

TP1

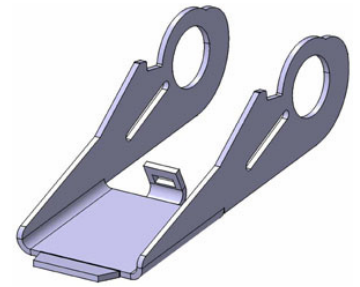
Support: strap operating lever

Objectives: implement the sheet metal workshop

procedure :

Access to the Sheet Metal Part Design Workbench

Choose the Mechanical Design -> Sheet Metal Design option from the Start menu.



The Sheet Metal toolbar appears and is ready to use

Defining Sheet Metal Parameters

Click the Sheet Metal Settings icon



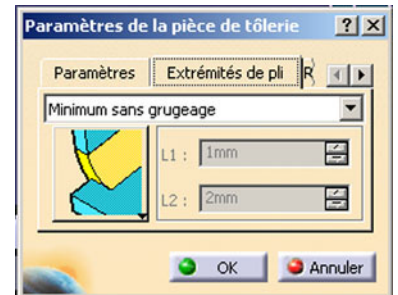
The Sheet Metal Part Parameters dialog box appears.

Come in **1.8mm** for the Thickness parameter.

Come in **2.5mm** in the Bend Radius box.

Select the Bend Ends tab.

Select **minimum without relief** in the Bend Ends list. Click OK



Click the Sketch icon

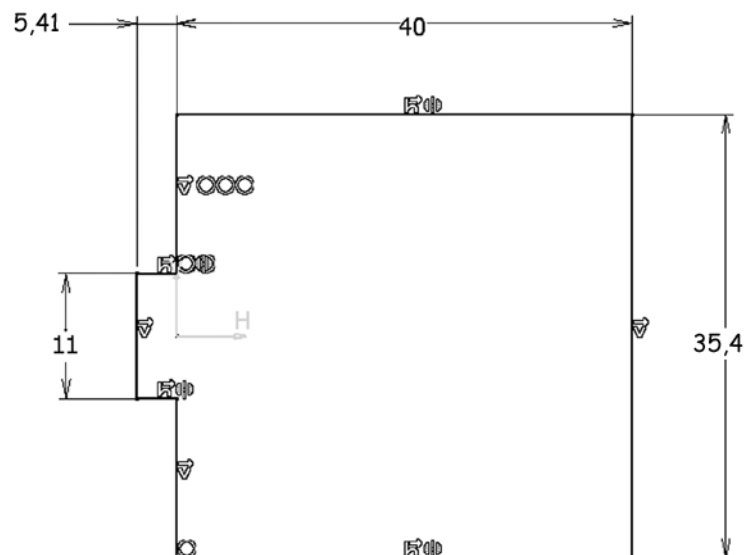


and select the **xy-plane**.

Select the Outline icon



Sketch the outline as shown opposite:



fully constrain the part (symmetry and origin)

Click on the Exit sketch icon to return to the 3D universe.



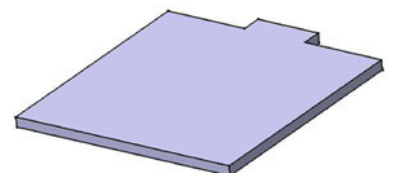
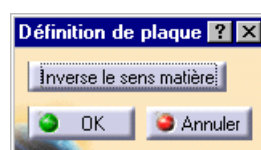
to

Click on the Plate icon



The Plate Definition dialog opens.

Click OK.



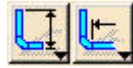
The Plate.1 component is added to the specification tree **Creation of the bend at the end of the plate**

Click the icon **Plate on edge**.



Select the edge marked with the arrow.
The Plate Definition dialog opens.

