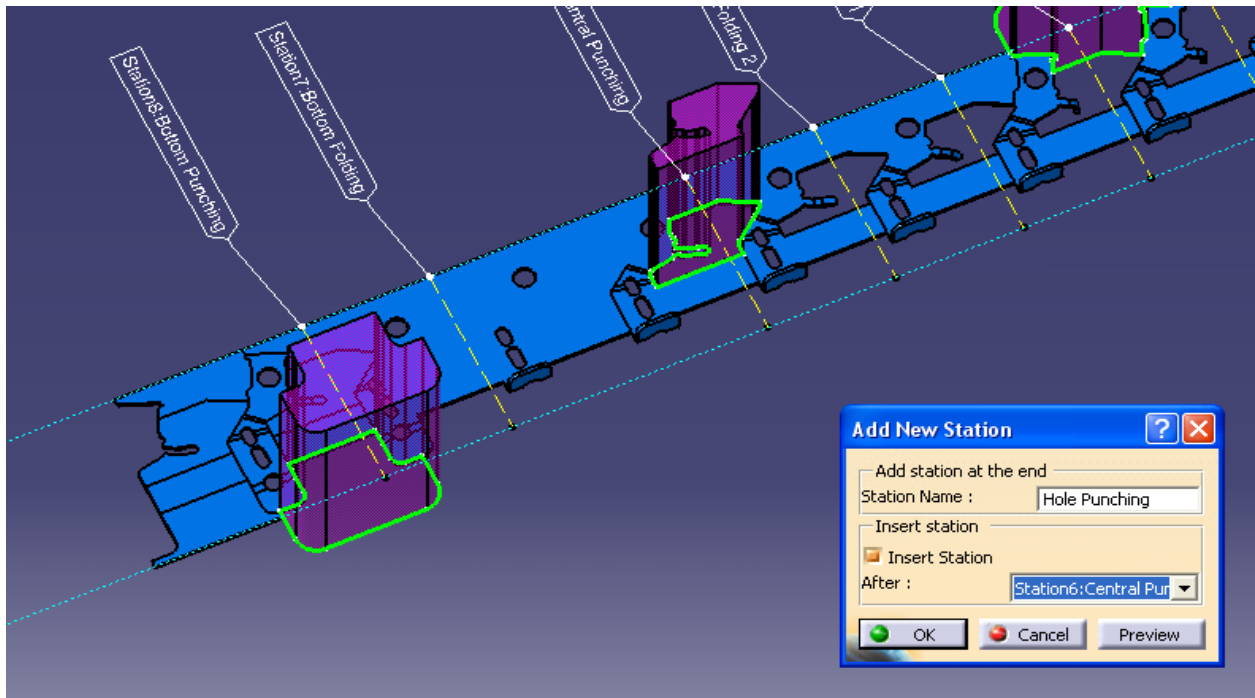


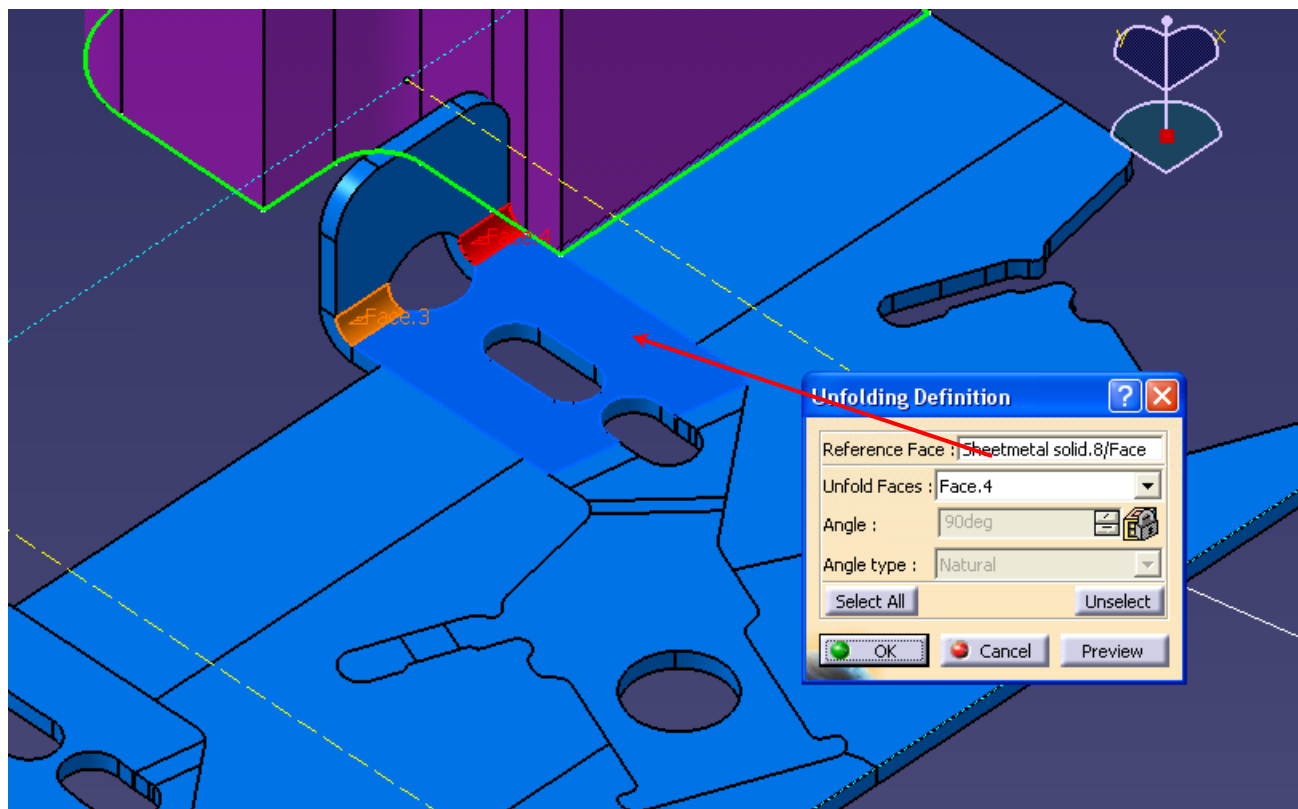
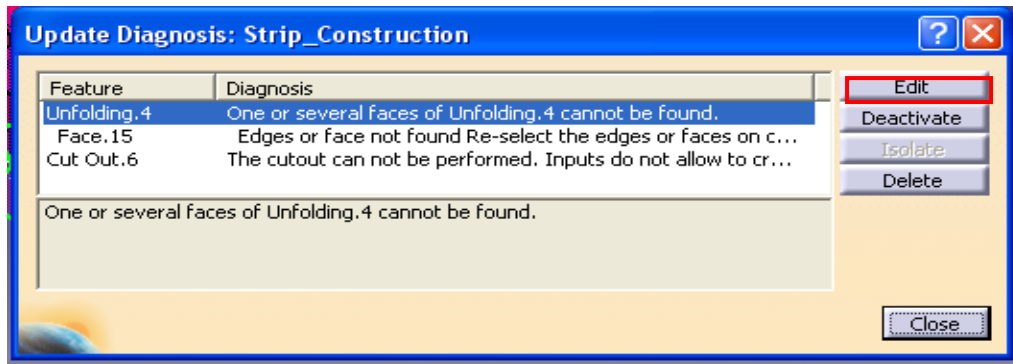
### ➤ Add the step which will receive the punching operation:

- ✎ Select the **Add Step** function by selecting the dedicated icon and select the Insert Station Checkbox.
- ✎ In the Selection Combo Box select Station6: Bottom Folding.
- ✎ Enter the name of the station (i.e.: "Hole punching")
- ✎ Validate with the **OK** button.
  - ✎ The Part is not updated in the case of Insert Station. This is due to the Unfolding feature. If there is an unfolding feature after the station which is being inserted the unfolding fails. In that case user has to perform following Steps:
    - ✎ A Update Diagnosis Dialog Box will pop up indicating that unfolding has failed.
    - ✎ Press on Edit, The Station which should have unfolding is defined in work object and Unfolding Dialog is launched.
    - ✎ User has to give the inputs to the unfolding feature once again and after giving the inputs to unfolding Insert Station is successful.
    - ✎ If there is no unfolding after the Inserted Station then the Update will be successful and there wont be any problems.
    - ✎ Please Refer the Pictures below.

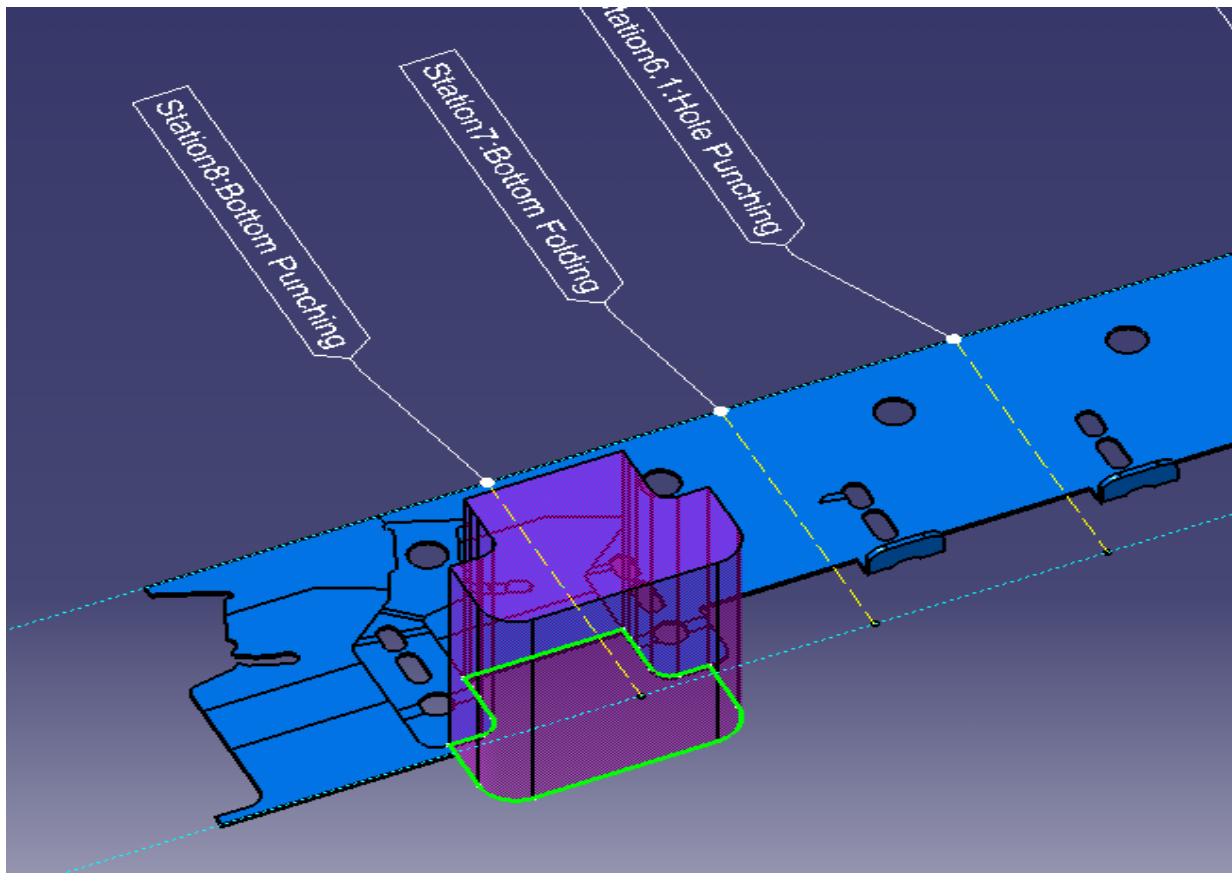
### ➤ Mandatory:

- ✎ **After Insertion of Station in Between User will have to do a Manual Update Always.**
- ✎ **If there is an unfolding feature after the station which is inserted Update will give a dialog box of Update diagnosis showing Unfolding Feature has failed, and User will have to edit it for a successful update.**










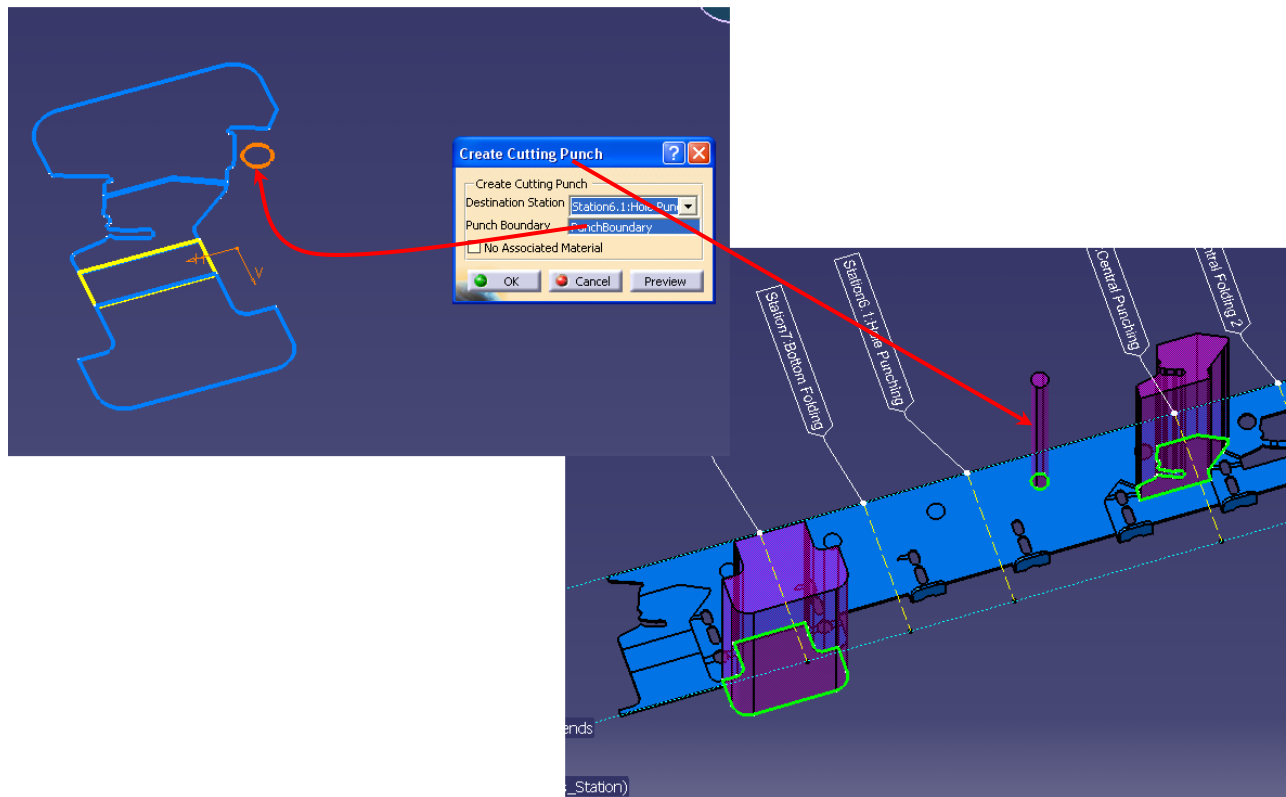
- 🔧 The “Station6.1: Hole punching” is created in the “PartBody” body.
- 🔧 A parameter “Station6.1\_FreeStep” has been created and set to 0.



