Reel

Purpose : Use the SHELL function

We are going to create a hollow part that is used to rewind the wire of a kite. It is a half reel of ergonomic handle in plastic material.
We will make a shape of revolution on 180°, the shell, reinforcements, spoils, fillets and finally holes.

Here is the final result (fig.1):

In what follows, the "Horizontal Plane" is the base plane parallel to the (X,Y) plane of the coordinate system.

Create a new document of type "**CATPart**".

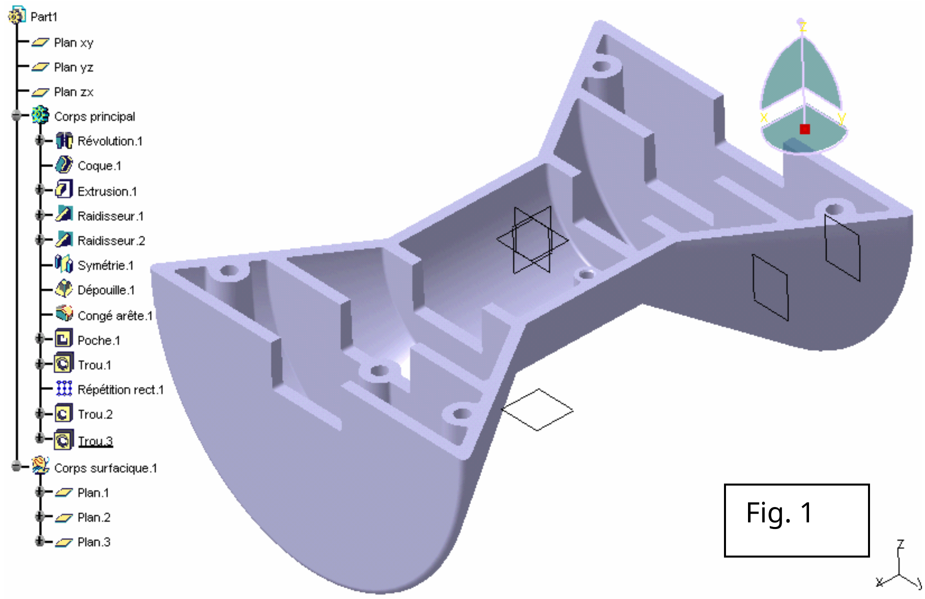


Fig. 1

1° Creation of the half revolution:

Select a (horizontal) plane and create the following profile with its dimensions and constraints (fig. 2):

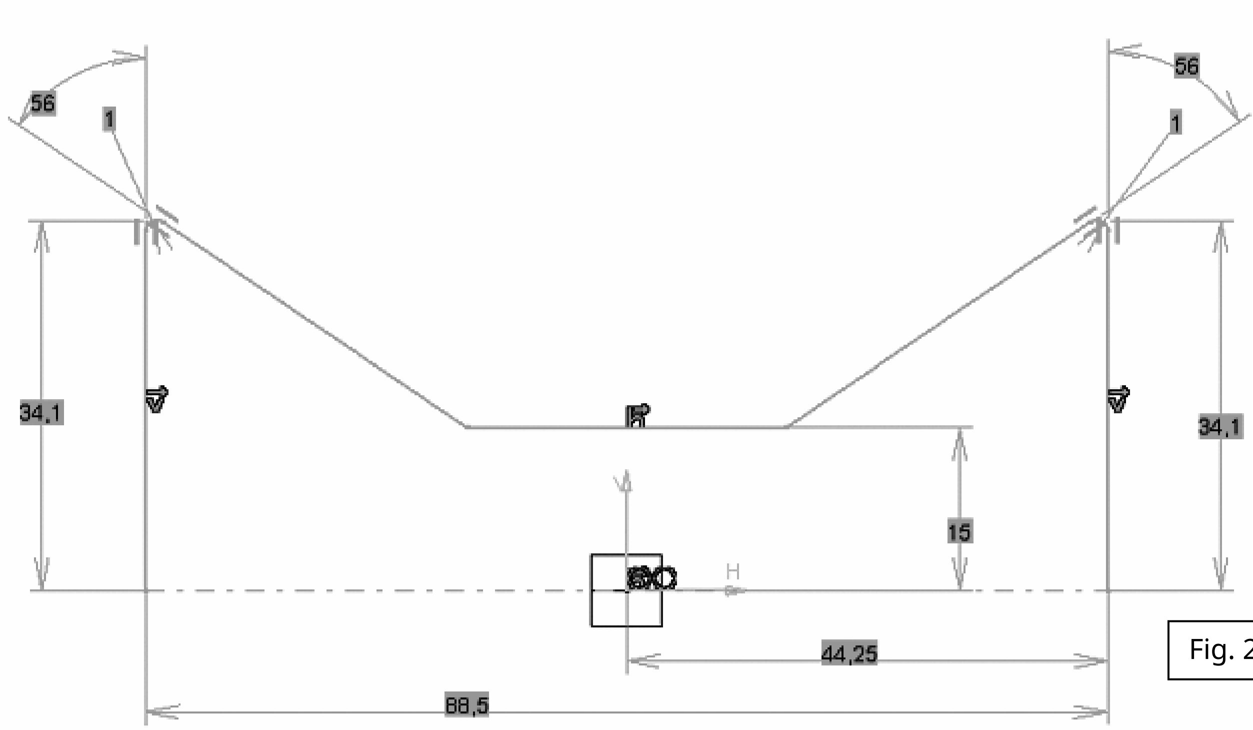


Fig. 2

Remark : it is necessary to designate a line as axis to be able to use the 3D function "Revolution" except if the sketch has a single axis (axis of sktech) which is the case in figure 16.

Make a “Revolution” through 180° (fig. 3 and 4):

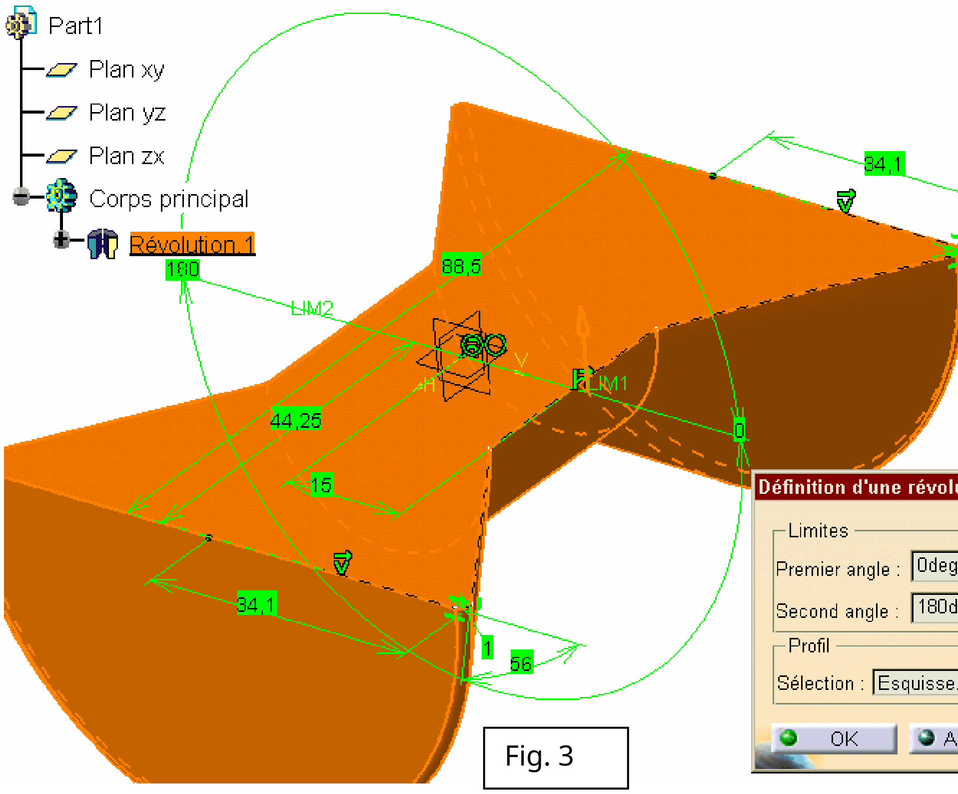


Fig. 3

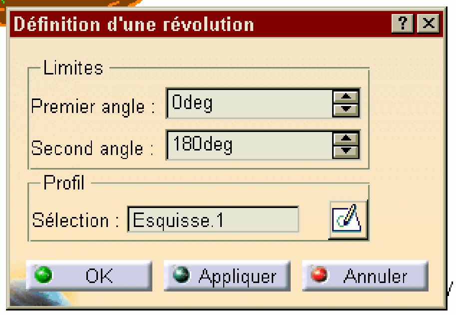


Fig. 4

2° Creation of the shell:

Click on the tool**Shell**(fig.5):

The following dialog box opens (fig.6):

