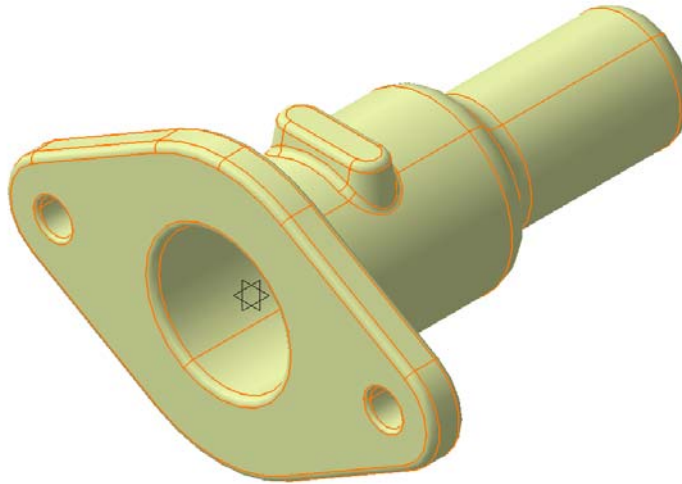


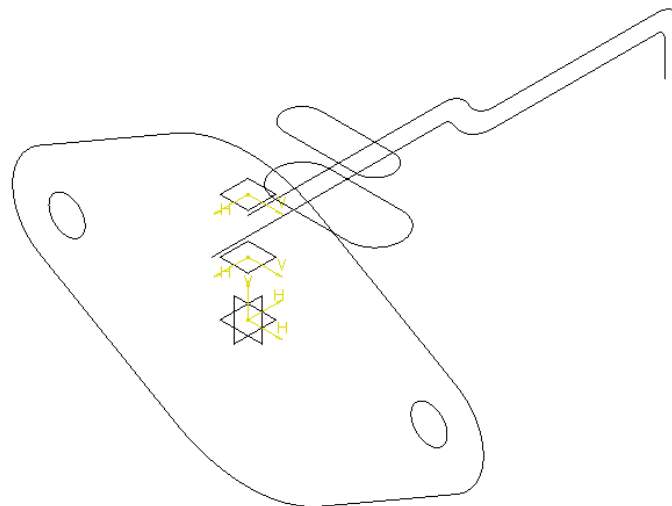
The objective of this lab is to create, using the Generative shape design workshop of CATIA V5, the master cylinder below.



Strategy implemented.

The analysis of the geometry of the object to be modelled, leads to setting up a certain number of judiciously chosen sketches, on which we will rely to generate elementary surfaces. Then using certain procedures we will cut, connect, join these surfaces to obtain the final object.

The analysis of the geometry of the master cylinder led to the use of the profiles drawn below in sketches located in different planes.



Prerequisites be able to:

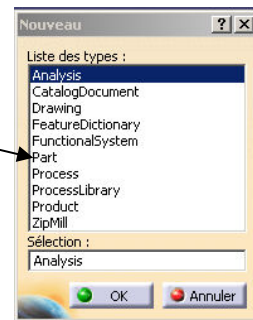
- create constrained sketches in reference planes
- create planes parallel to reference planes

This lab will allow you to learn how to:

- start the Generative shape design workbench.
- generate elementary surfaces a)
surface of revolution
b) filling area
c) extrusion surface
d) offset surface
e) guided surface
- modify the elementary surfaces using the cutting tools with assembly or not.
- assemble the surfaces using the tools:
a) connection between two surfaces or on an edge
b) joining two surfaces (Join)
- manage the tree structure in order to easily find the surface entities
It is recommended to systematically rename the entities created in order to facilitate the management of the operations of transformation of modification of junction assembly etc....