set position parameters to



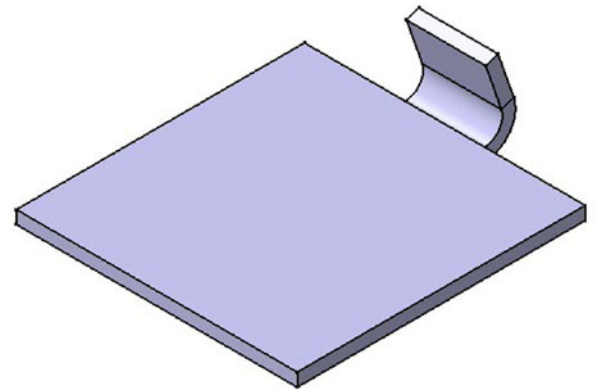
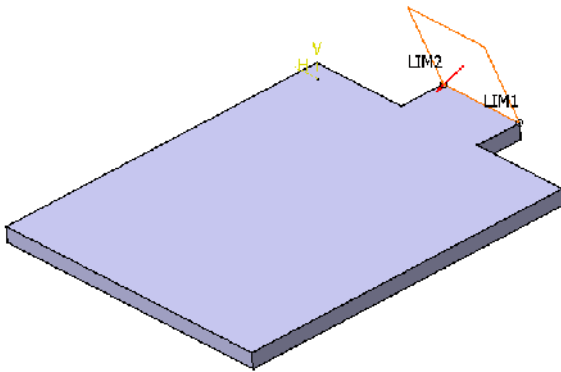
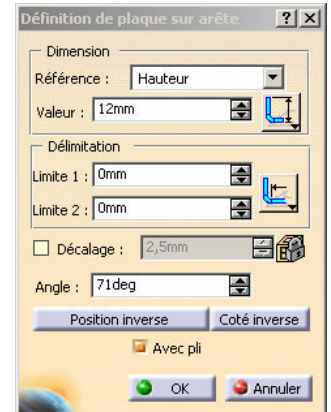
Come in **12mm** in the height area.

Come in **71 degrees** in the Angle area,

check the box **with fold**

adjust the direction of the material as in the figure (red arrow inwards) by clicking on **reverse side**

adjust, if necessary, the contour as shown in the figure by clicking on **reverse position**. The app allows you to preview the plate.

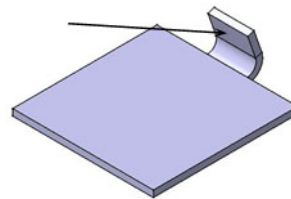


Click OK.


The first bend is created.

Creating a cutout

open a sketch on the marked face



draw the outline opposite

Click on the Exit icon sketches the  to come back in 3D universe.

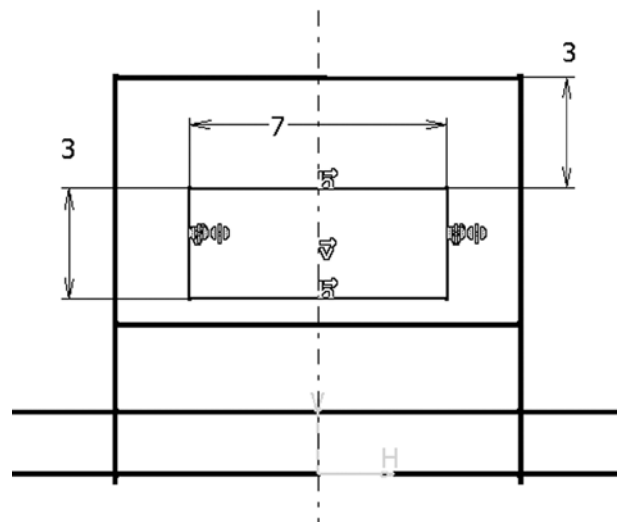
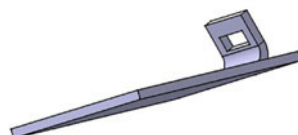
Select the Cutout icon



Set Type to **Until next** to define the limit of the cut.

Click OK.

the cutout is created



Creation of the end fold

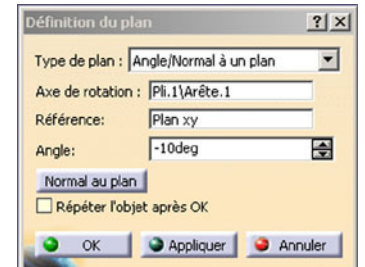
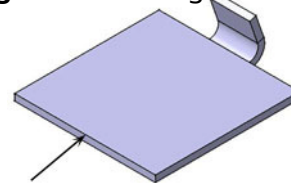
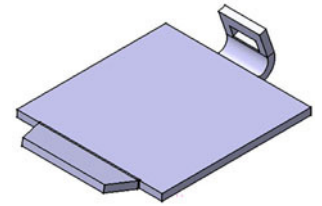
you must create a plane oriented at 10° to build the sketch

select the icon The dialog box **outline definition** opens

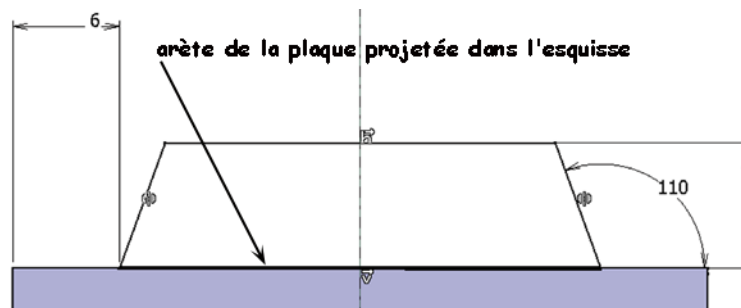
set type to **Angle/Normal to plane**

click on the edge marked with an arrow to define the **rotation axis**

choose the **xy-plane** as reference plane enter **-10 degrees** in the angles area click OK



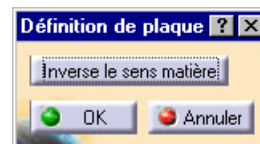
select the plane you just created then click on  to edit the next sketch



Click on the Plate icon 




The Plate Definition dialog opens.