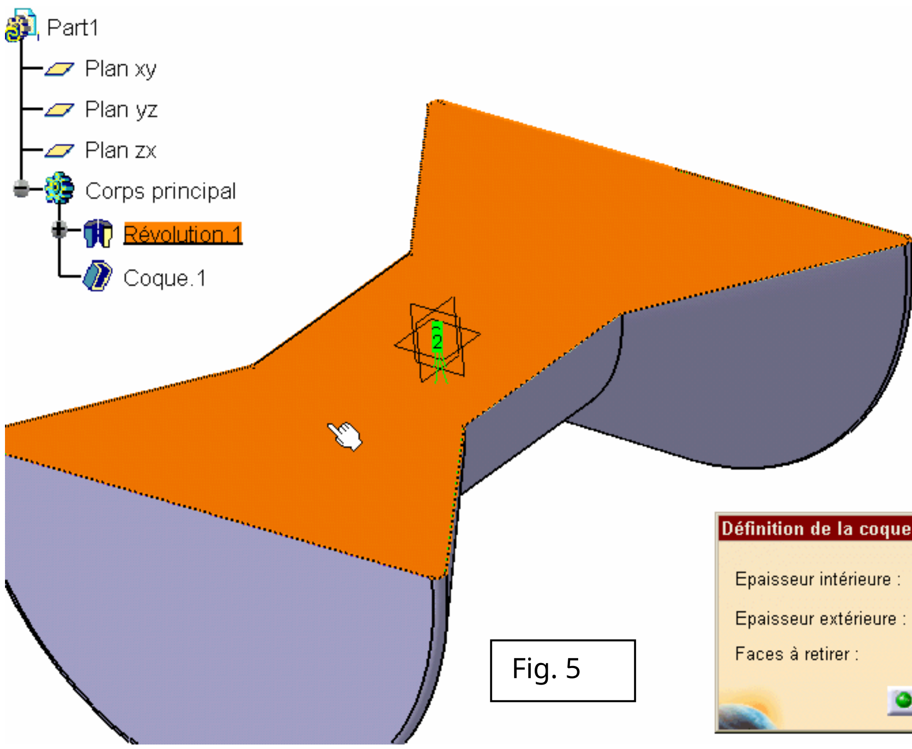


Fig. 5

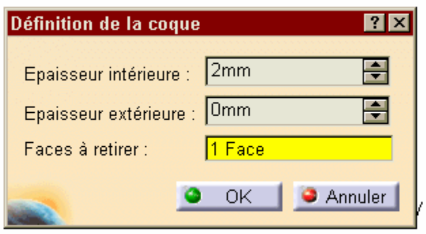


Fig. 6

The inner thickness is 2mm and the outer thickness is 0mm: this means that the current outer surface of the part will serve as the outer boundary of the part produced by the shell feature. If we invert the 2 thicknesses above, the part produced by the shell feature will be larger because the current exterior surface of the part will serve as the interior boundary.

Click on surface<u>plane</u> to indicate that this face will be open.

Click on “OK”, here is the result (fig. 7):

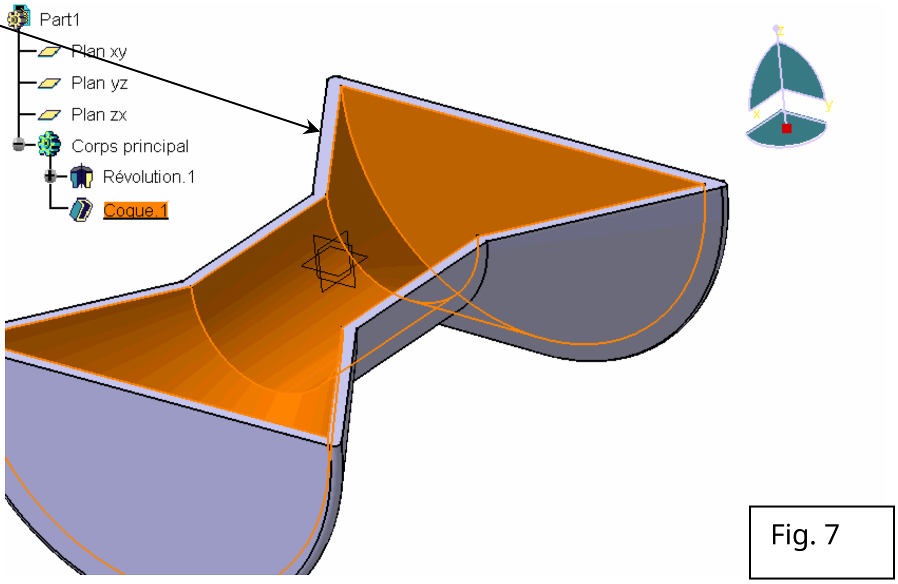


Fig. 7

3° Multiple extrusion:

Make a multiple extrusion to make the different bosses and then make holes in them.

Select the horizontal plane, activate the sketch workbench and draw the profile of the following figure (fig. 8).
Note: the line at the edge of the part is a projection 3D of the part edge and has been trimmed (eraser). The dimensions are not important for the moment, the important thing is to make sure that the 2 straight lines are perpendicular to the external edge of the part and tangent to the arc of a circle. As always for an extrusion the profile is**firm**.

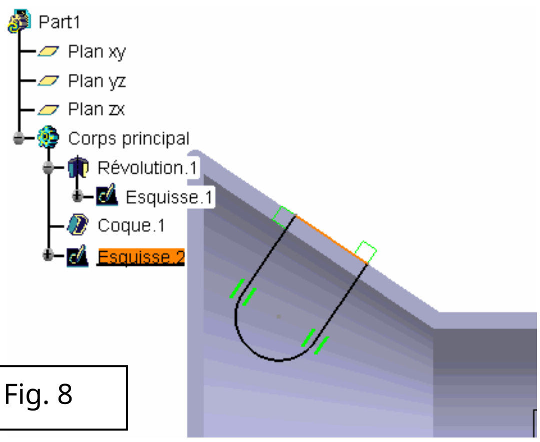


Fig. 8

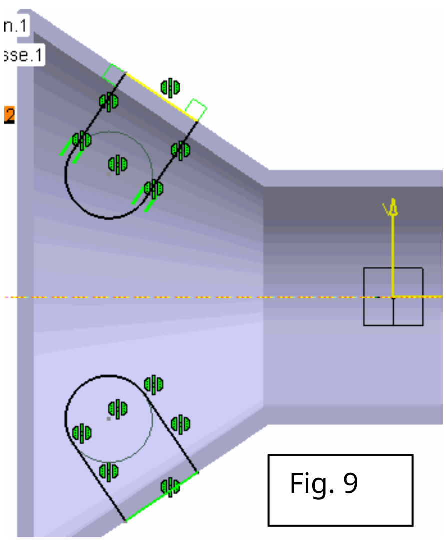


Fig. 9

Duplicate this profile by symmetry: select the profile by making a window around it and starting outside the part or an automatic search (right click on an entity of the profile) then click on the "Symmetry" icon then select the axis H as axis of symmetry (fig. 9):

Catia accepts multiple profiles in a sketch provided

whether they are<u>all closed</u> . Check: Tools, Sketch Analysis 2D  of the Analysis toolbar or drop-down menu.

Repeat the operation: select the 2 profiles by making a window around them then click on the icon **Symmetry**  then select the vertical plane (or the V axis) as the axis of symmetry (fig.10):

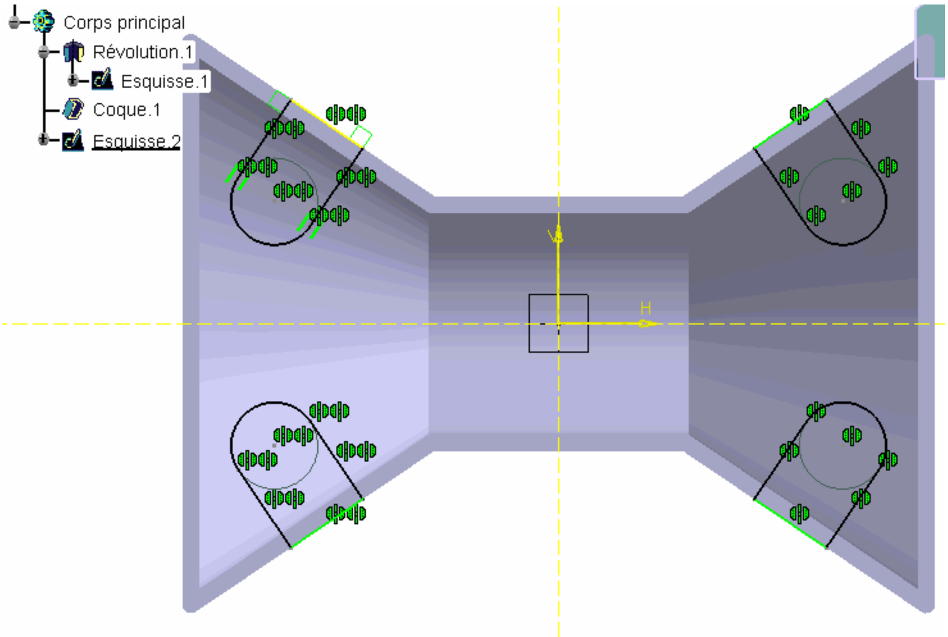


Fig. 10

It's time to compel the 1er profile: dimension the position of the center of the arc of a circle with respect to the horizontal axis of symmetry (25.17mm) and (36.35mm) with respect to the vertical axis. Dimension the radius of the arc (3mm) (fig.11):