Launch CATIA V5 and open a new file

Picking out **File>New>Part**



Open the Generative Shape Design workbench

Picking out **View>Toolbars>Workbenches**

To obtain the bar opposite



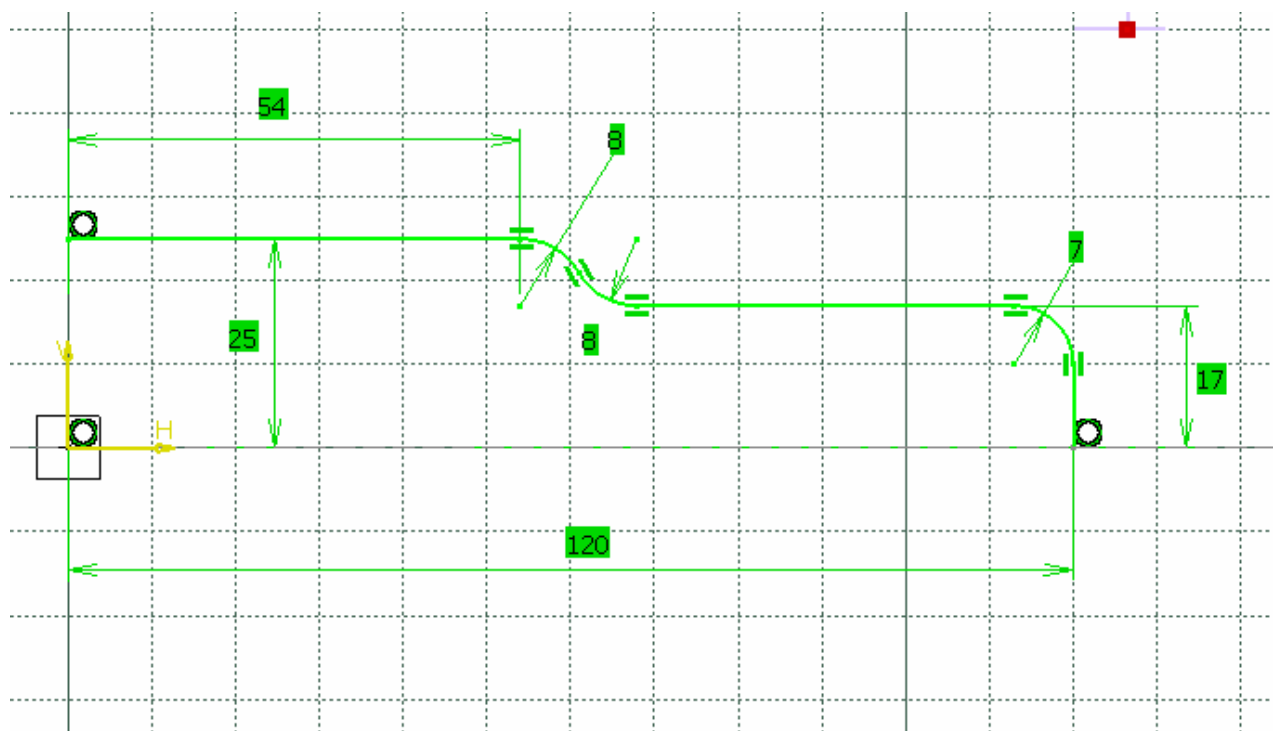
Choose icon **Generative Shape Design**

1 - Constructions of the different profiles

a) Construction of the profiles of the surfaces of revolution