## Creation of Punch in Inserted step: A punching step (2/2)

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

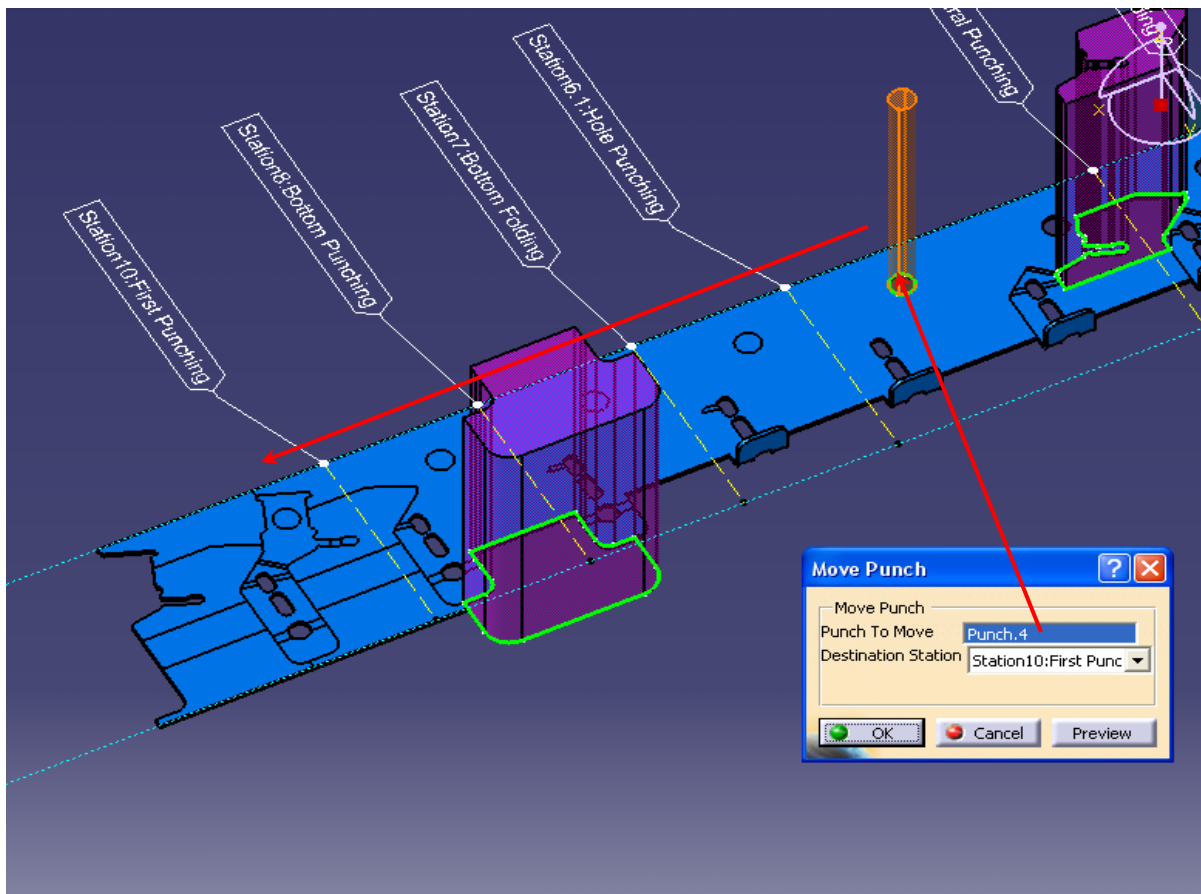
-  Use the Cutting Punch function by selecting the dedicated icon.
  -  The "Create Cutting Punch" window will be displayed in top left corner of the screen.
-  Select the "Station6.1 – Hole Punching" in the combo box in dialog.
-  Select the right top **blue punch contour** that will instantiate the 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 "Punch4" and "PositionningPunches" open bodies and in the Station 6.1 As well.









## Creation of the tenth step: Movement of Punch



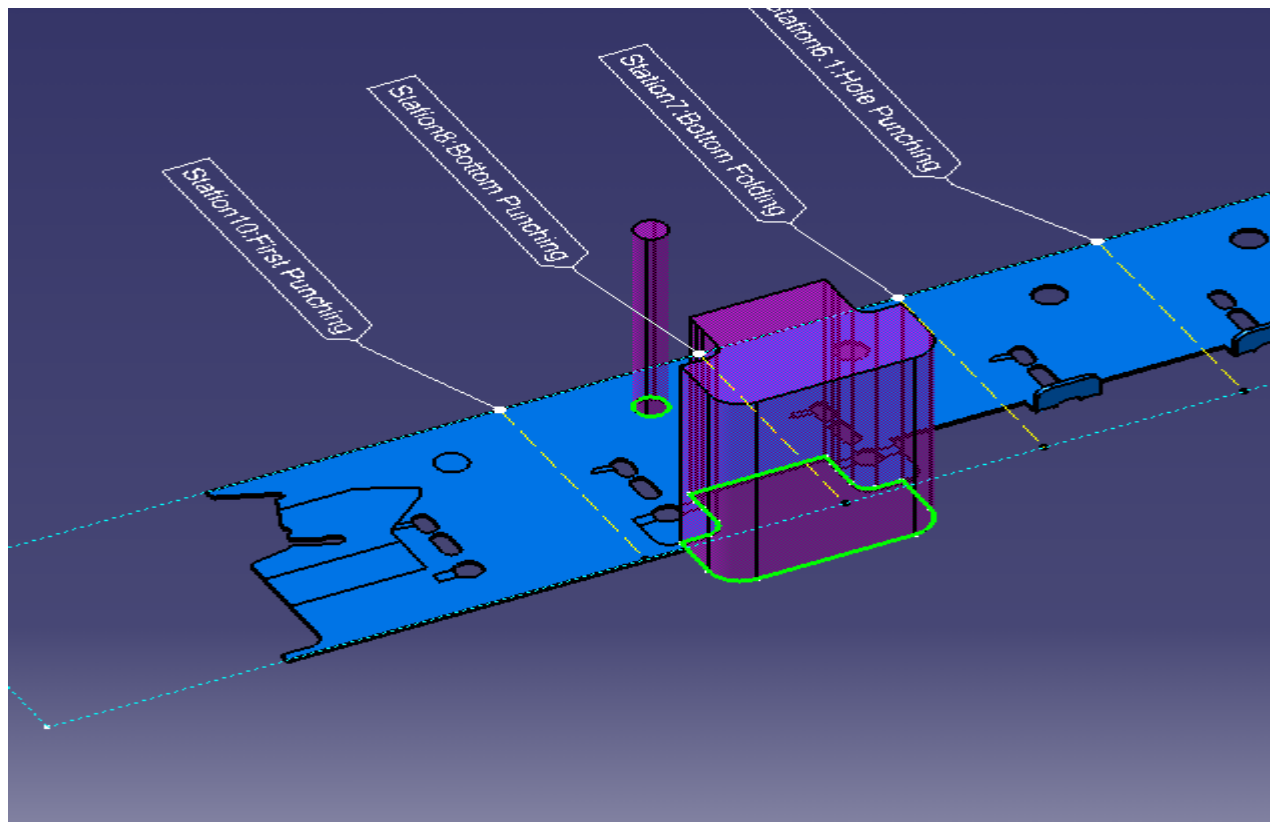
- **Add the step which will receive the punching operation:**
  - 👤 Select the **Add Step** function by selecting the dedicated icon and follow the same process as in the previous steps (Don't check the Insert Station Check box).
  - 👤 Enter the name of the station (i.e.: "First Punching")
  - 👤 Validate with the **OK** button.
    - 📌 The "Station10: First punching" is created in the "PartBody" body.
    - 📌 A parameter "Station10\_FreeStep" has been created and set to 0.



#### **Move the Punch to the Station:**

-  Select the **Move Punch** function by selecting the dedicated icon.
-  Select the Punch to move.(Select the Internal Punch).
-  Select the Destination as Station 10:First Punching.
-  Validate with the **OK** button.
  -  *The Punch is moved to Station 10:First Punching*
  -  *All the Features of Punch are moved to Station 10:First Punching.*





## Deletion of the previously created step

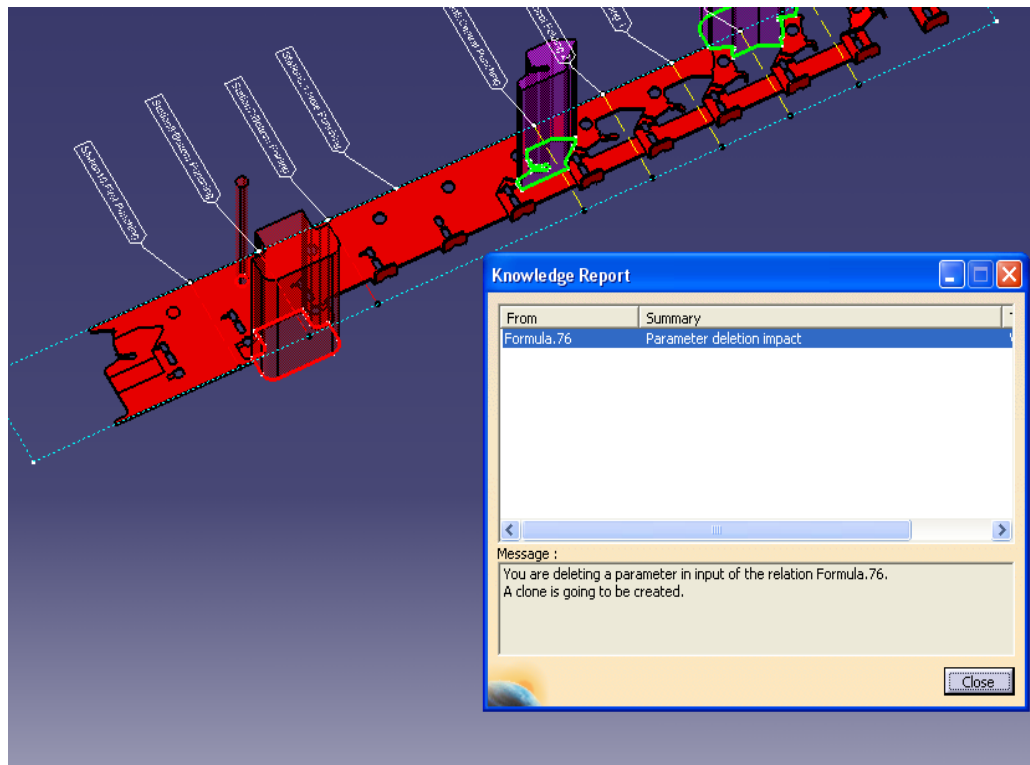
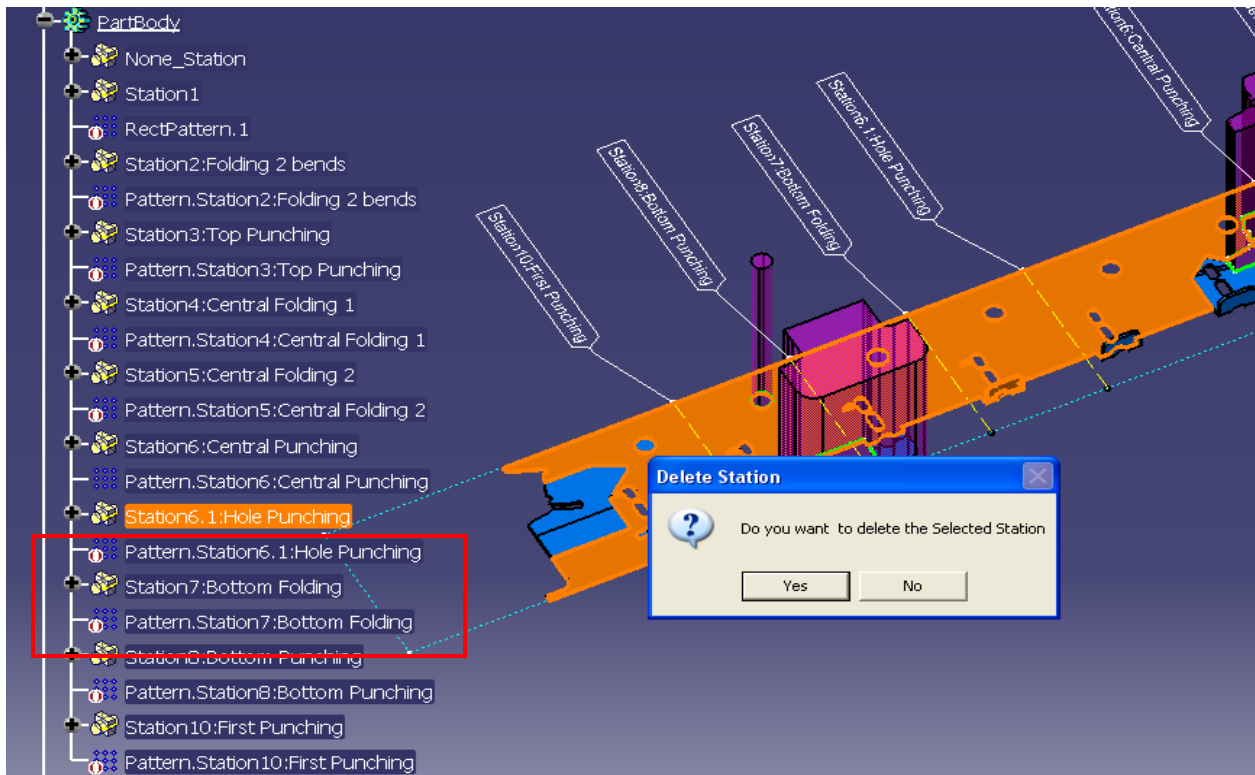