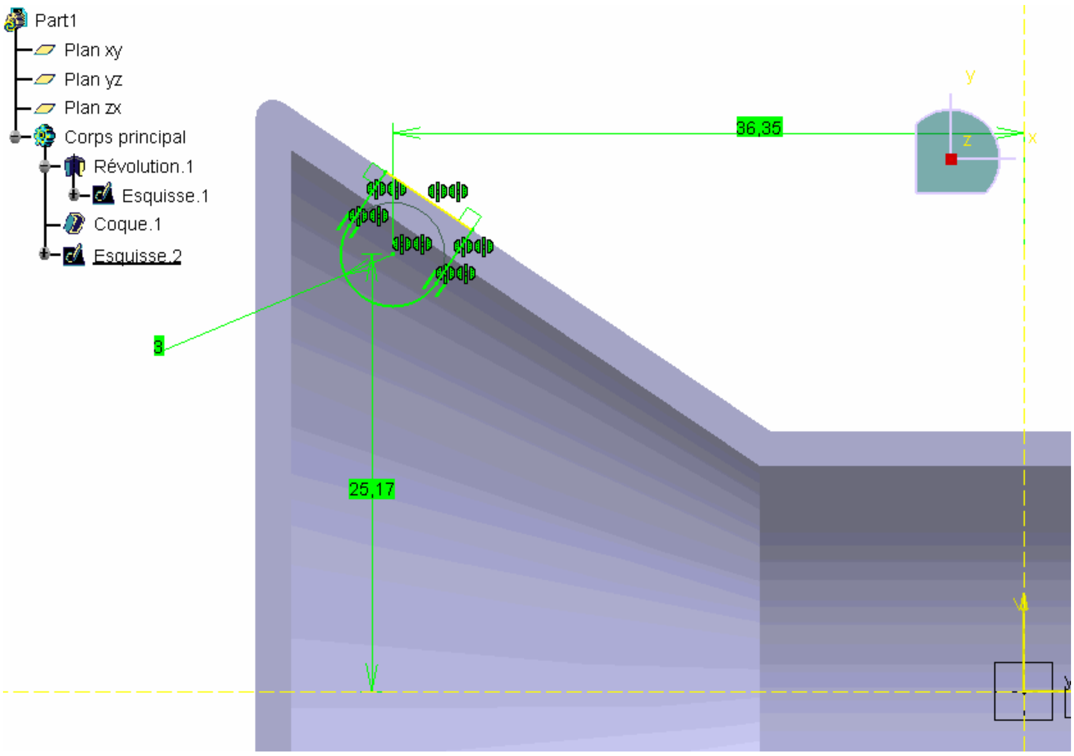


Fig. 11

In principle, the other 3 profiles readjusted automatically (due to symmetry constraints).

Now place a circle with a diameter of 6mm aligned with the horizontal plane. Dimension the position of the center of the circle in relation to the left edge of the part (74.25mm) (fig. 13):

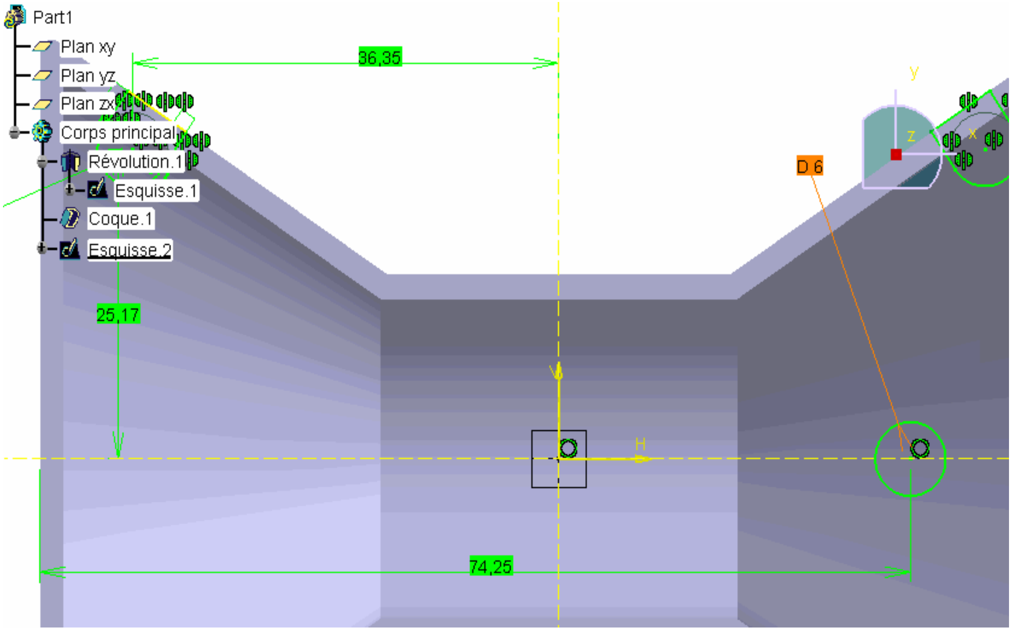


Fig. 13

Place another circle with a diameter of 4mm aligned with the vertical plane and whose center is 12mm from the horizontal plane (fig. 14):

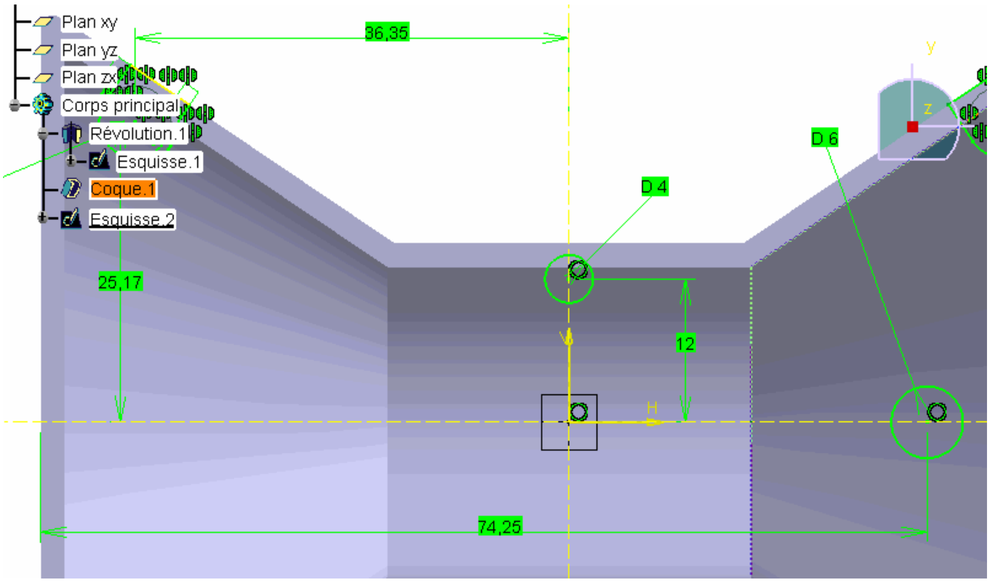


Fig. 14

Close the sketch workbench then click on the "Extrusion" function and choose as the type of limit: "Up to the next". The direction of the extrusion is downwards, if this is not the case click on the "Reverse direction" button or on the orange arrow in the center of the view. Click on "OK" (fig. 15):

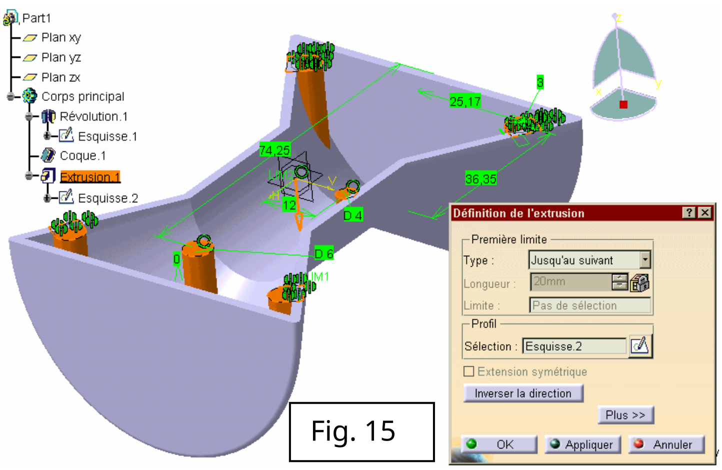


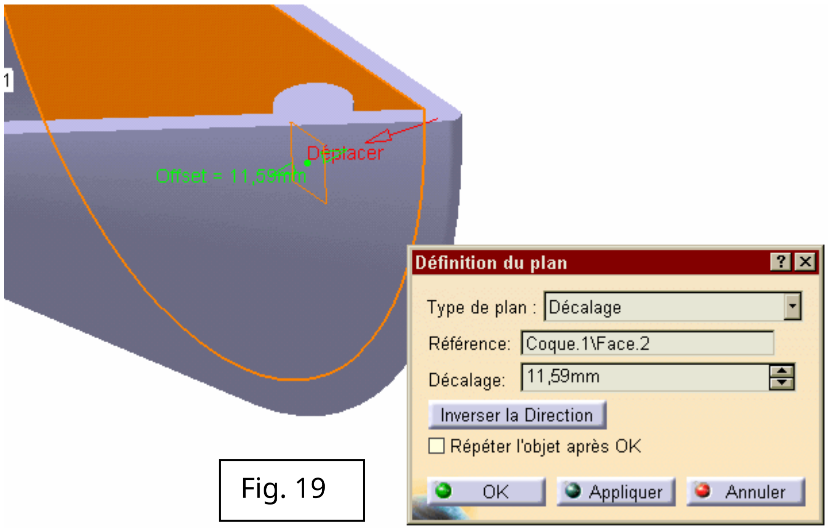
Fig. 15

4° Creation of stiffeners:

Create stiffeners some distance from the end of the part. To draw the profile of the stiffener in the right place, we will need a different plan from the existing ones: indeed the plan used for the sketch of the stiffener will give the orientation of the stiffener itself!