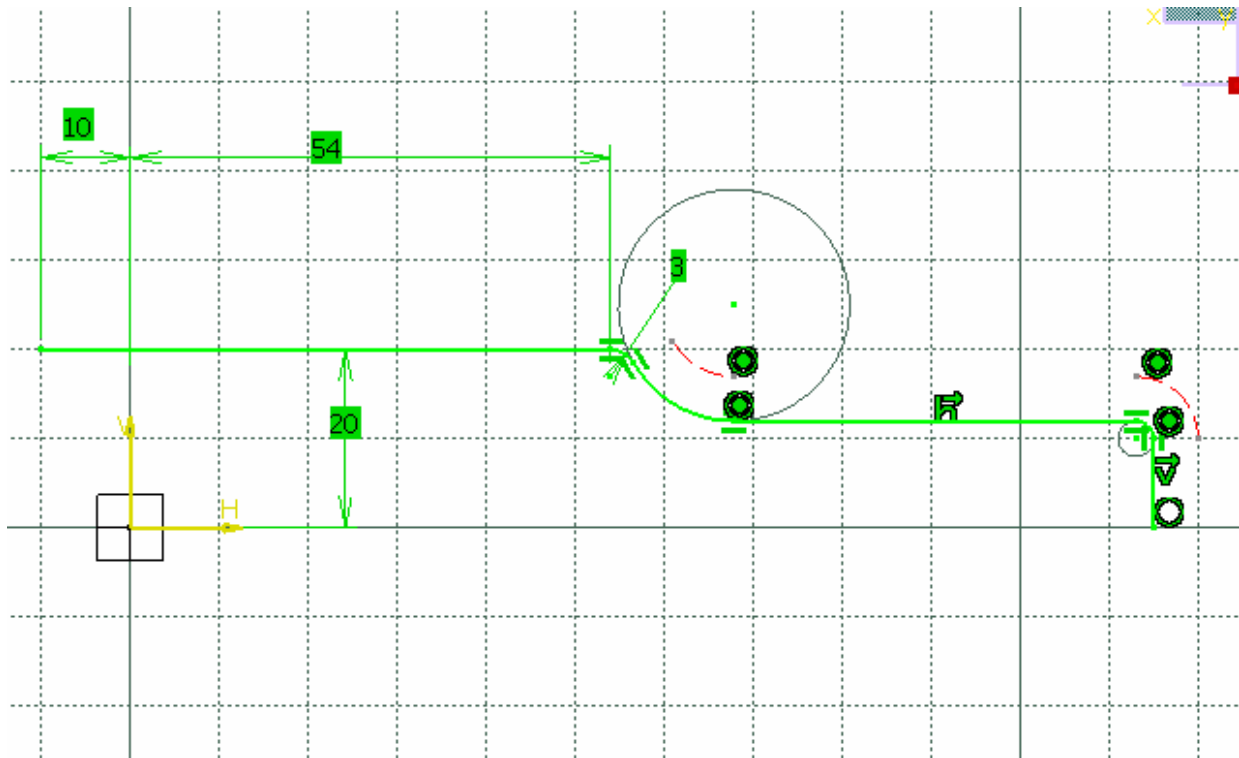
- Choose the xz plane open a sketch and build the constrained external revolved profile and close the sketch.

Rename this profile in the tree **Exterior revolution profile**



- Choose the xz plane open a sketch and build the constrained interior revolved profile and close the sketch. (this profile is shifted by 5 mm compared to the previous one)