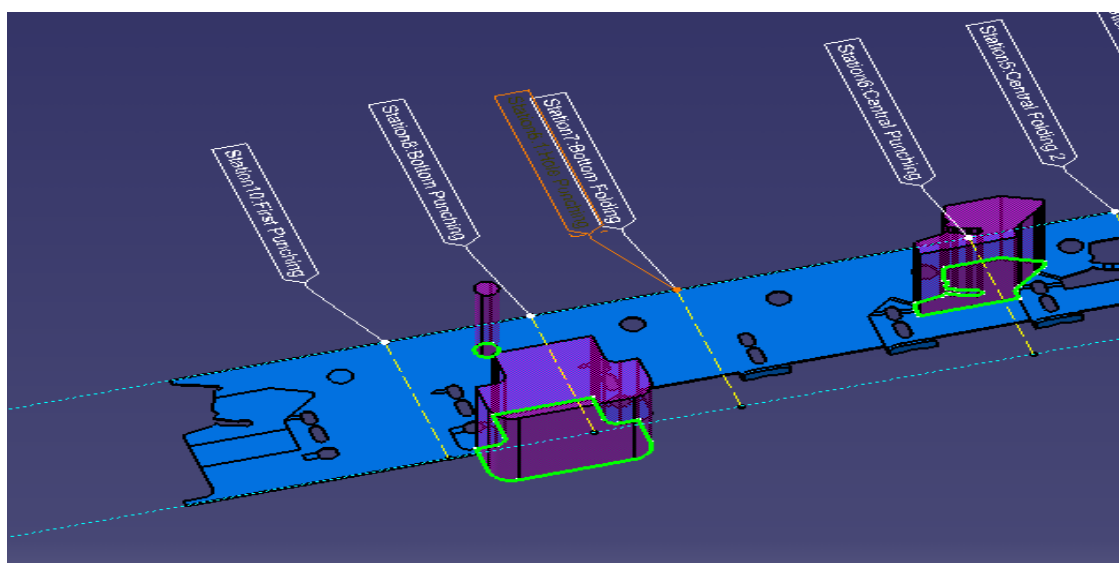
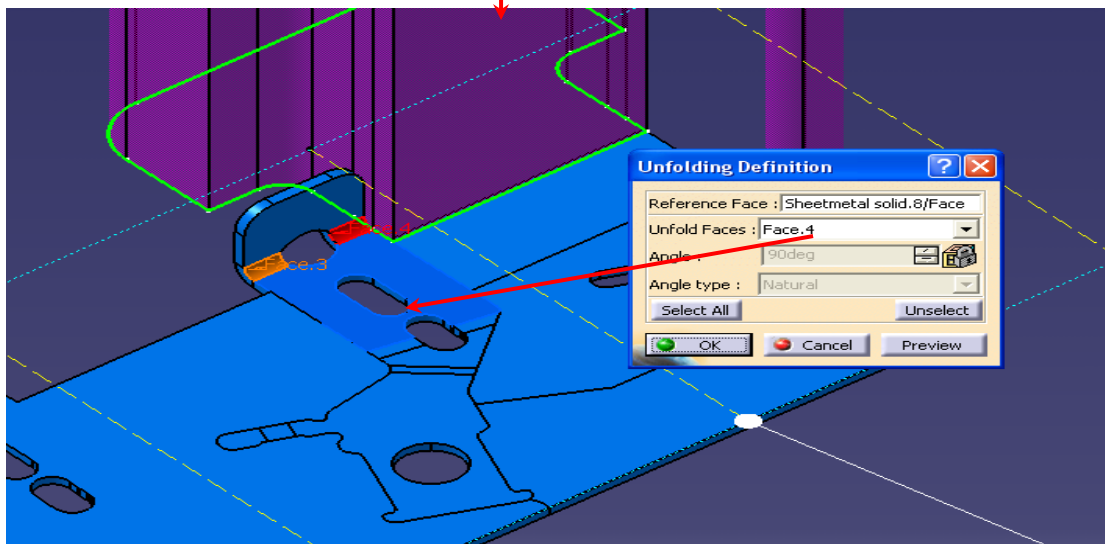
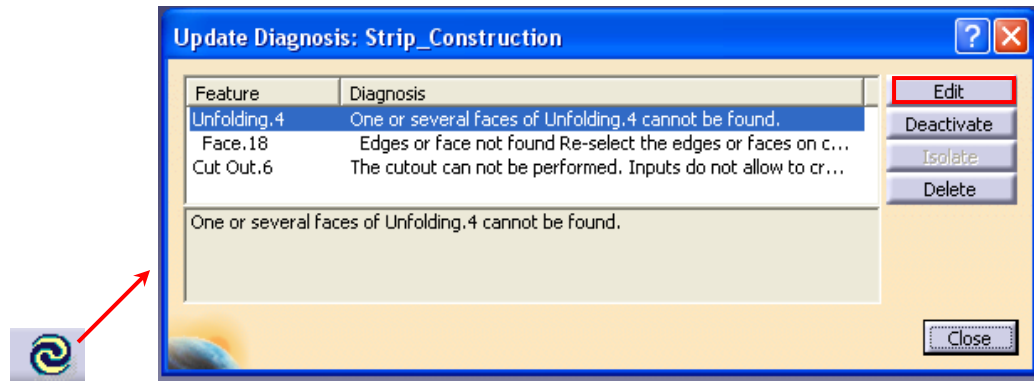
### Delete the previously created Step:

- ✎ Select the **Delete Station** function by selecting the dedicated icon and you will see a message in CATIA Status Bar Saying "Select the Station to Delete"
- ✎ Select the Station to Delete. ("Station6.1:Hole Punching")
- ✎ Validate with the **OK** button.
  - ℳ The Part is not updated in the case of Delete Station (Similar to Insert Station). This is due to the Unfolding feature. If there is an unfolding feature after the station which is being deleted the unfolding fails. In that case user has to perform following Steps:
    - ℳ After Manual Update, Update Diagnosis Dialog Box will pop up indicating that unfolding has failed.
    - ℳ Press on Edit, The Station which should have unfolding is defined in work object and Unfolding Dialog is launched.
    - ℳ User has to give the inputs to the unfolding feature once again and after giving the inputs to unfolding Insert Station is successful.
    - ℳ If there is no unfolding after the Deleted Station then the Update will be successful and there won't be any problems.
    - ℳ After Deletion of Station just select the Annotation and Press Delete to Delete Annotation
    - ℳ Please Refer the Pictures below.
    - ℳ The "Station6.1: Hole punching" is Deleted.






Select the Station to delete





**Mandatory:**

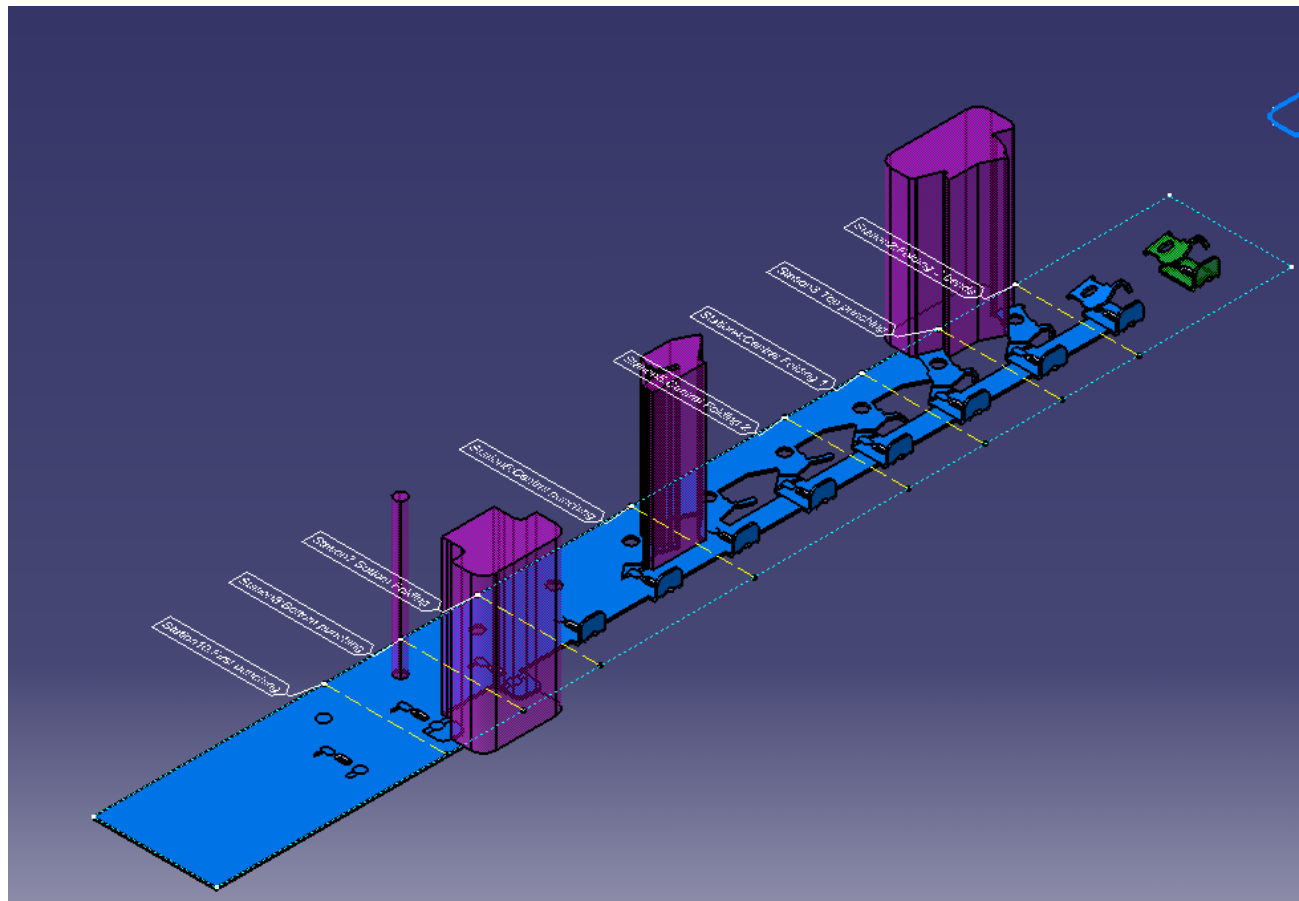
-  **After Deletion of Station User will have to do a Manual Update Always.**
-  **If there is an unfolding feature after the station which is being deleted Update will give a dialog box of Update diagnosis showing Unfolding Feature has failed, and User will have to edit it for a successful update.**
-  **If the Station to be deleted has a punch, the punch has to be moved or deleted from the station for deleting the station.**

## End of Strip and Trimming the Strip



**Delete the previously created Step:**



-  **Select the Trim Strip (End of Strip) function by selecting the dedicated icon.**
-  **Strip is closed and End of Strip is achieved.**