It must therefore be chosen or created according to this: Create an offset plane of 11.59mm d**and the internal face**of the room as in the following figure (fig.18): (the new plane is towards the inside of the room)

Fig. 18



Click on "**okay**".

Fig. 19

Repeat this by creating a second plane offset by 14.06mm from the first plane, always towards the middle of the room. This second plane will be used for a second stiffener.
Click on "**okay**» (fig.19):

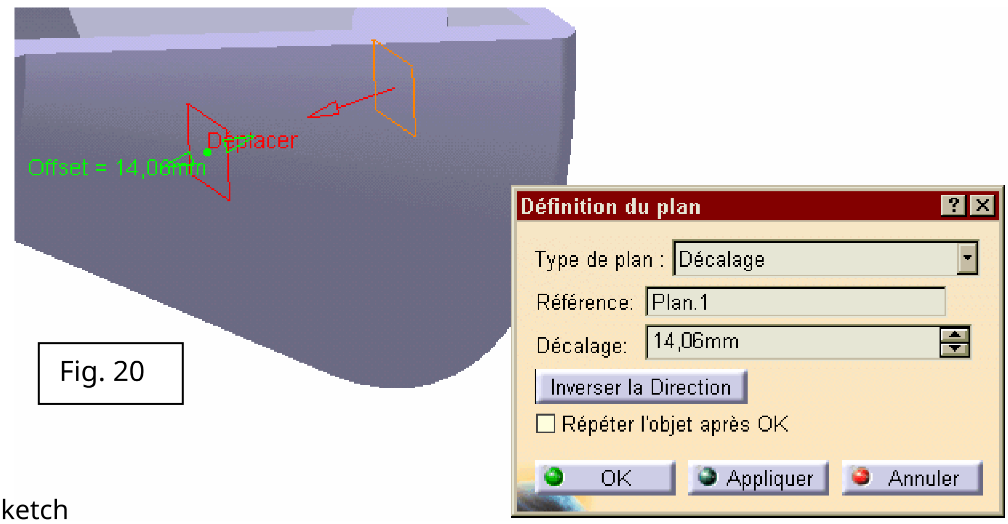


Fig. 20

Select 1erplane and activate the sketch workbench. Draw a deliberately short line segment that will be horizontal and aligned with the horizontal plane (fig.20):

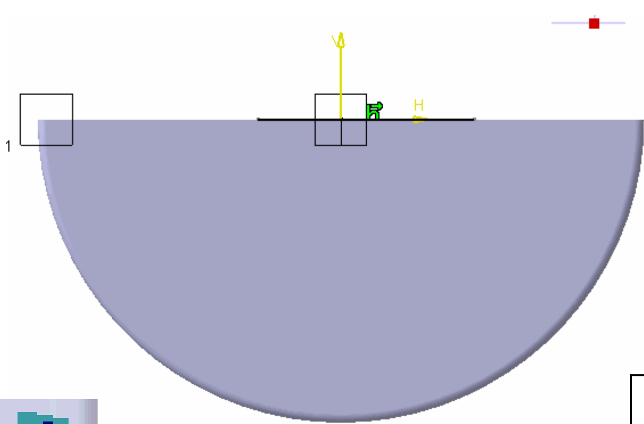
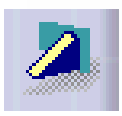


Fig. 21

Exit the sketch studio
and click on the “Stiffener” icon  (fig.22):

From the line drawn in the sketch, Catia automatically extends it until it meets material laterally and creates an adjustable material thickness in the dialog. Indicate a thickness of 2mm then impose that this veil of material is not symmetrical and that it is directed towards the middle of the part, the direction of the depth being directed towards the part (fig.23):

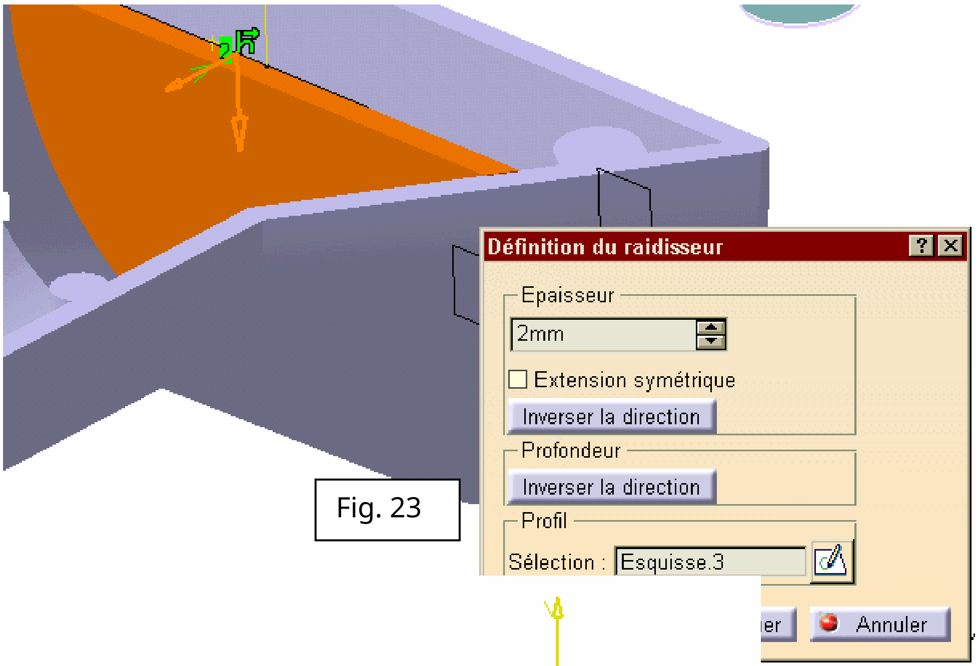


Fig. 23

Repeat what has just been done from the second plane created: new sketch (fig.24):

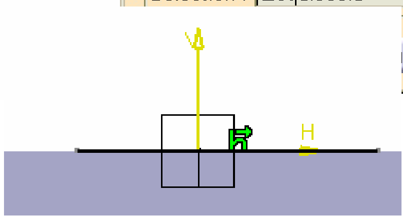


Fig. 24

Then new stiffener 2mm thick, non-symmetrical and directed towards the middle of the part (fig.25):

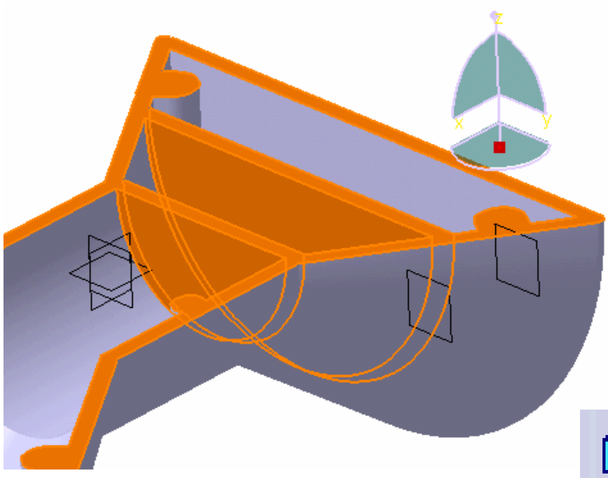
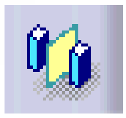


Fig. 25

Make a copy by symmetry of these 2 stiffeners: select the stiffener in the tree structure **Stiffener.1**and holding the key**CTRL** pressed, select the stiffener**Stiffener.2**. Now click on the icon**Symmetry**(fig.26):



<u>Remark</u> : if no 3D function of the tree structure is selected before calling the “symmetry” function, the entire part will be duplicated by symmetry!

Select as plane of symmetry the central plane parallel to the plane (Y, Z), then click on "**okay**".Here is the result (fig.27):

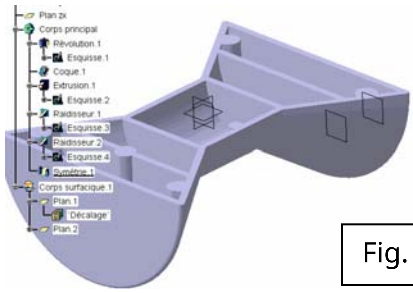


Fig. 27

5° Spoils:

Create a draft of 1° on the internal vertical faces of the part. Click on the icon**Bare**and select the 19 faces (planar or vertical cylindrical). Then click in the box**Neutral element**: **Selection**then click on the<u>horizontal plane</u> . Click on "**okay**» (fig.28):