Rename this profile in the tree **Interior revolution profile**

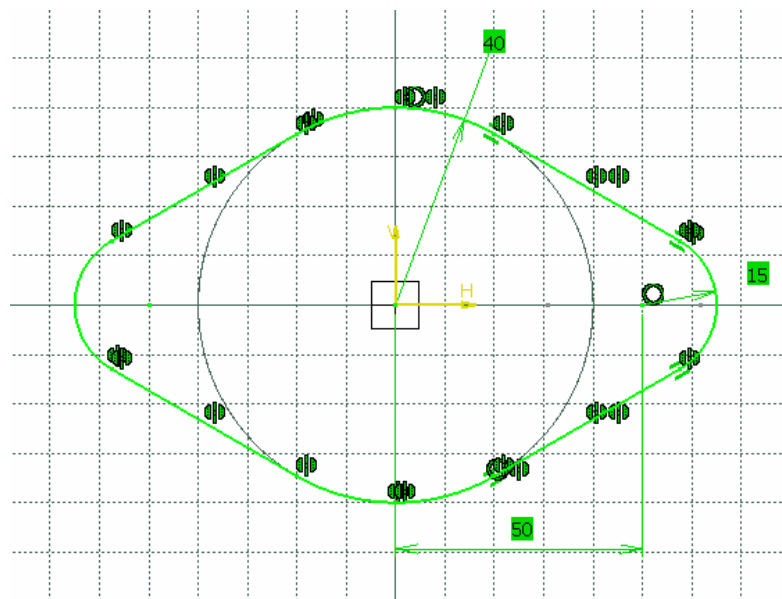


b) Construction of the base profiles

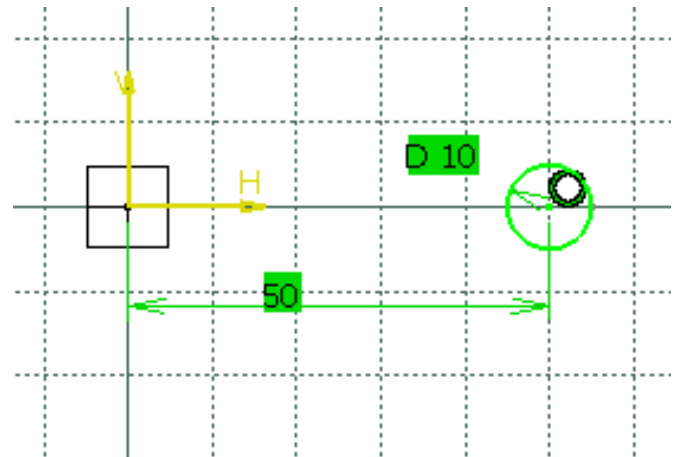
- Choose the yz plane open a sketch and construct the constrained base profile and close the sketch.

Rename this profile in the tree

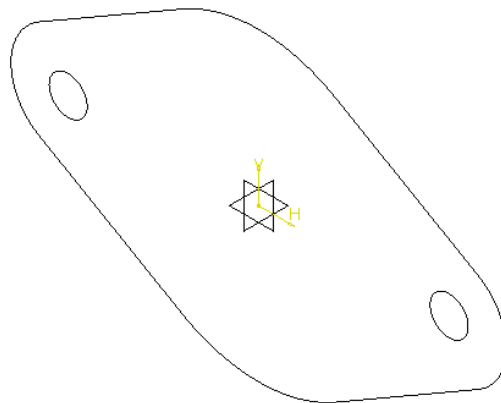
Basic profile



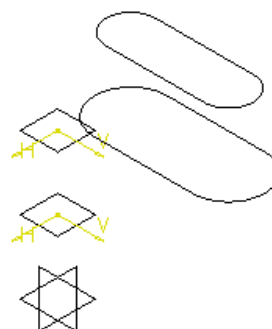
- Open a new sketch in the base plane and construct a circle with a diameter of 10 mm according to the attached definition. Close the sketch and rename the circle **Profile hole1** in the tree.
- Open a new sketch in the base plane and construct a circle with a diameter of 10 mm symmetrical to the previous one with respect to the XZ plane. Close the sketch and rename the circle **Profile hole2** in the tree.





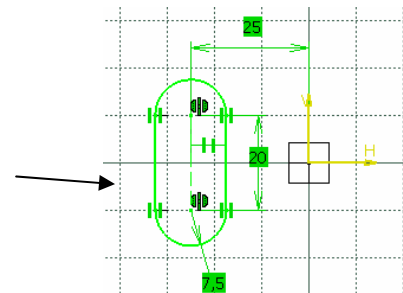
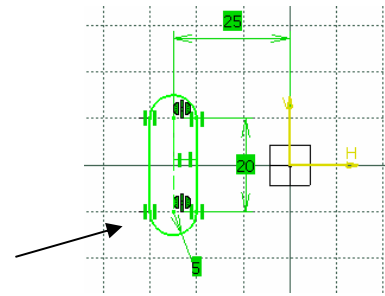
In the base plan there must be three different sketches as below



c) Construction of the boss profiles



- Build 2 planes shifted 15 and 30 mm upwards with respect to the xy plane.
- Rename the Lower plane bos (15 mm offset) the Upper plane bos (30 mm offset) in the tree.
- Open a sketch in the **Plan sup bos** and create the sketch opposite using the Oblong outline icon . Rename this sketch **upper base**
- Open a sketch in the **Plan below bos** and create the sketch opposite using the Oblong outline icon . Rename this sketch **lower base**