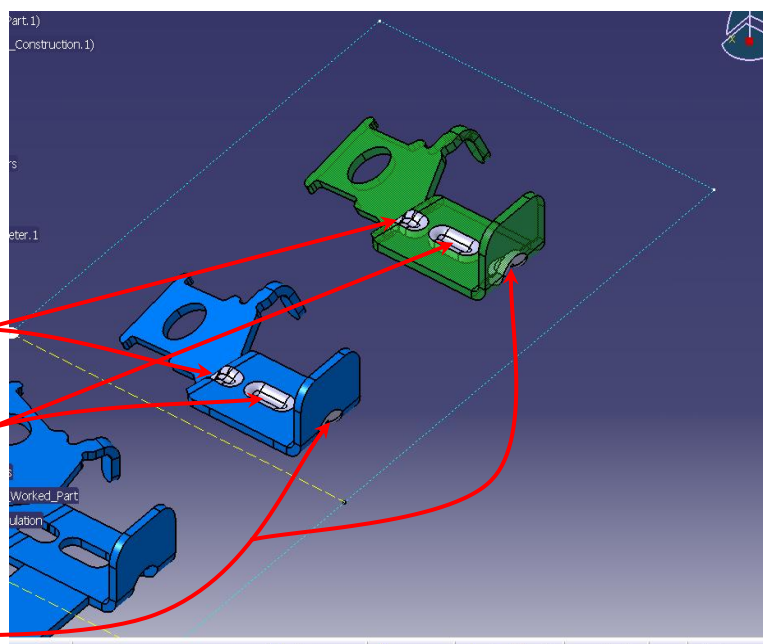
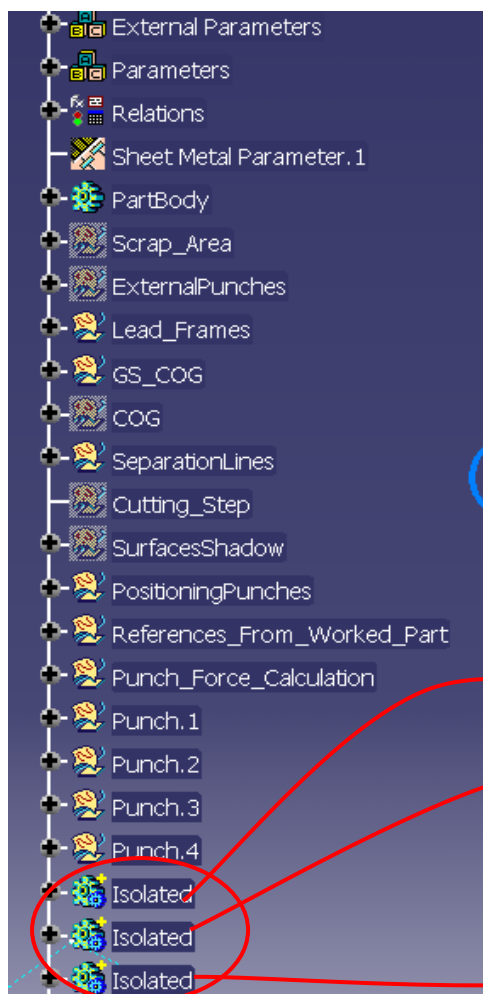


**Mandatory:**

-  **In order to achieve correct end of strip ensure that all the external scrap area is properly removed by punches.**

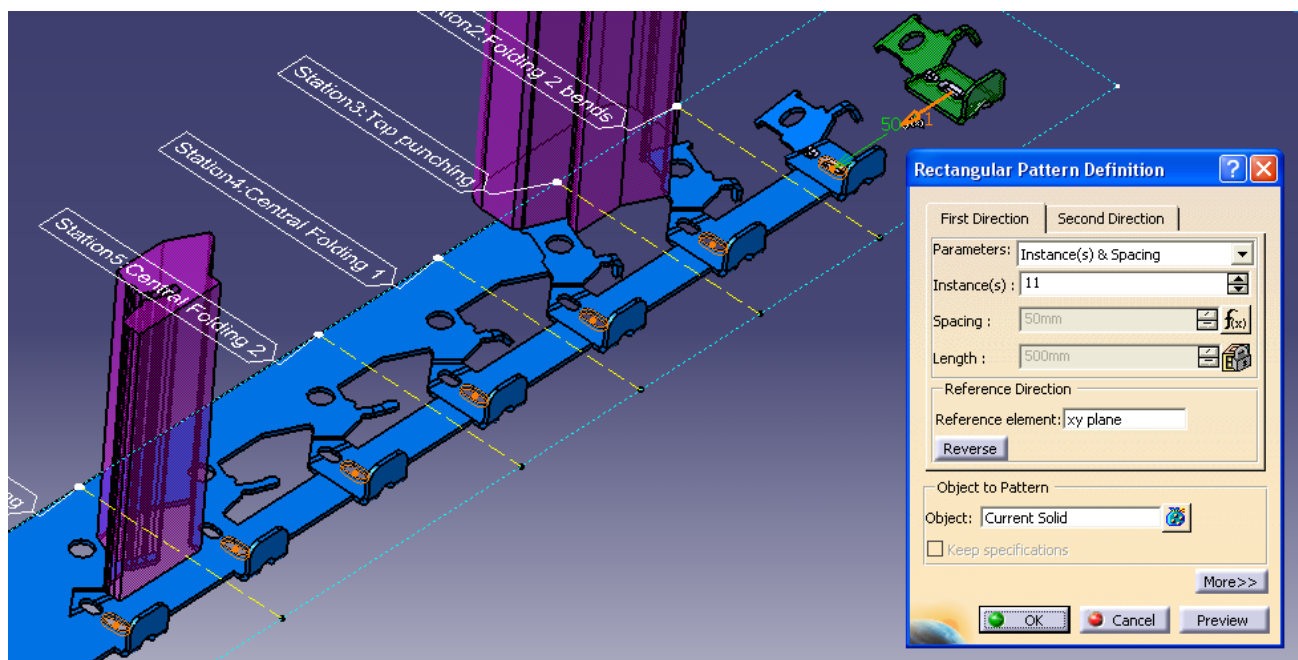
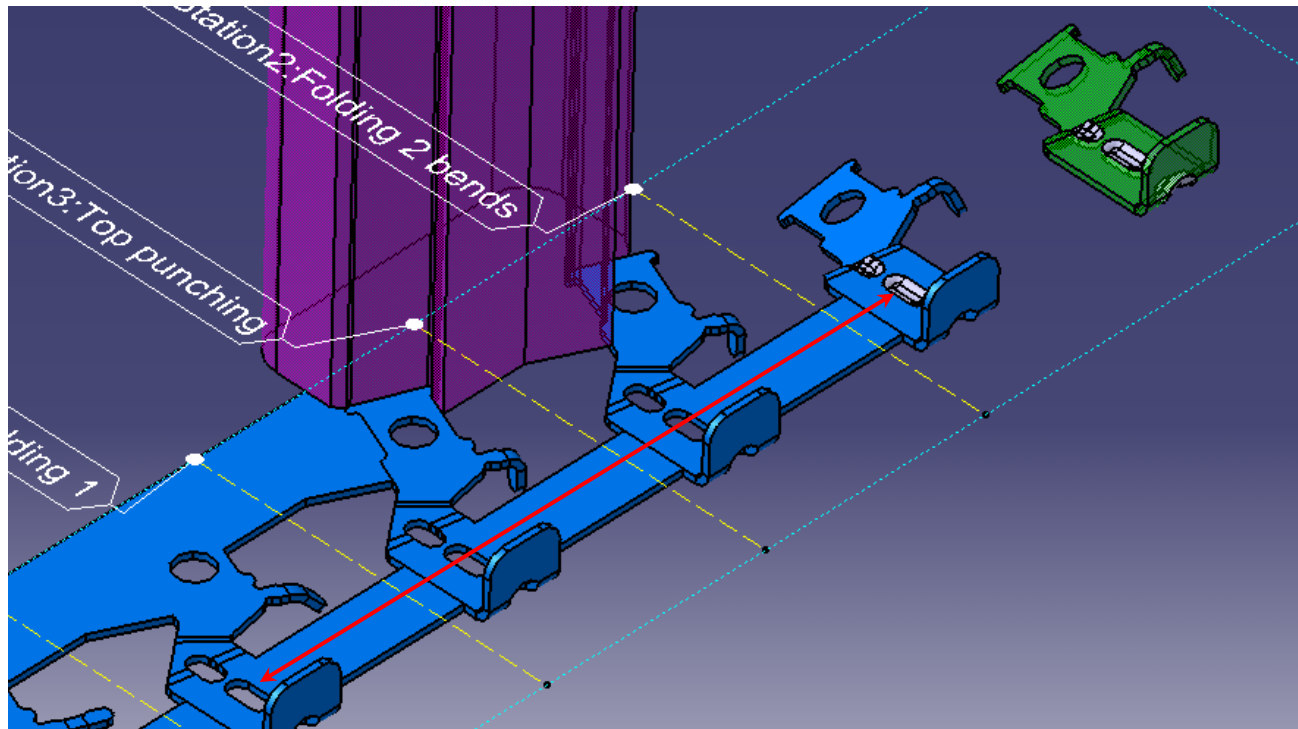
# Stamps, ribs and complex geometries recovery

- **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 Strip Construction part.
  - ⤿ Launch the “Automatic Recovery” function.
    - ⤿ *The isolated geometries are copied to the None\_Station and the Station1 – Last punching.*
    - ⤿ *Three “Isolated” bodies are added to the “Strip\_Construction” part.*



Translating geometries to the other steps.

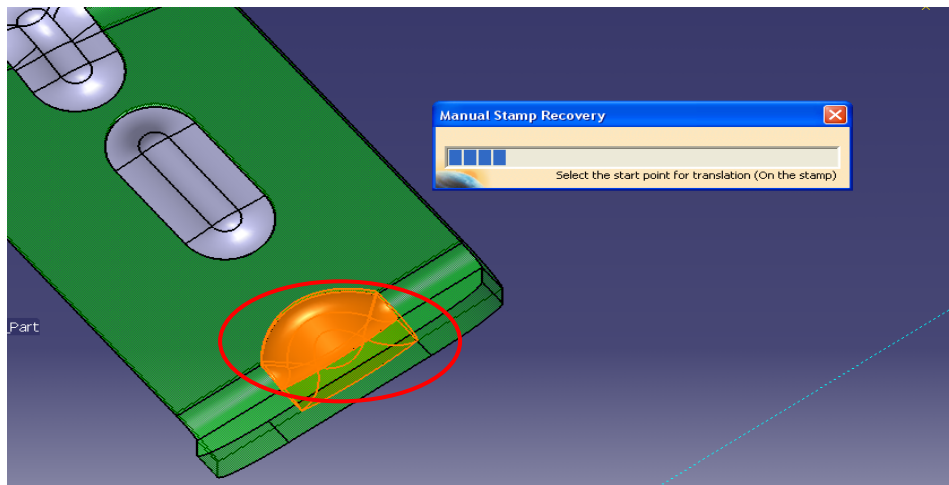
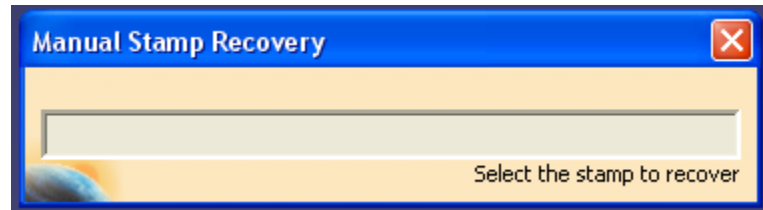
- Expand the PartBody and all Isolated bodies.
- If Rotation of Stamp is not required user can select the Pattern Feature and Increase the number of Instances based on Number of Stations.



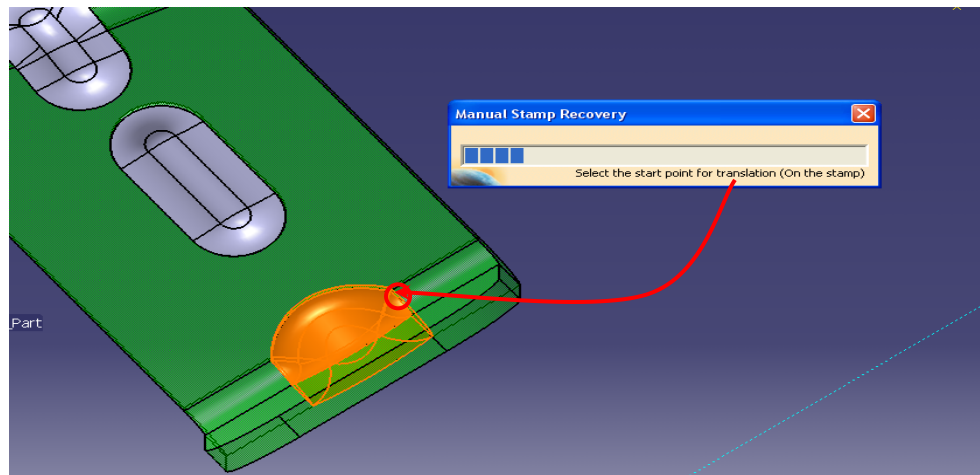


*Another Method to recover Stamps or complex geometries is by using Manual Recovery Function*

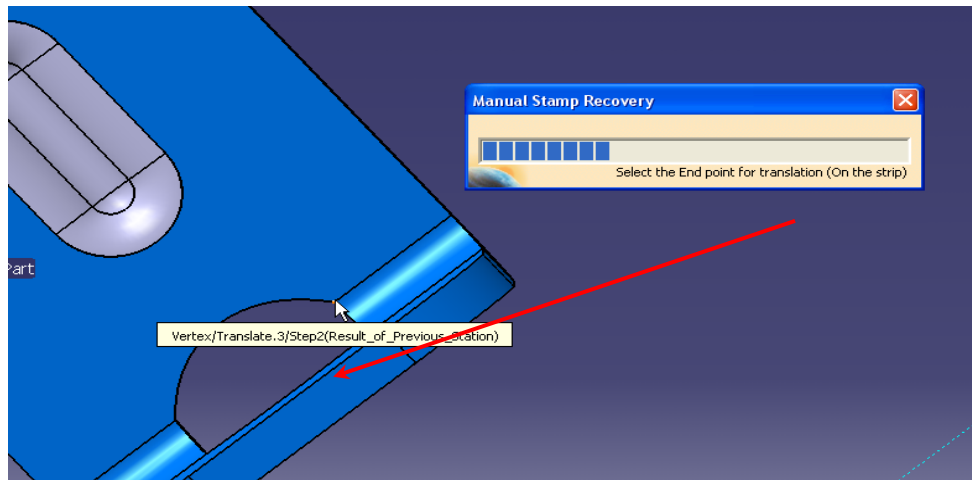
- 👤 Launch the “Manual\_Recovery” function to translate the missing geometry to Station2.
  - 👉 The geometry is close to the Station2.
  - 👉 It is asked to select solid.