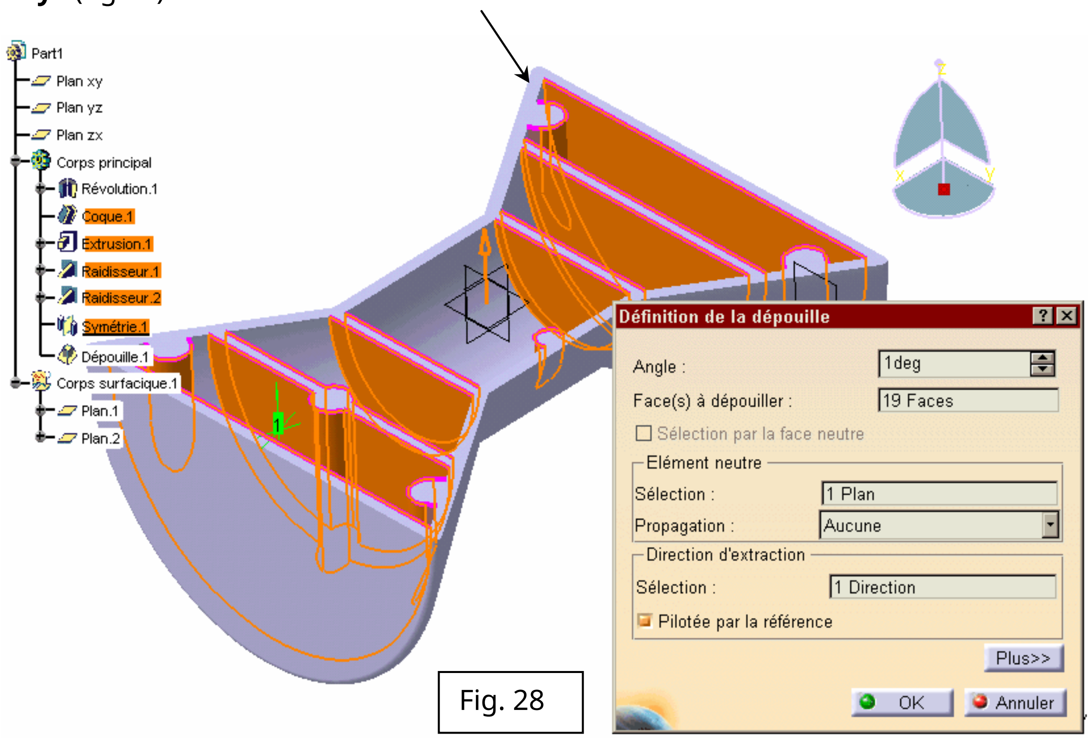


Fig. 28

Place 0.5mm fillets on all edges of the part except the edges of the top flat face. Click on the icon**Leave** and select the 5 inner cylindrical faces of the part. And the 4 edges: BE CAREFUL to select only the<u>4 edges</u> and not faces! Click on "**okay**» (fig.29):

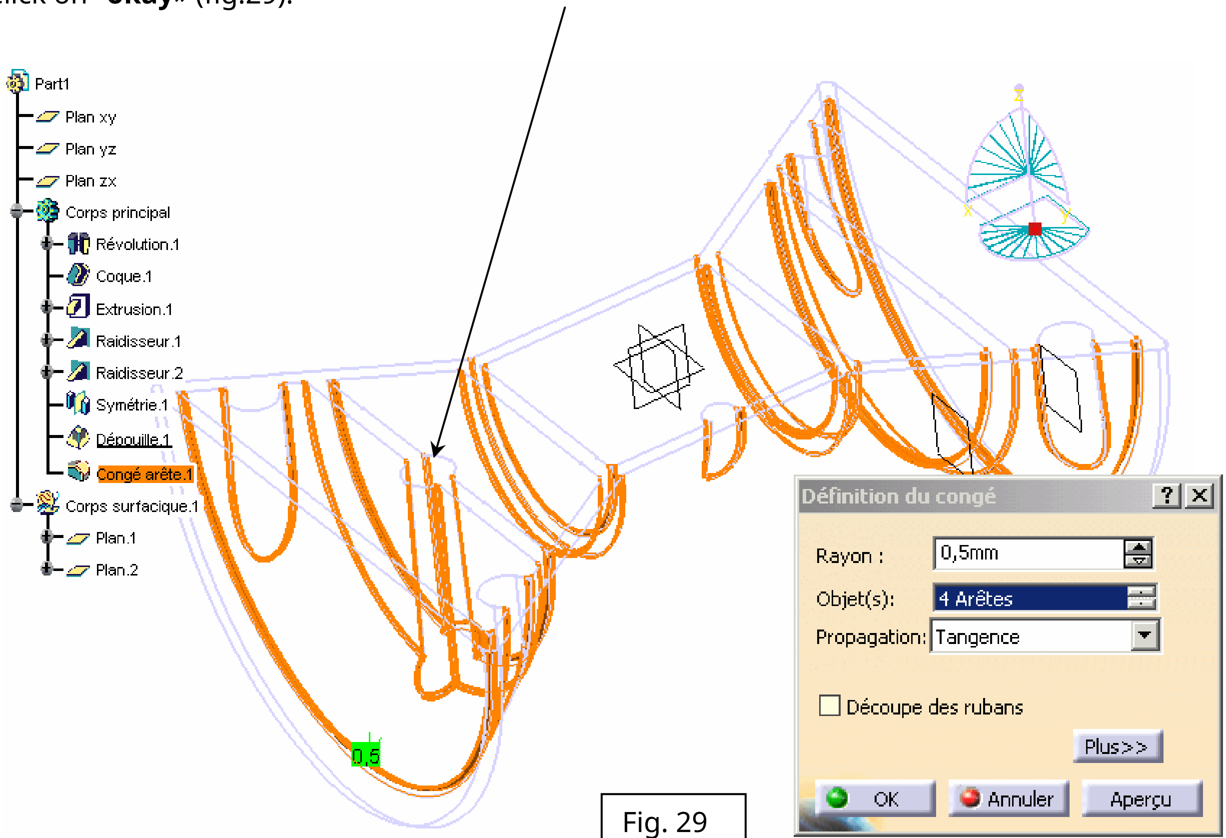


Fig. 29

6° Pocket:

Make a pocket: select the middle plane which is parallel to the (Y,Z) plane and open the sketch workbench, then draw the**open profile**following with its constraints (fig.30):

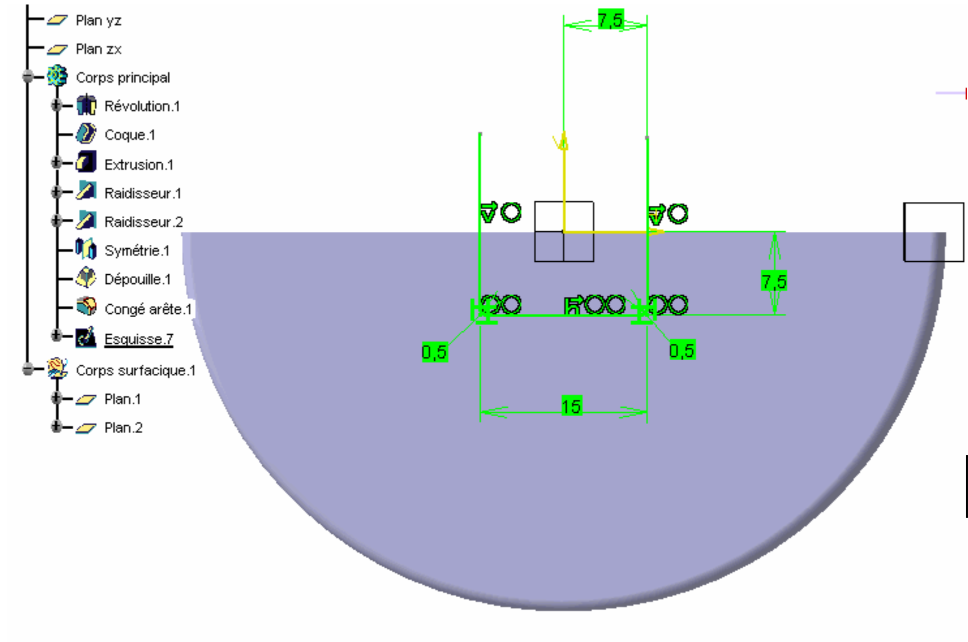


Fig. 30

Close the sketch workshop then click on the icon**Poached**. In the dialog box click on the button **More>>**so that all the parameters appear. Choose as limit type**Until the last one** for the first and the second limit. The material that will be removed from the part by the "Pocket" function can be either "inside" or "outside" our profile. This is chosen with the "Reverse direction" button. Choose the “inner” side as shown in the following figure (fig.31):

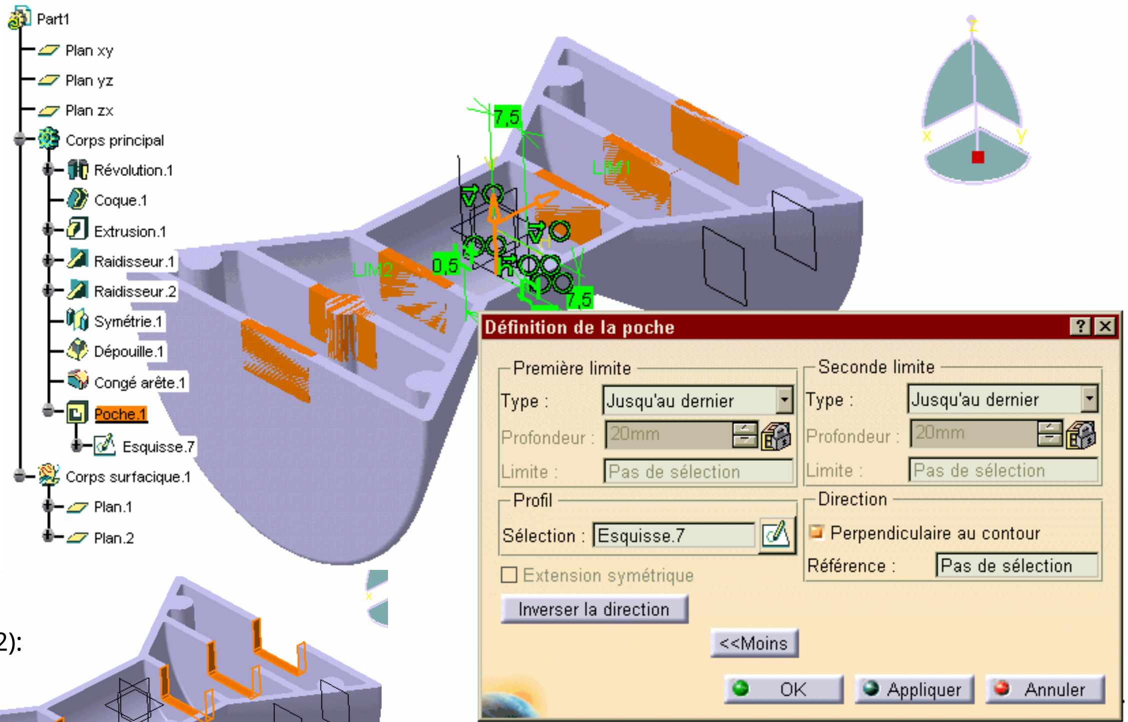


Fig. 31

Here is the result (fig.32):