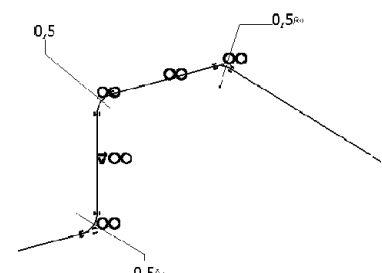
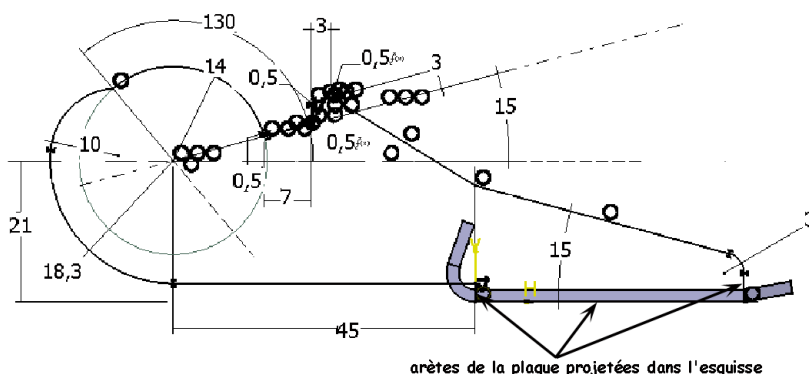
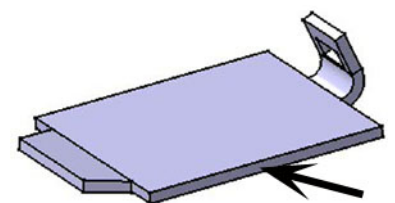


. pay attention to the meaning of the material

Click OK.

Creation of the side plates

select the face marked with an arrow then click on  to edit the next sketch



Click the Exit Sketch icon

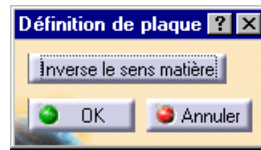


to return to the 3D universe.

Click on the Plate icon



The Plate Definition dialog opens.



pay attention to the direction of the material (towards the outside of the plate) Click on OK.

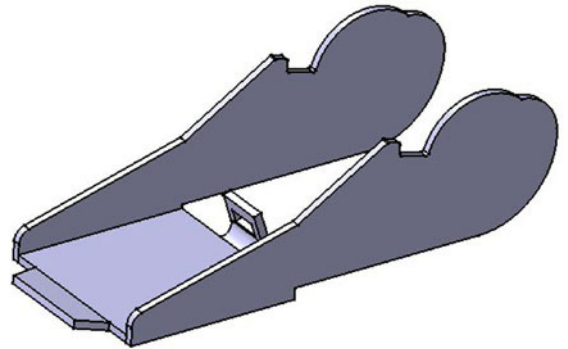
copy the sketch just created

select the other face then paste the copied sketch

Click the Plate icon.



pay attention to the direction of the material (towards the outside of the plate) Click on OK.



Creation of automatic folds

Click the Auto Folds icon



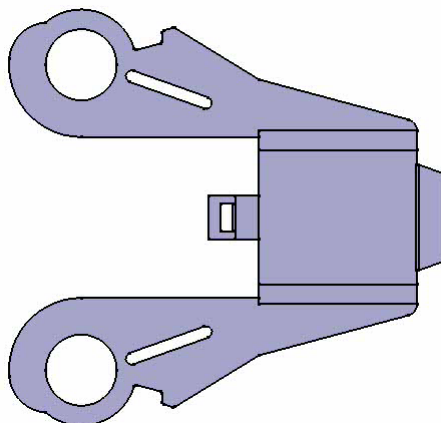
Folds are created. make a cut out of the $\varnothing 18$ make a cut out of the light

Part unfolding

click on the unfold icon



The part is unfolded according to the reference plate plane as shown below.



click again on this icon



to fold the room

the room is finished