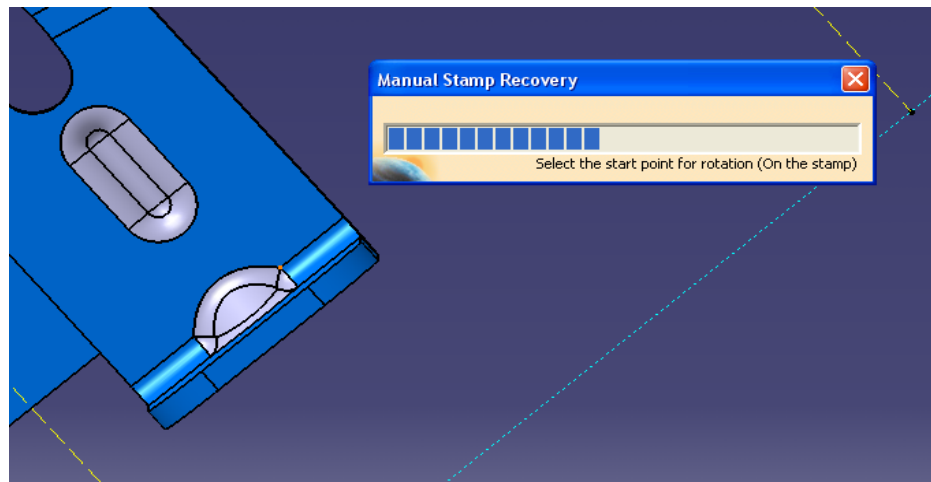
- 👤 It is asked to select start point for translation on the stamp



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

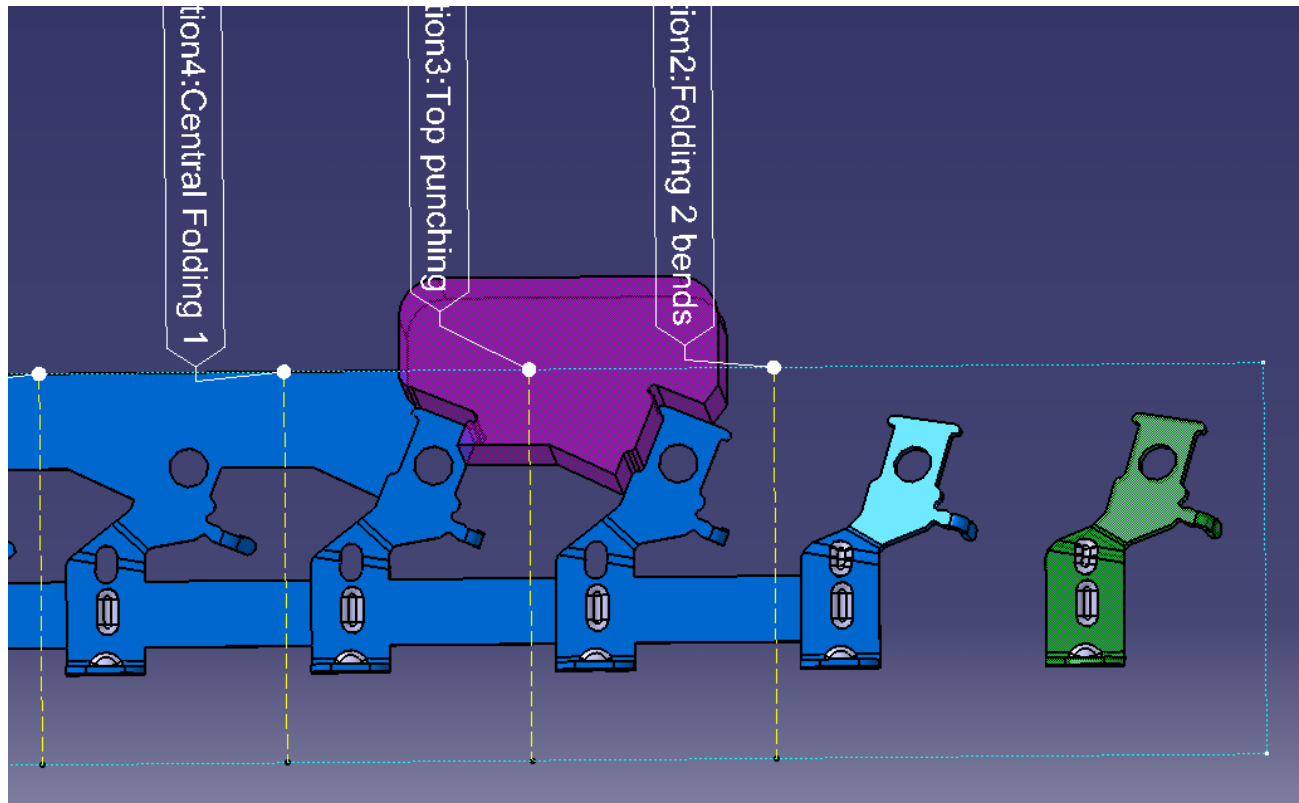


- 👉 The geometry is translated and expected result is obtained.
- 👉 Close the Manual Recovery Function.



You will see that this stamp is already translated and we have received expected results with just 1 translation. In some cases where it is required to rotate the stamps this function is very useful. In order to use this function just follow the steps correctly.

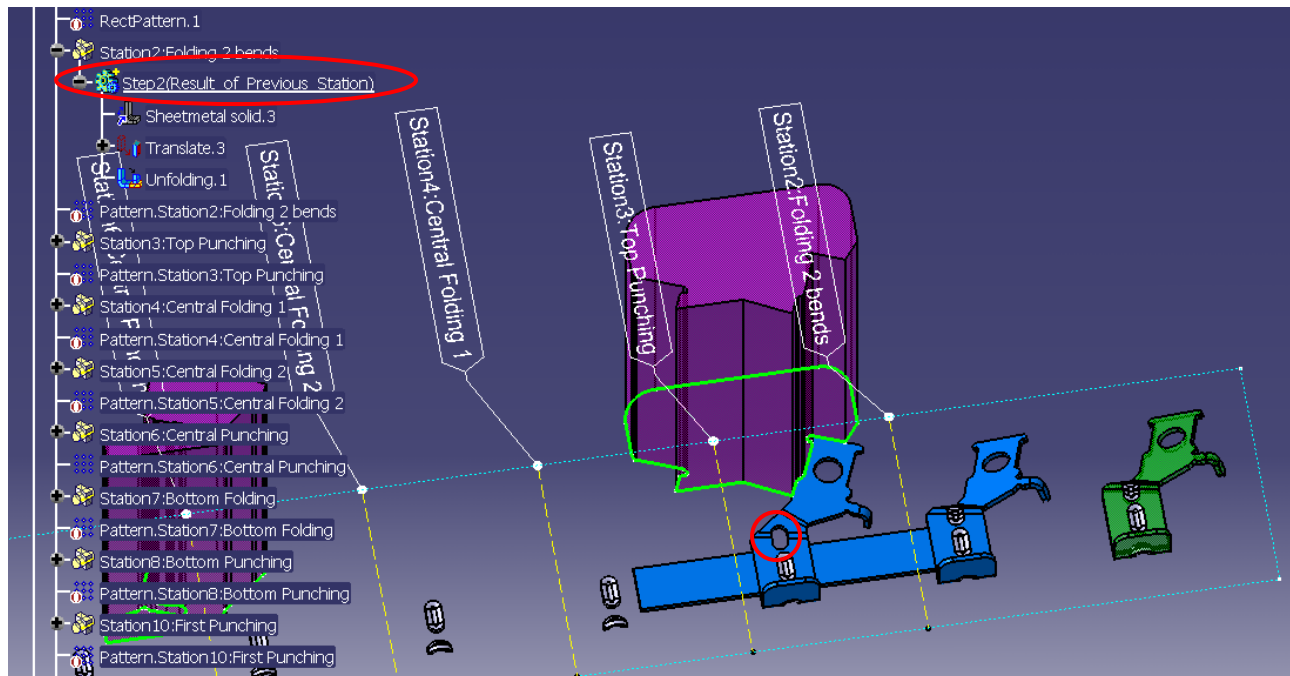
- 👉 Recover the stamps at rest of the stations using the pattern Methodology or Manual Recovery.







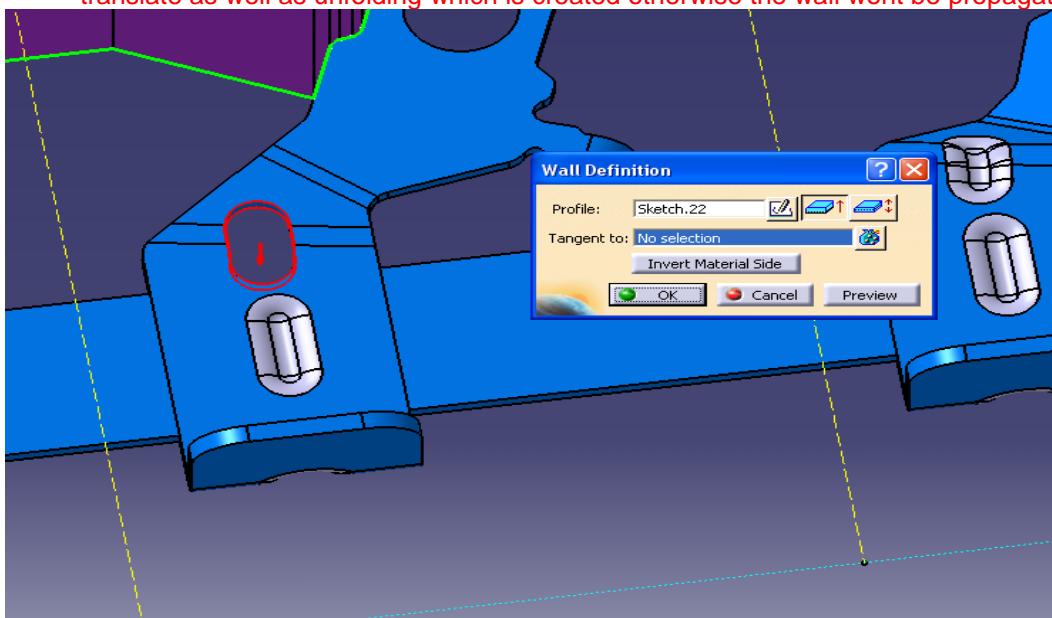
## Removed geometry holes filling

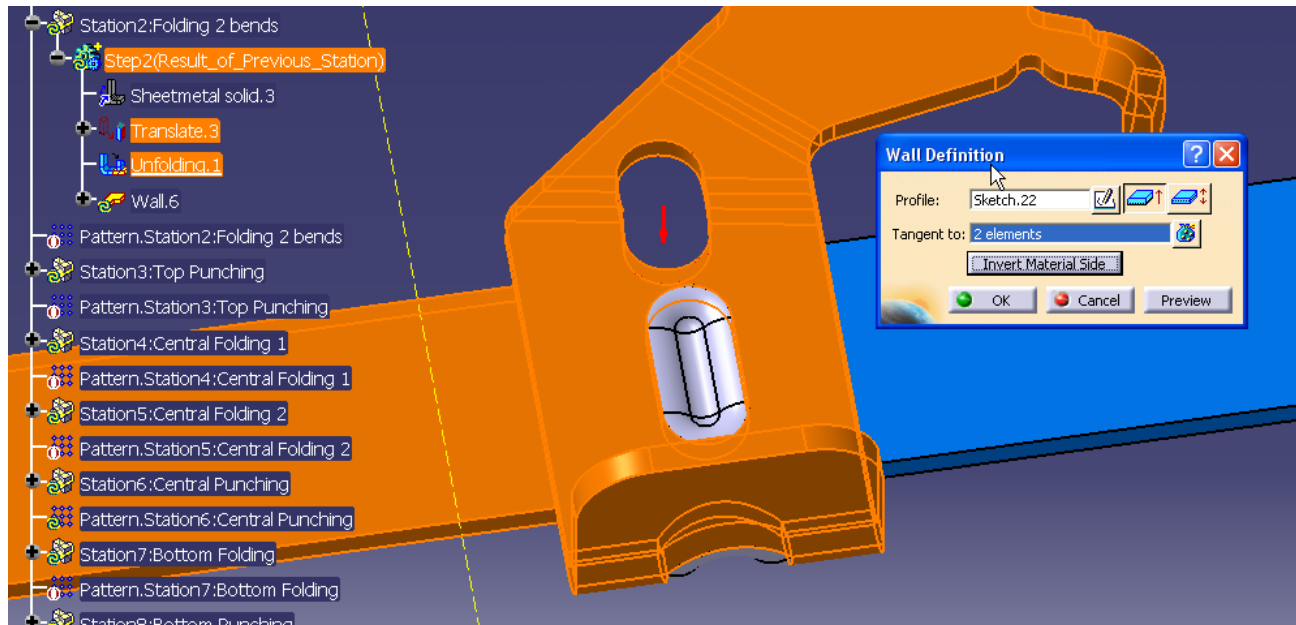
- As you can see in your strip, there are lots of holes to fill. The following operation can be done whenever you want during the strip construction. We have decided to group them at the end only for clarity.
- We will illustrate the two main possibilities to fill the holes, but if you are familiar with 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.