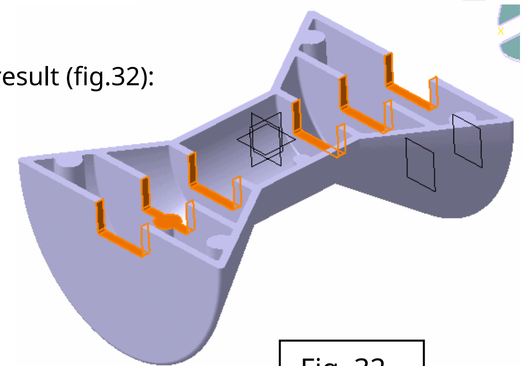
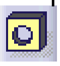
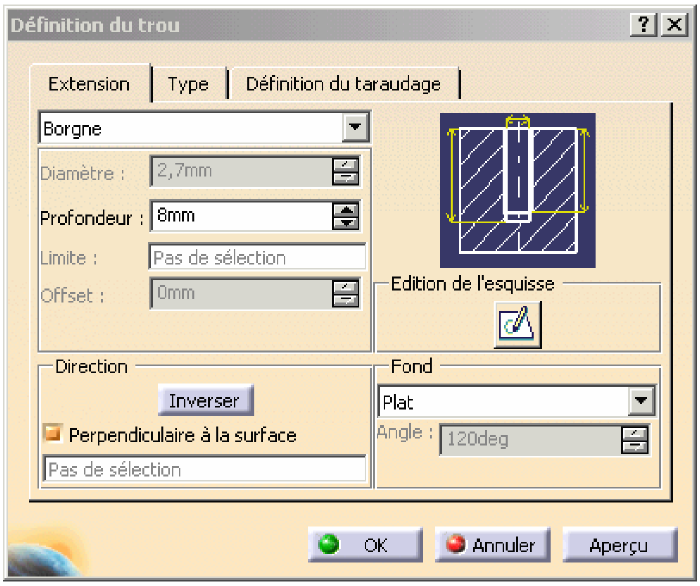


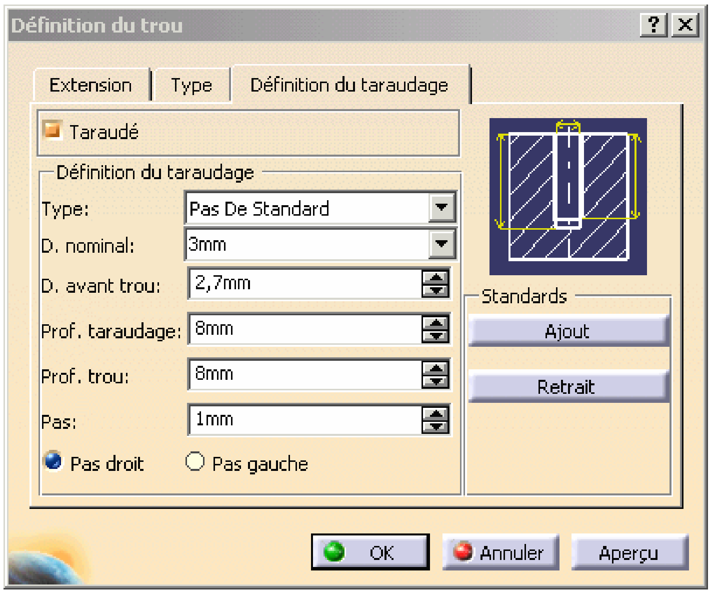
Fig. 32

7° Drillings:

Now make an M3 tapped hole: Click outside the part to cancel any selection. To click on the icon**Hole**  then on the upper surface of the leftmost boss of the preceding figure. The position of the center is arbitrary for the moment but this will be rectified later.

In the dialog box, choose a hole**one-eyed**whose diameter is 3mm and 8mm deep. If Catia displays an error message, you will surely have to reverse the drilling direction using the button **Reverse**. See the following figures (fig.33 and 33 bis):





Then click on the thread definition tab and check the box**Tapped**. The drilling diameter automatically adjusts to the value of 2.7mm. Indicate in the box**Teacher. tapping**a depth of 7.5mm.

Click on tab**extension**to return to this dialog box then on the button**Sketch editing** to place our hole with all the necessary precision. See the following figure (fig.34): select the point (it represents the center of the drilling) and holding the key **CTRL**on the keyboard, select the circular edge of the boss; click on the icon**Constraints**, check the box **Concentricity**then click**okay**. Exit the sketch.
Click on**okay**
Drilling is complete.

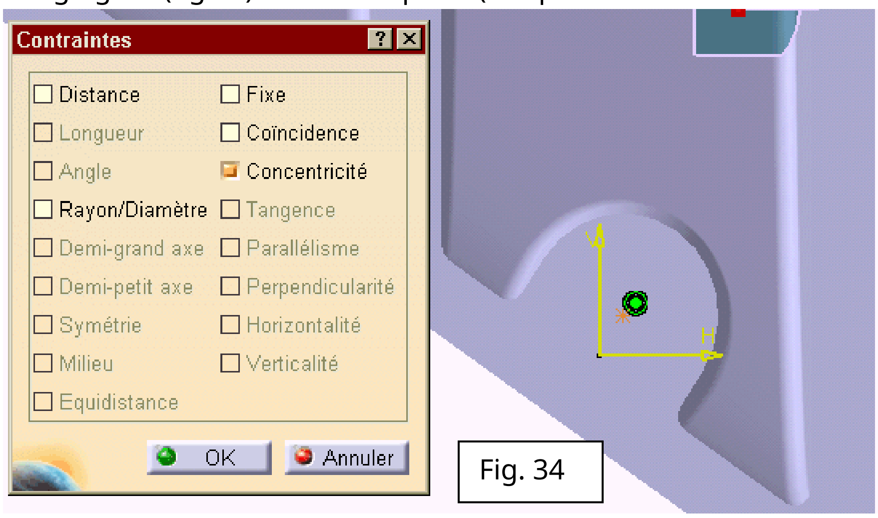
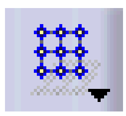


Fig. 34

Duplicate this hole on the other three bosses: Click on the icon**Rectangular repeat**(fig.35): 

Click on the hole that has just been made, a dialog box opens.
Indicate 2 instances, a spacing of 72.7mm then click in the box “Reference direction: Reference element” then click on the central horizontal plane parallel to the plane (X,Y). Click on the “Second direction” tab, indicate 2 instances and a spacing of 50.34mm. If direction 1 and direction 2 are not the same as the following figure, you can click the button**Inversion**(fig.36):

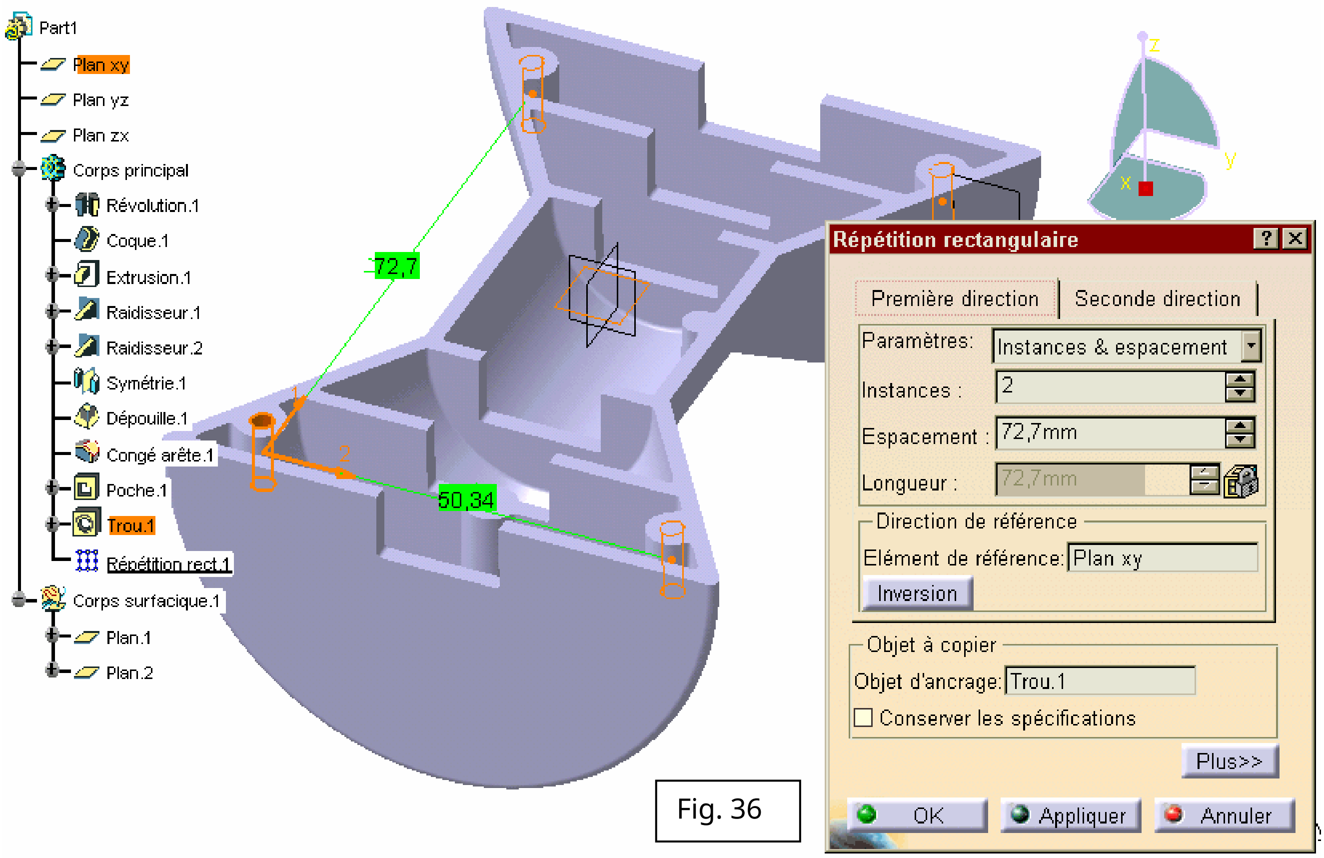


Fig. 36

Make a new hole on the center boss this time.

Click on the icon**Drilling**then on the upper surface of the central boss.

This hole will be**one-eyed**, with a diameter of 2mm, a depth of 3mm and a "Flat" bottom (fig.37): -->

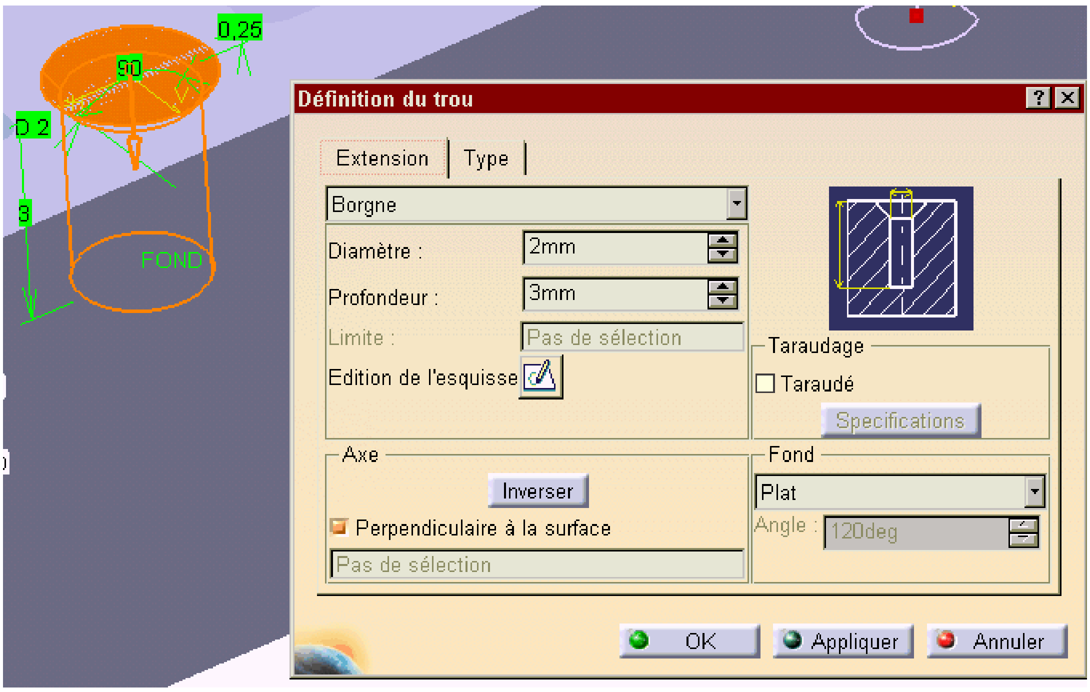
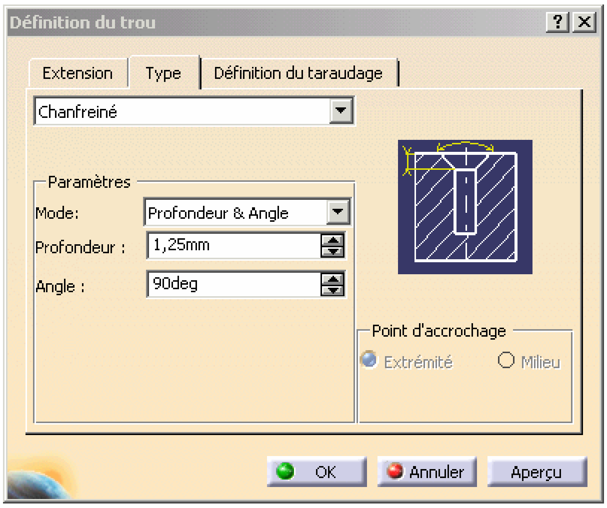


Fig. 37

Click on tab**Kind**and choose**Chamfered**, the depth is 0.25mm and the angle 90° (fig.38):-->



Go back to the tab**Extension**and click on the button **Editing the sketch**.

Fig. 38

Make the center of the hole concentric with the circular edge of the boss (fig.39):-->

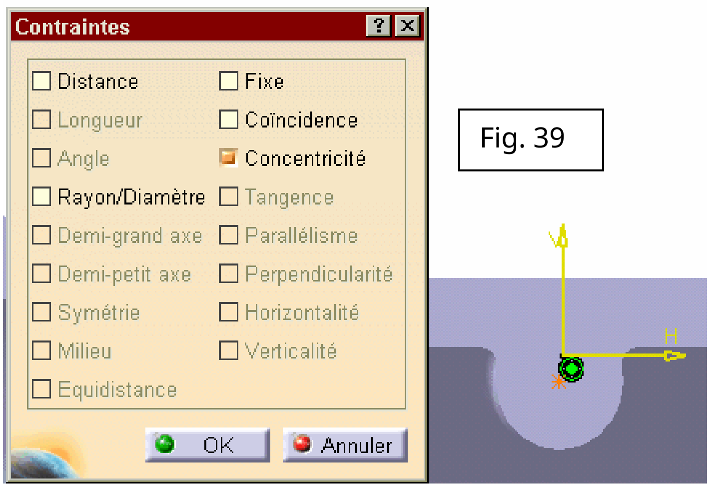


Fig. 39

Close the Sketch Workbench, the hole is complete.

There is only one hole left to complete the piece. The particularity of this last hole is that it emerges and that it has a counterbore on the outside of the part!

To make a counterbored hole, simply choose the appropriate option in the function**Drilling**. But the counterbore is perpendicular to the plane chosen in the drilling function: here there is no flat surface to start drilling from the lower side of the part.

You must therefore create a new plan:
Click on the icon**Plan**and choose a type plan**Gap.**
Click on the horizontal plane to define the reference, the offset is 27.5mm downwards.
Click on**Reverse direction**if needed.

Click on**okay**(fig.40):-->

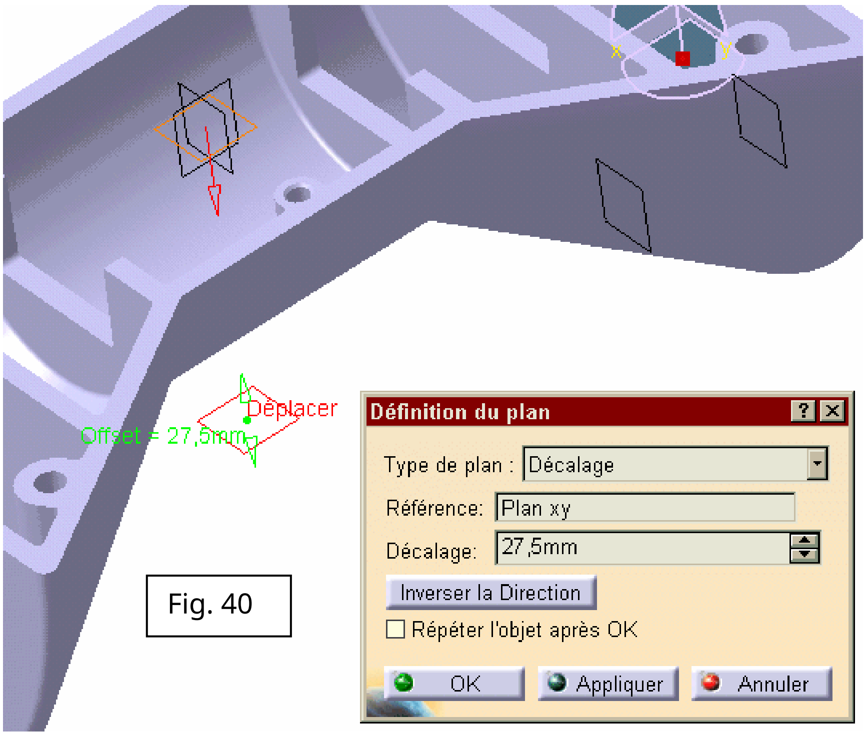


Fig. 40

Click on the icon**Drilling**then click on the plan that has just been created. Enter the parameters of the following figure: diameter M3 tapped over 10mm, the extension of the hole will**Until the last one**, the drilling direction is upwards (fig.41):