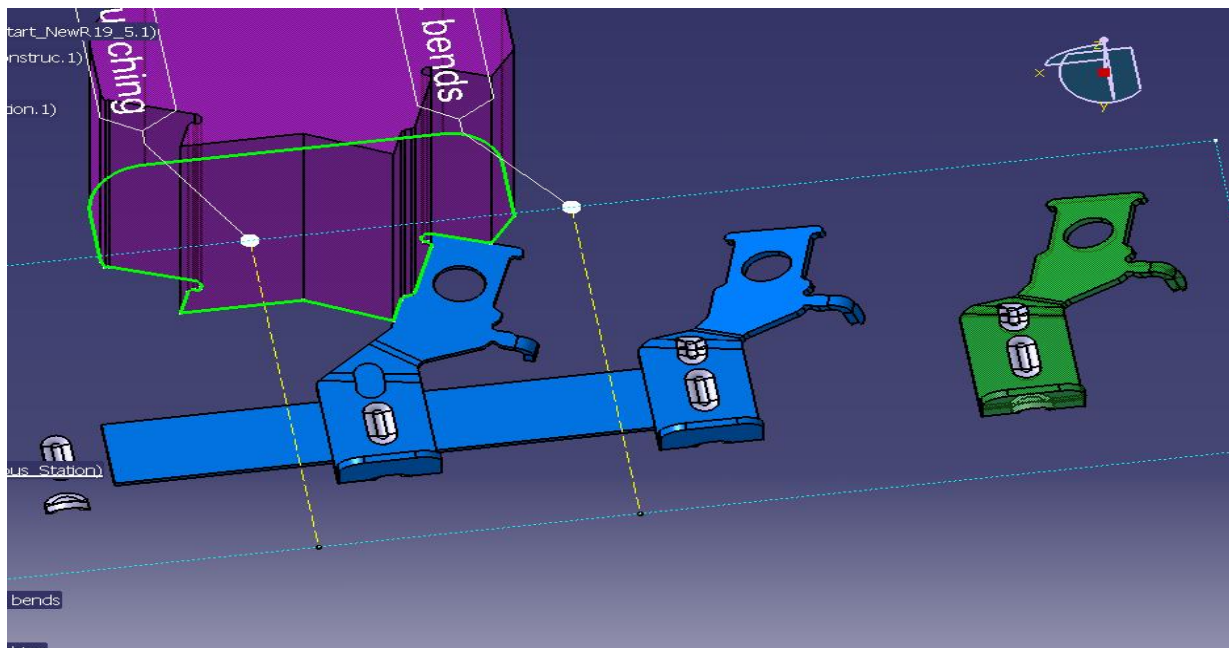


-  Launch the “wall” function.
-  Create a sketch on the upper face of the strip and double-click on the “Project 3D Elements” icon (  ).
-  **Mandatory:** Wall has to be made Tangent to the last sheet metal solid feature in this case translate as well as unfolding which is created otherwise the wall wont be propagated.

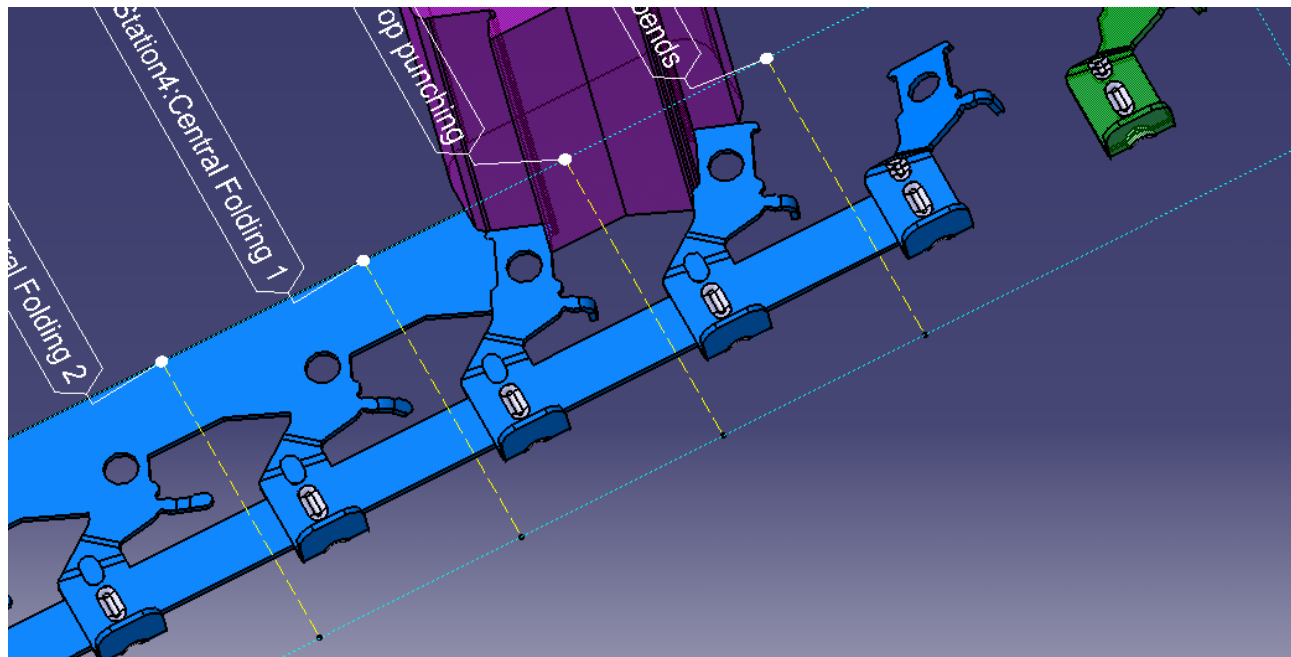




- Click OK to validate.
- The hole is filled.

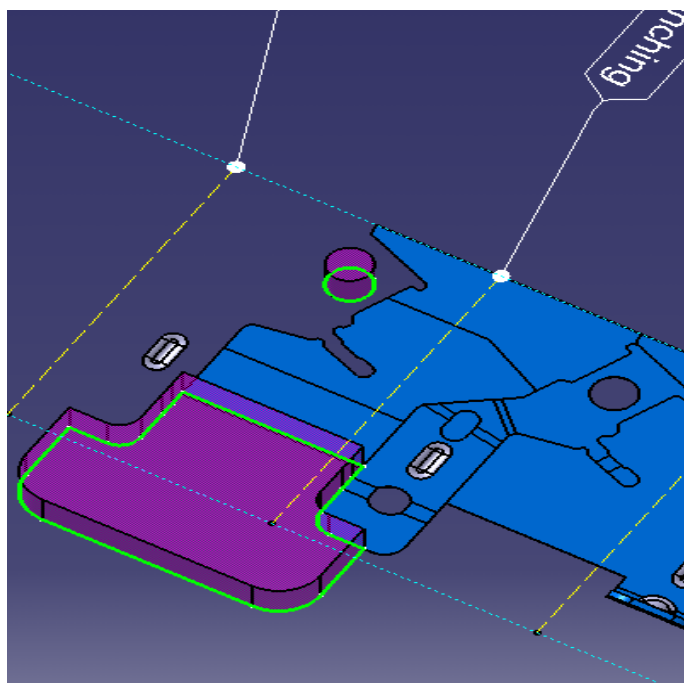






- Define the PartBody as in work object
- The fill is propagated to all the following steps.

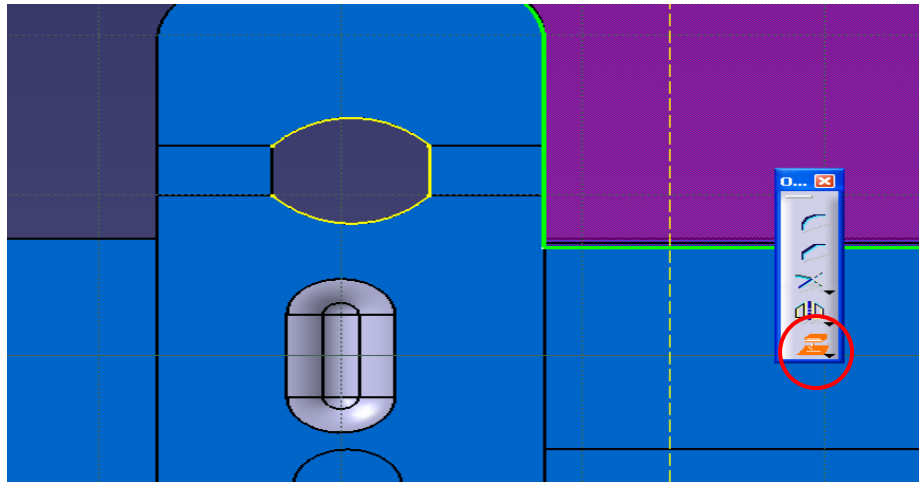


#### **Filling the Hole in Station 7.**

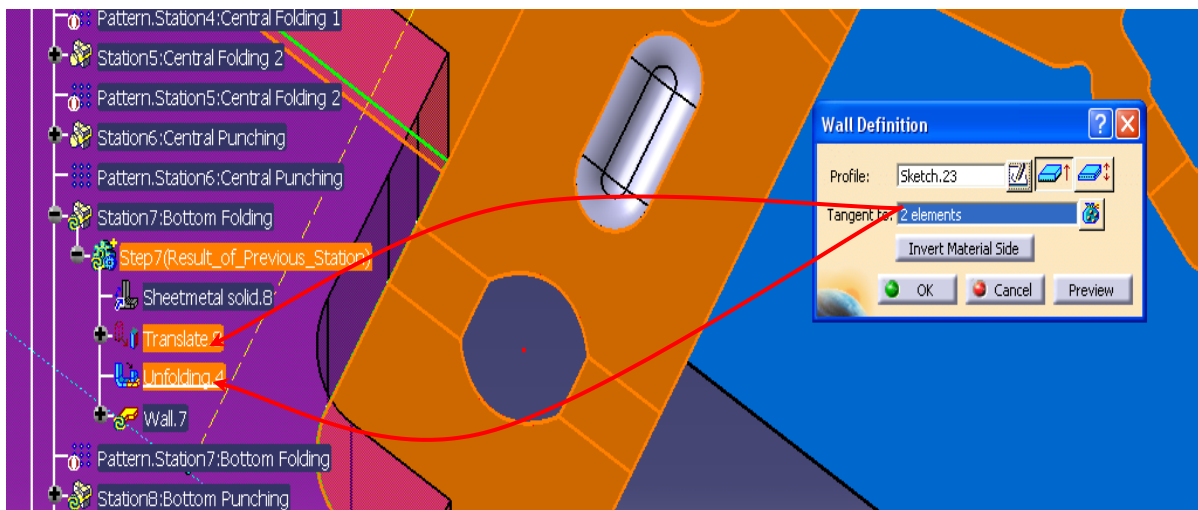
-  Define the body under Station 7 in work object.
-  The following steps are hidden.



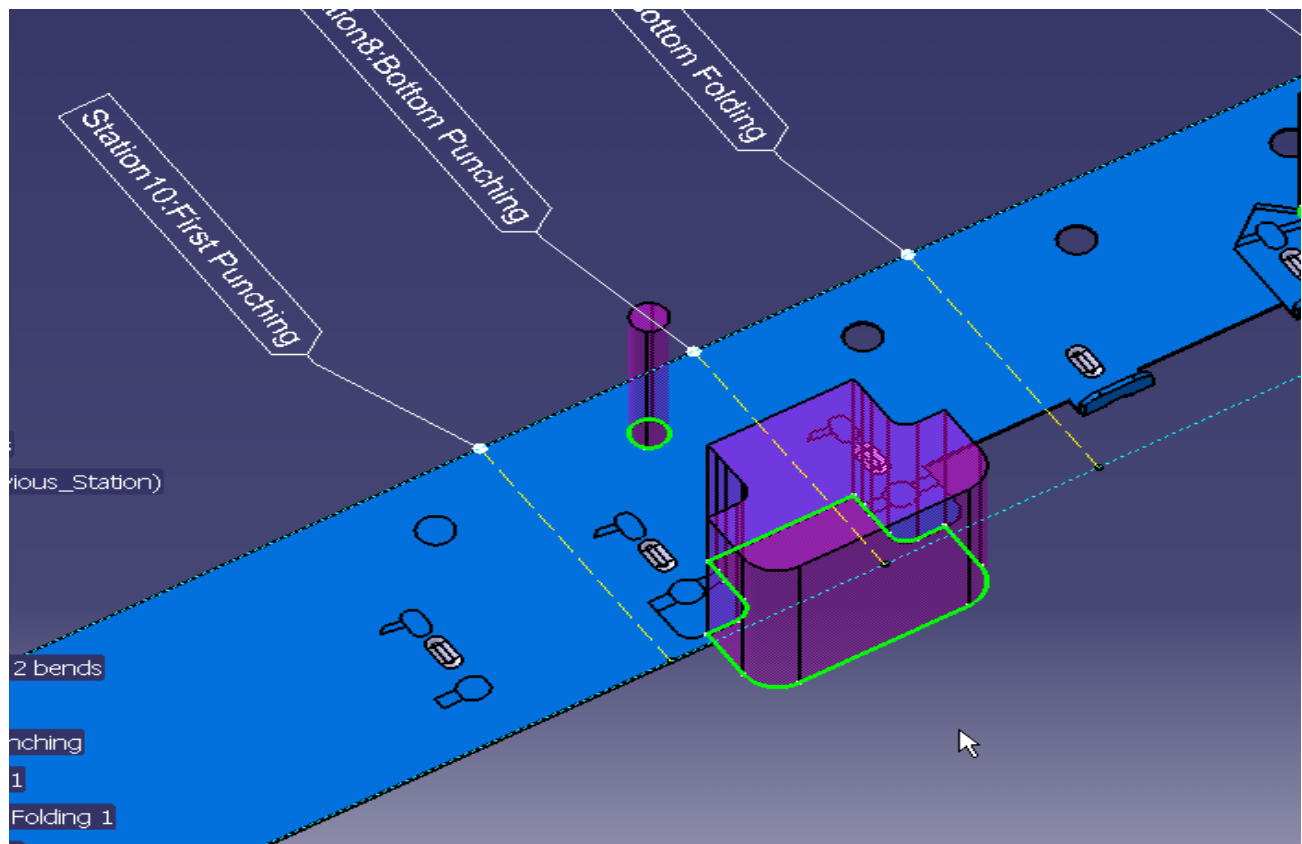
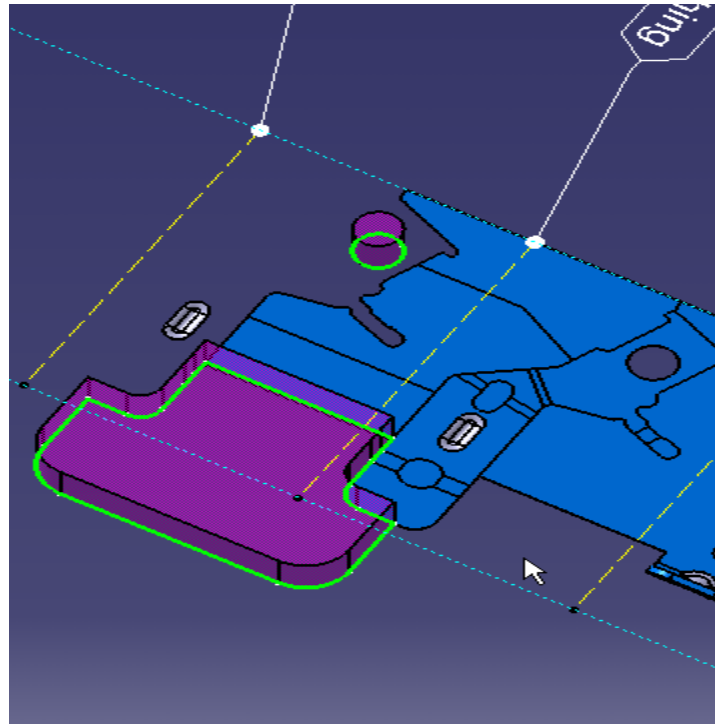
-  Create a sketch on the upper face of the strip and double-click on the “Project 3D Elements” icon ().
-  Pick, one by one, all the edges of the contour of the hole you want to fill.
  -  The edges are projected in yellow on the sketch.



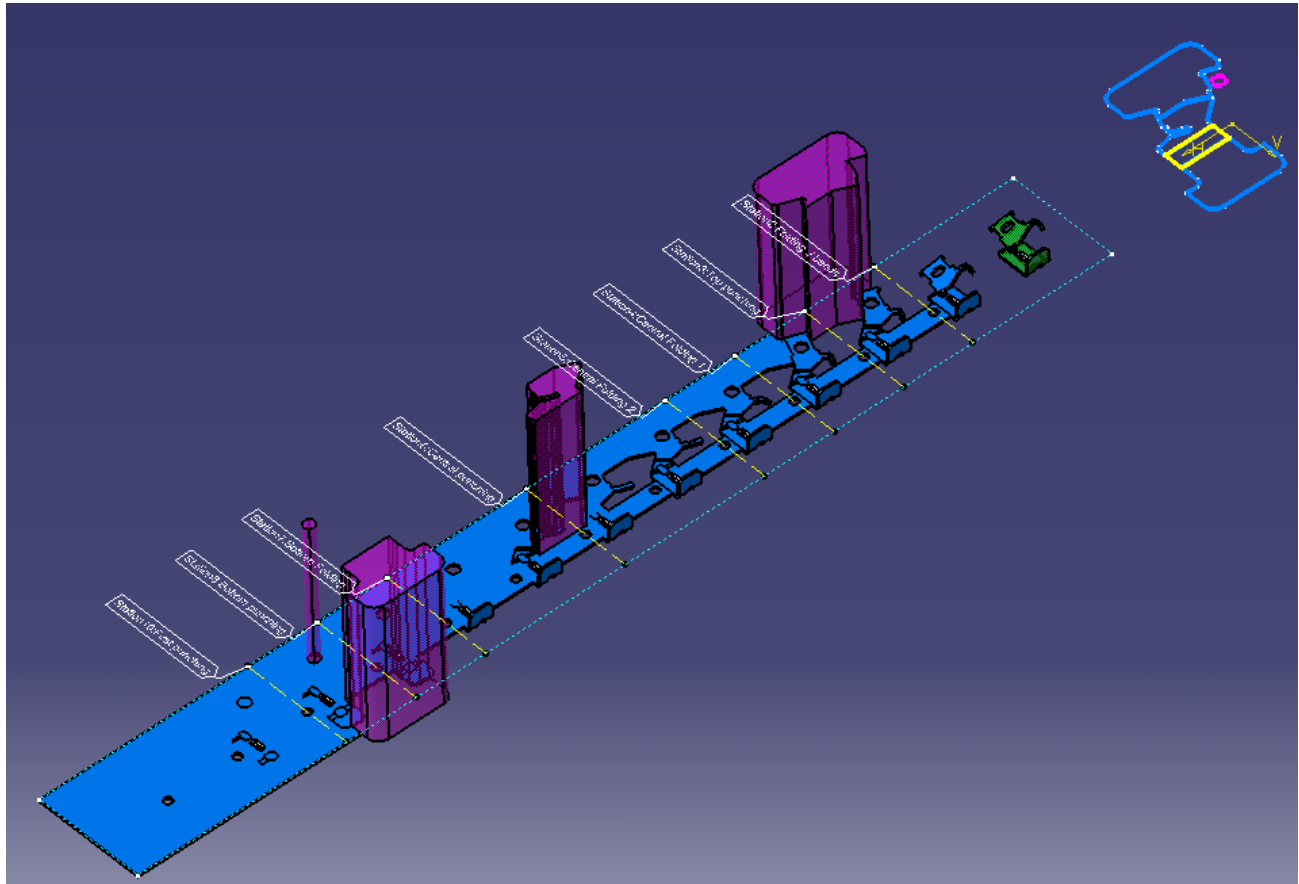
- 👤 Leave the sketcher.
- 👉 The sketch is visible with the hole's contour.
- 👤 Launch the wall Definition.
- 👤 Pick the previously created sketch, check the extrusion direction.
- 👤 Also Make it tangent to the Solid translate in respective station.



- 👤 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.















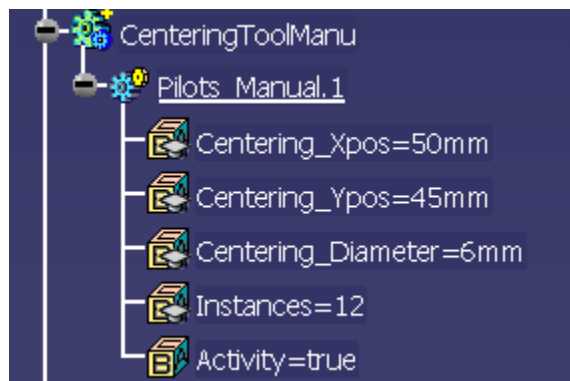


-

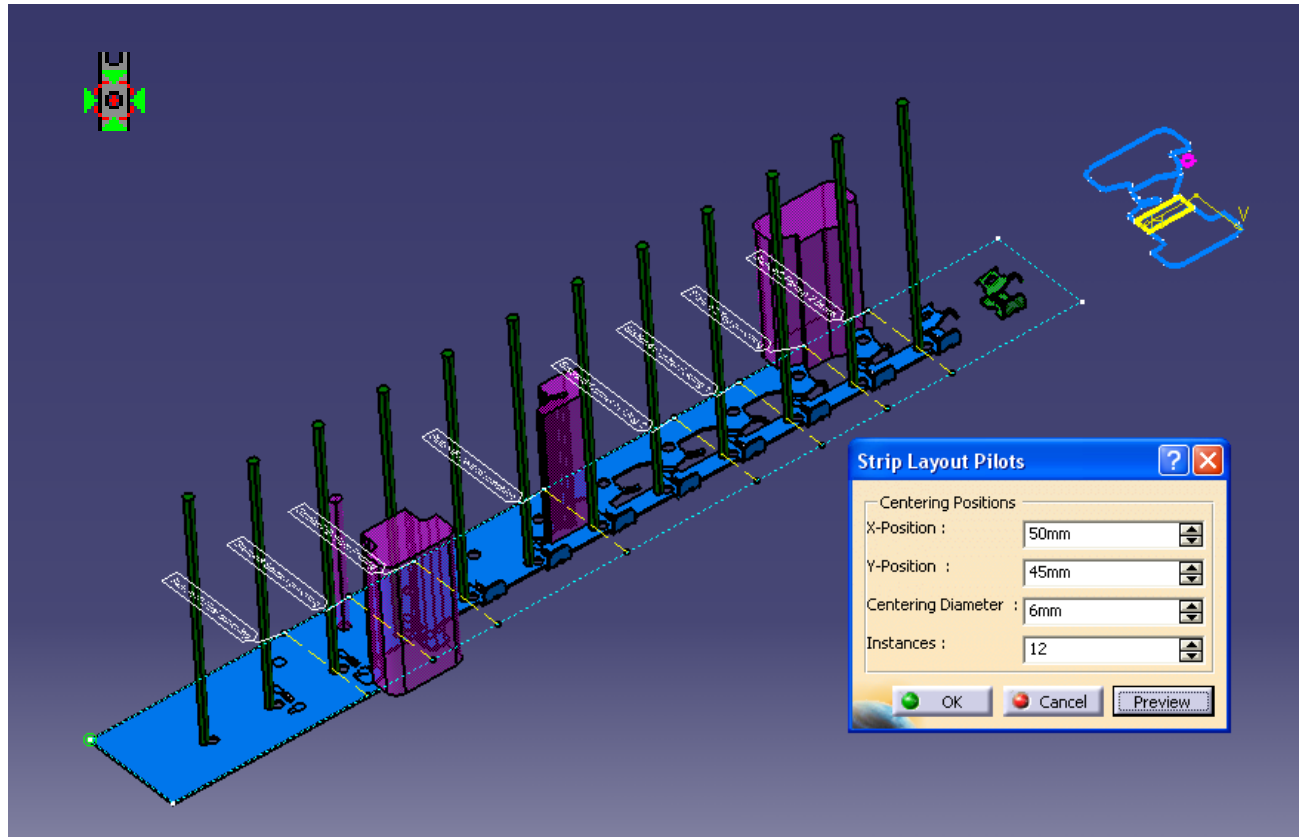


## Instantiation of the centering tools (Manual pilots)

- The centering tools can also be instantiated manually if the user wants to instantiate his own number of tools:
  -  Activate the Strip Construction part.
  -  Select **Centering\_Tools\_Manual** function.
  -  Change the values:
    -  Set **Centering\_Xpos= 50 mm**
    -  Set **Centering\_Ypos= 45 mm**
    -  Set **Centering\_Diameter= 6 mm**
    -  Set **Instances= 12**
  -  **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*



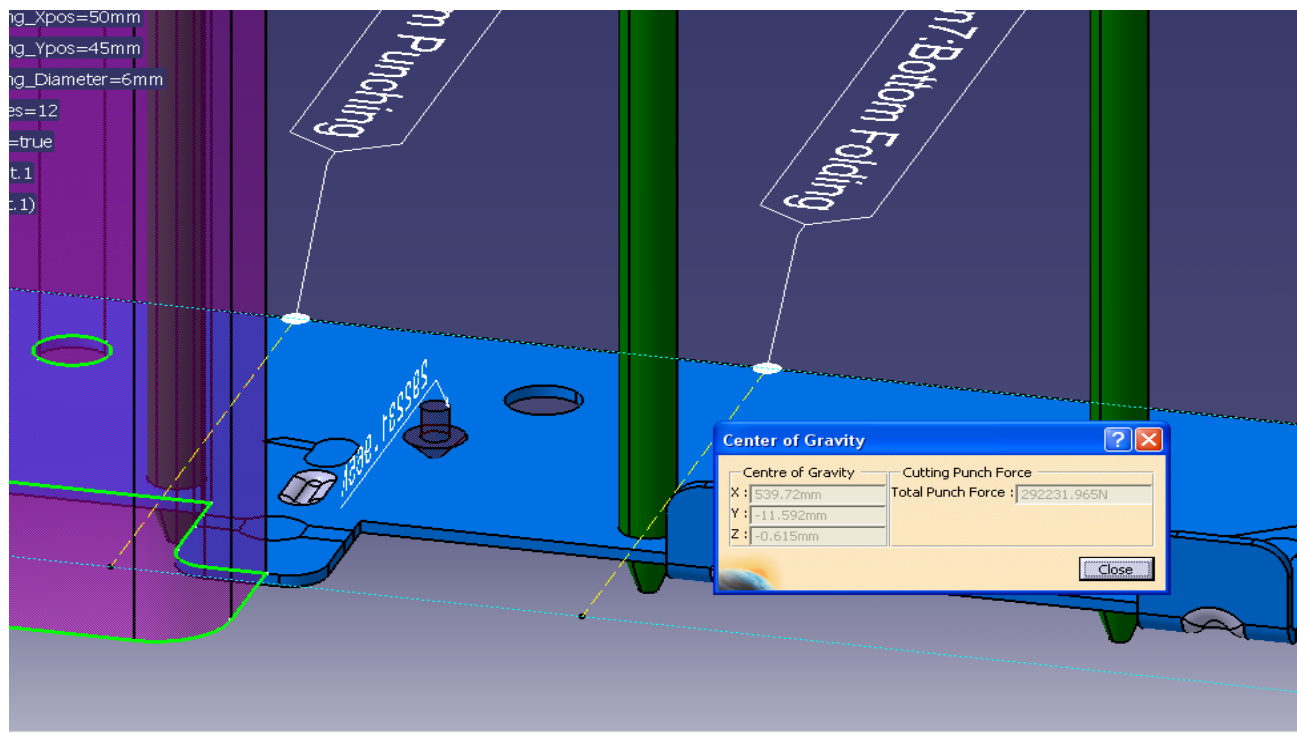
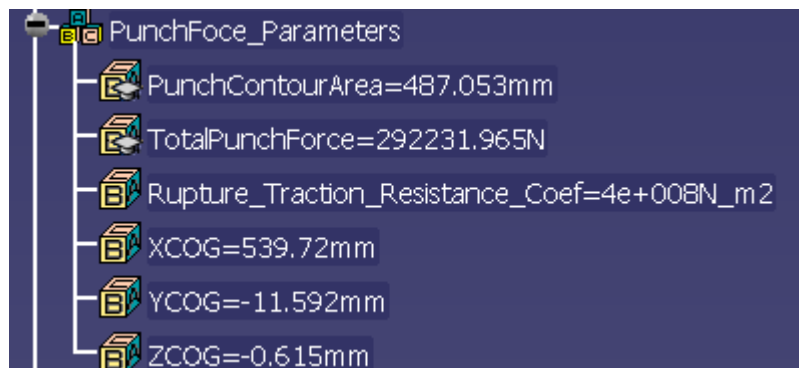




# Centre of Gravity and Punch Force

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


- ✎ Edit “Rupture\_Traction\_Resistance\_Coef” by double-clicking on it and put an appropriate value.
- ✎ Total Punch Force will automatically calculate and will be shown in parameter “TotalPunchForce”.
- ✎ Also User can launch Centre of Gravity function by selecting the dedicated icon. And user will be able to see the Centre of gravity coordinates as well as total punch force. This command can be launched at any point of time during design of the strip.
