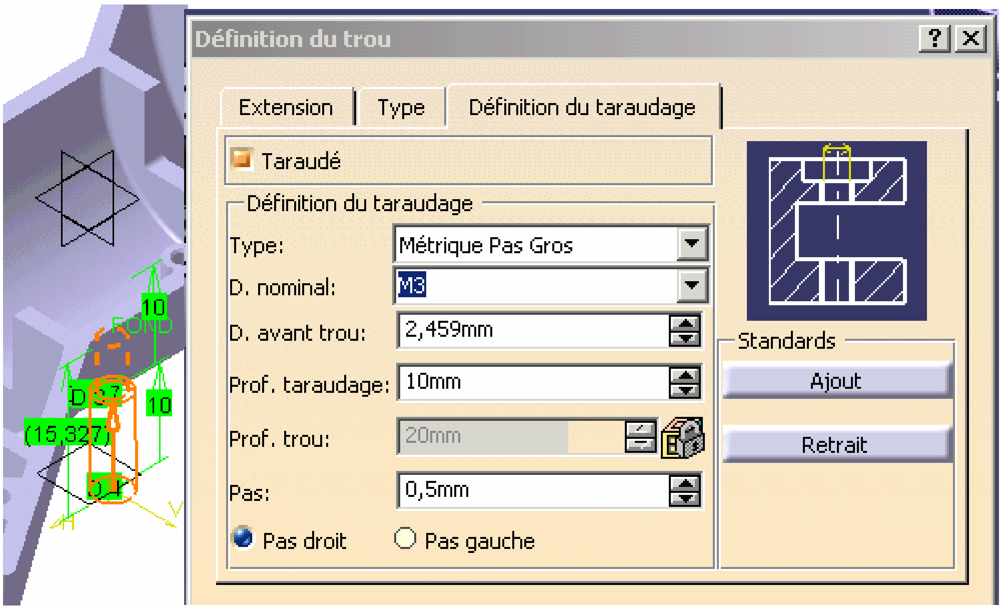


Fig. 41

The type of drilling is**Blade**:the counterbore diameter is 4mm over a depth of 10mm (fig.42):

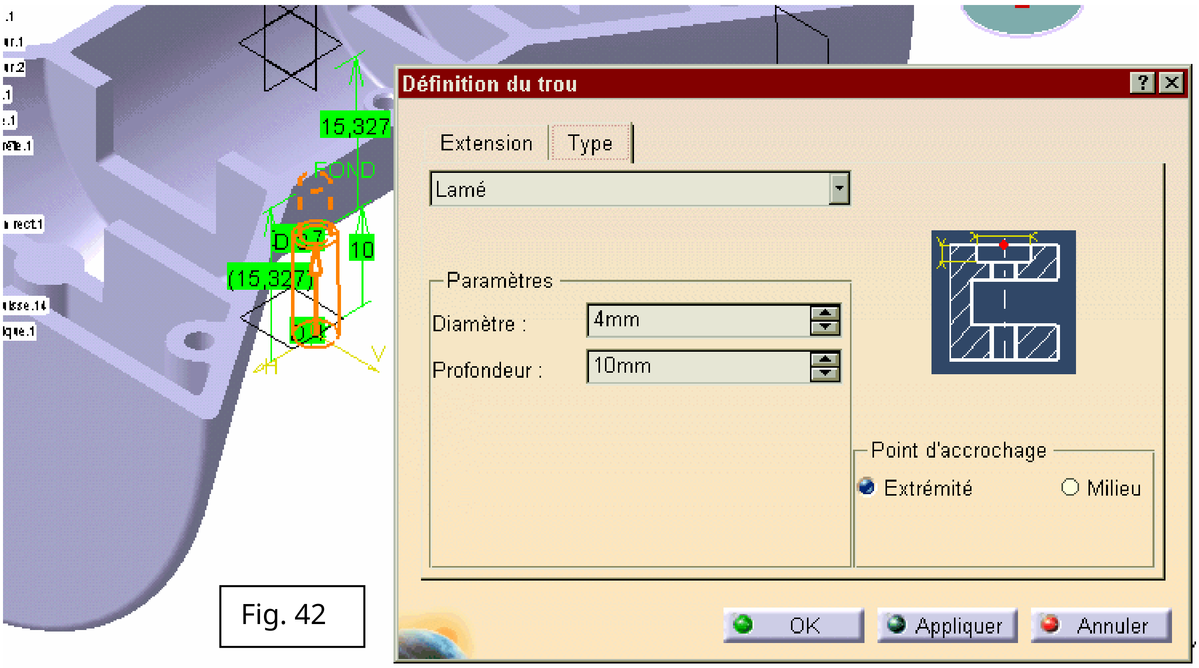


Fig. 42

Go back to the Extension tab then click on the button **Editing the sketch**.

Select the center point of the hole and a circular edge of the central boss and apply a concentricity constraint (fig. 43):

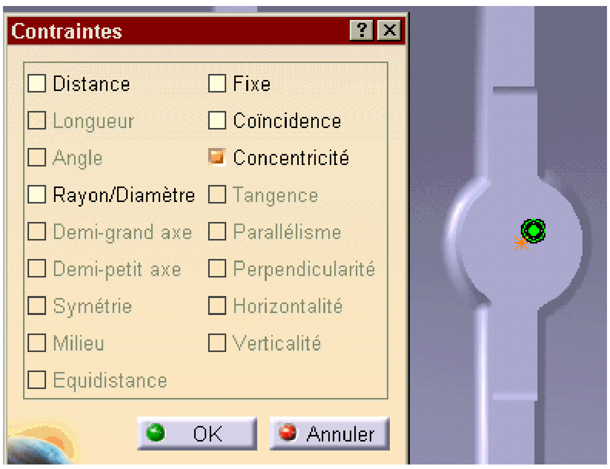


Fig. 43

Click on “OK”,**the room is finished**(fig.44):

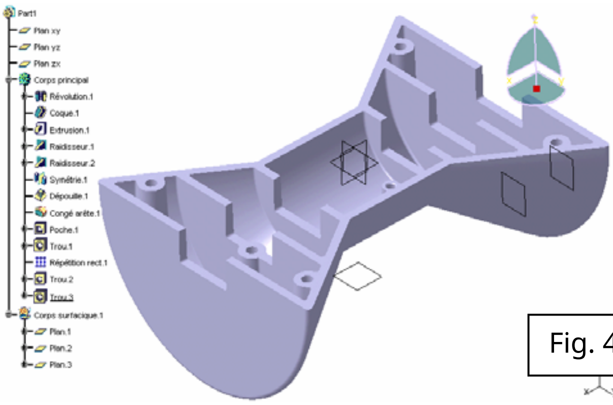


Fig. 44

REMARK : It may sometimes be preferable to make the holes BEFORE making a shell, especially if these are through holes (the previous part did not lend itself well to this).

Example : make a new piece that looks like a cube, regardless of the dimensions, make a counterbore hole, then make a shell by opening the face of the cube which is opposite to the counterbore (example fig. 45):

The boss for the passage of a screw comes from the function**Shell**.

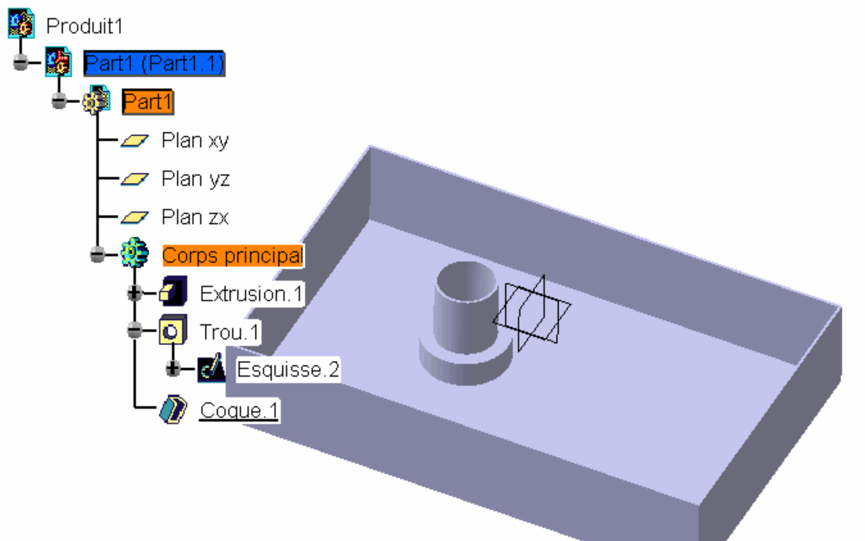


Fig. 45