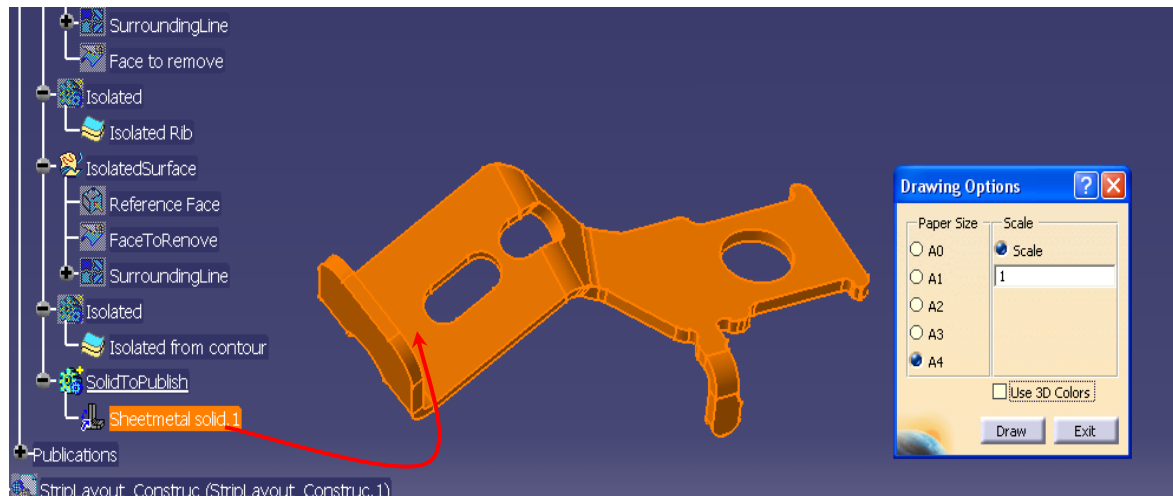
# Drawing generation

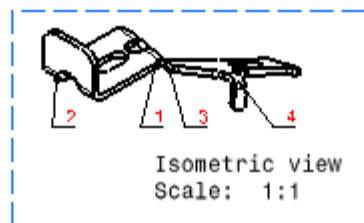
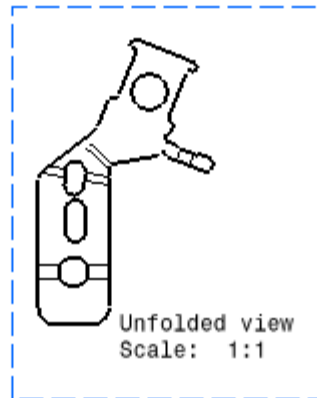


## Generation of Isometric View, Unfolded View of the Finish Part.

-  This methodology offers you the possibility to compute the Isometric View of your part for which strip is designed. The advantage of this method is that it creates a bend table with the drawing which displays parameters like Bend Angle, Bend Radius-factor.

-  Activate "Part for Strip" Part.
-  Launch the "Generate Flat View" function.
  -  Select the Sheetmetal Solid from SolidToPublish Body in the Part.
  -  This body is created during Strip preparation.
  -  The Drawing options window is displayed.
  -  Give the Scale as 1: 1 and select A4 Sheet Size.
-  Click Draw
  -  The drawing is generated.
  -  There are two views in the drawing.
    -  Unfolded View of the Solid
    -  Isometric View of the Solid
  -  In Isometric view Bend Annotations are also created.
  -  A bend table is generated.



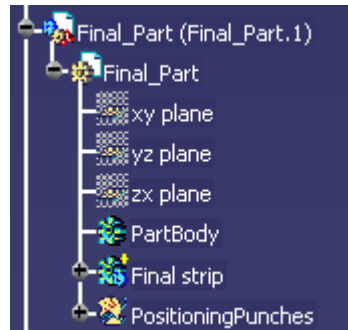


B.No	B.Angle	B.Radius	K-Factor	B-Length
1	52.250000	1.500000	0.325257	1.664515
2	90.000000	1.500000	0.325257	2.867108
3	37.750000	1.500000	0.325257	1.202592
4	90.000000	1.500000	0.325257	2.867108

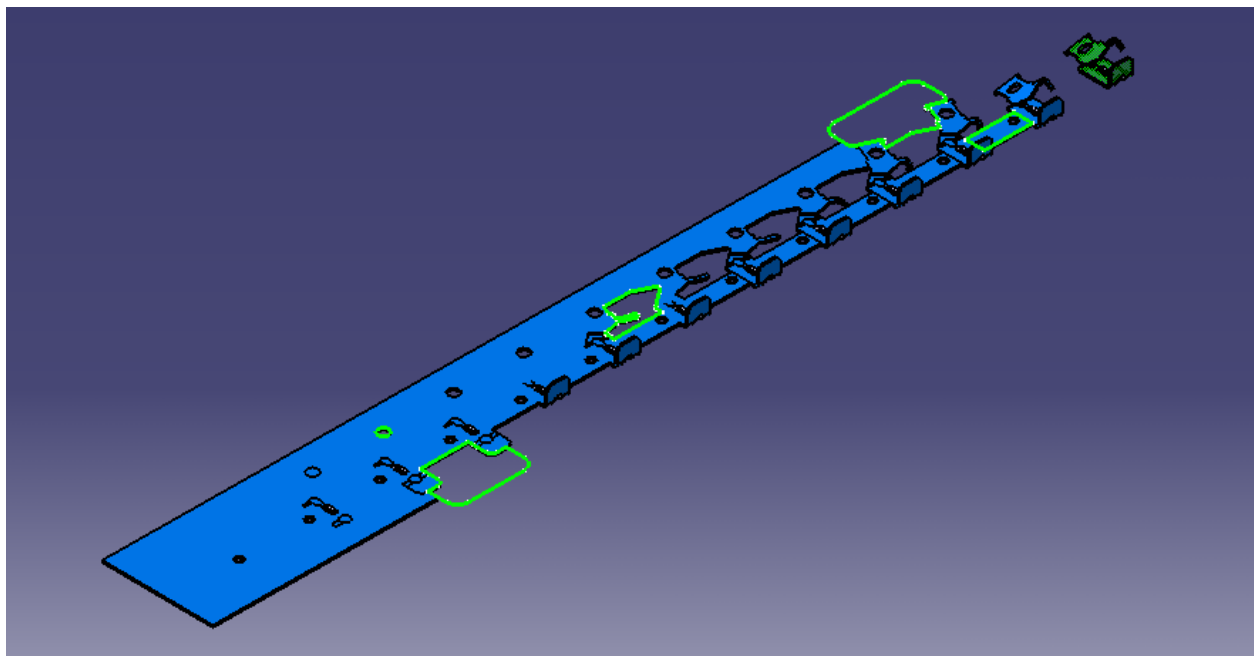


## Final part computation

- 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.






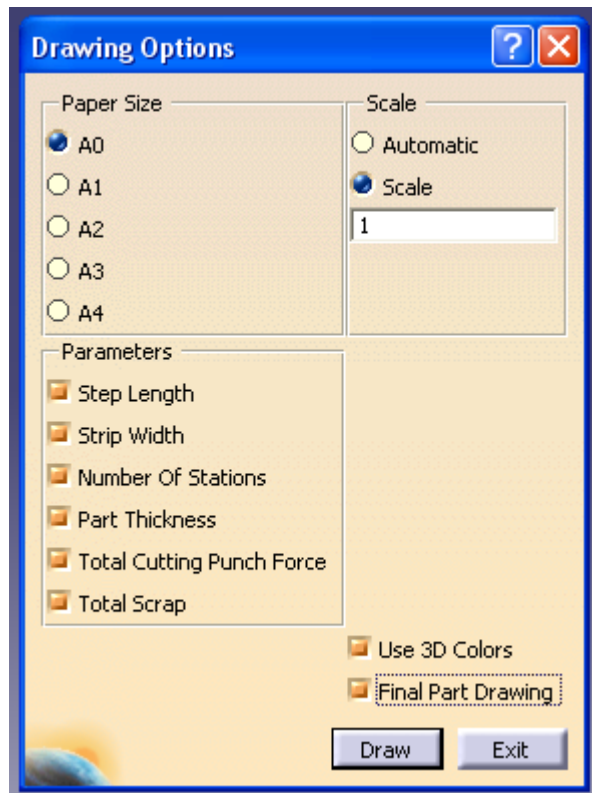
- Now, let's compute the Final part.**
  - Activate the root product.
  - Launch the final part function
    - The final part is computed.*
  - Hide the "Strip\_Construction" Part.
    - You can see the final strip.*

















## Creation of Automatic Drawing of the Whole Strip

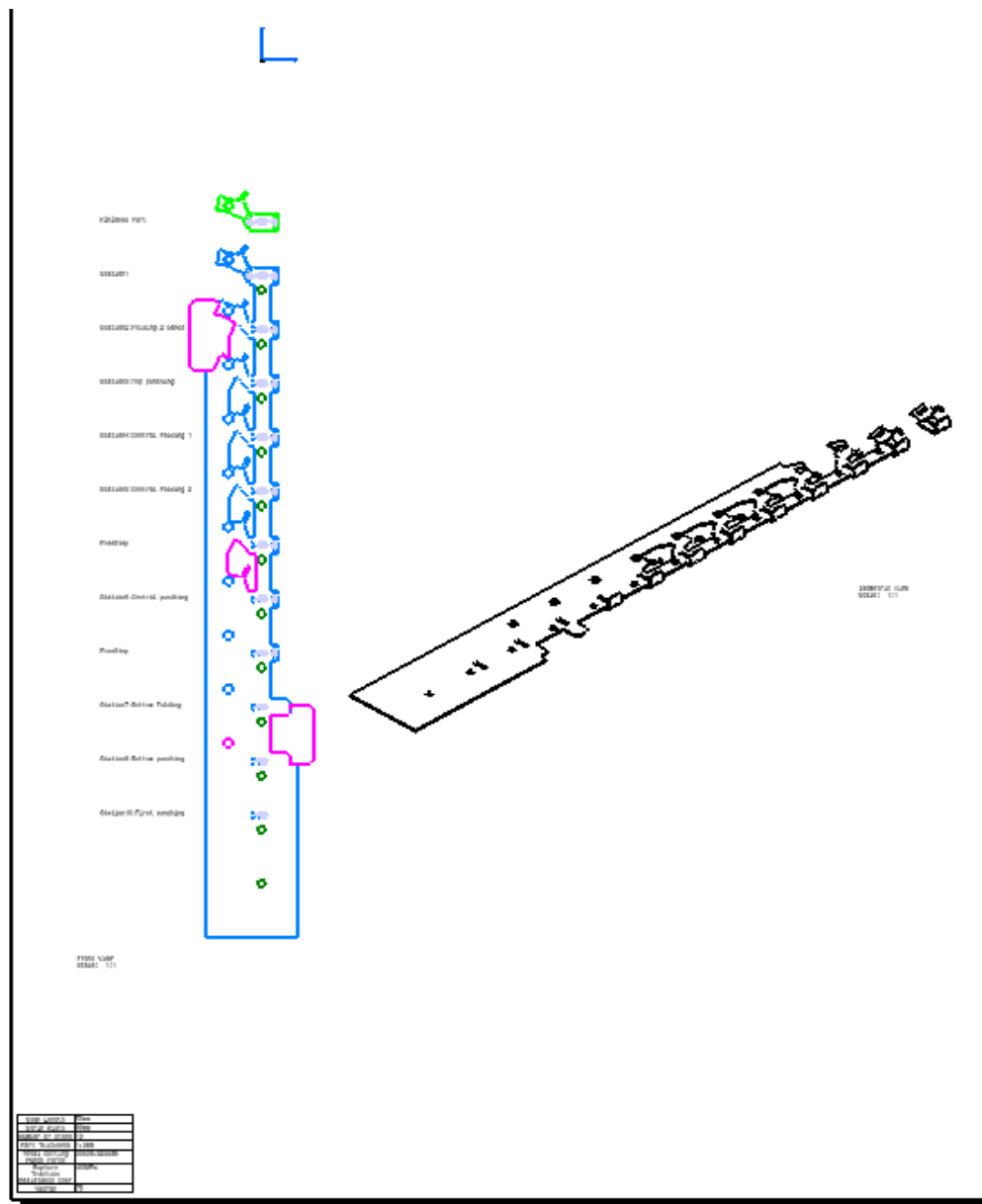
-  This methodology offers you the possibility to compute the top view of your strip.

-  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 colours, select the “Use 3D colours” 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.*
  -  *Two Views are Generated:*
    -  *Top View of the Finished Strip with Punch Contours*
    -  *Isometric View of the Strip. In Case of Isometric View if Final Part Drawing Button in the Dialog box is checked then Isometric View of the Final Part is generated. If it is not checked then Isometric View of Strip Construction Part is generated.*
  -  *The name of each station is written in front of then. Free steps do not have name.*

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





# Conclusion

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- **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**