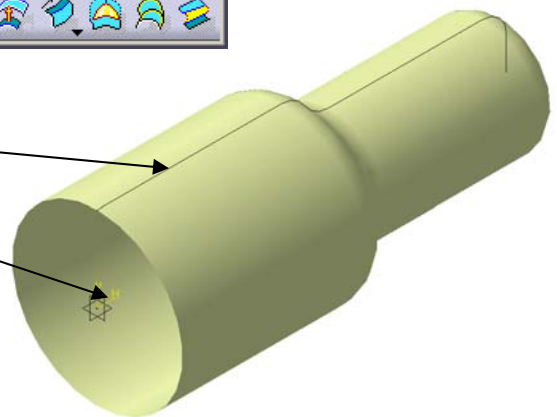
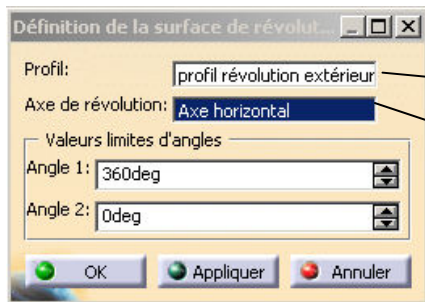


2 - Construction of the different surfaces

Hide all profiles in tree except **Exterior revolution profile**

- Creation of the external surface of revolution

Choose the Surface of revolution icon

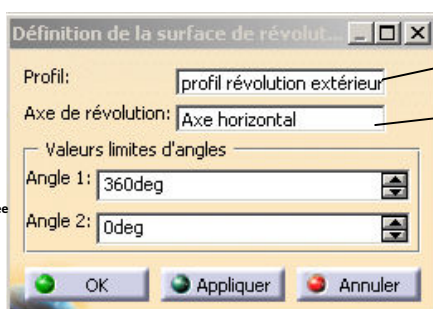


We get the revolution.1. Rename this revolution Exterior revolution Surface and hide the surface and its profile

- Creation of the inner surface of revolution

Show the **Interior revolution profile**

Choose the Surface of revolution icon

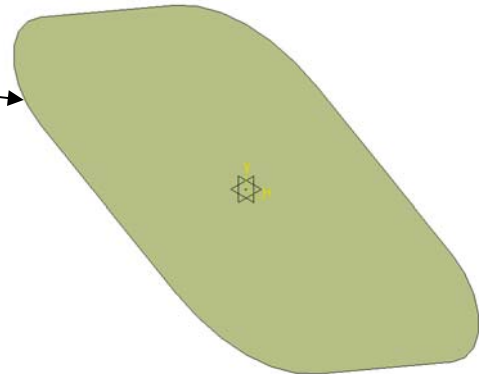
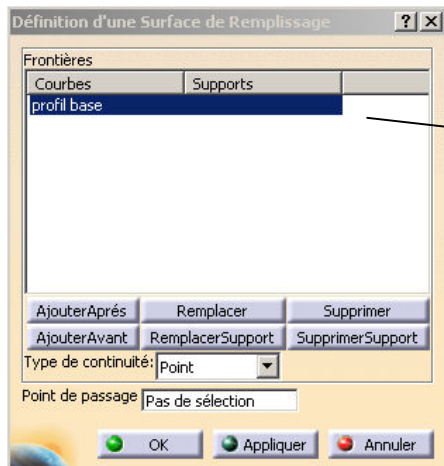


We get the revolution.2. rename this revolution **Interior revolution surface**, hide the surface and its profile

- Creation of the flat surface of the base

Show the **Basic profile**

Choose Fill icon

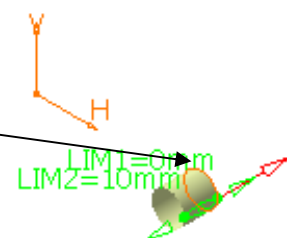
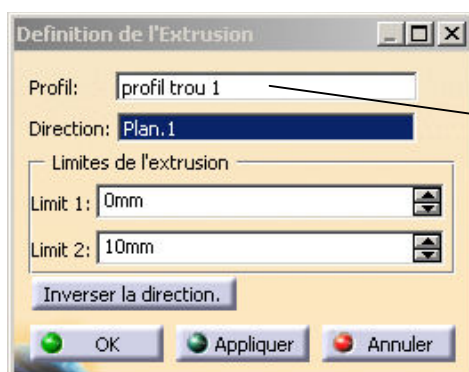


We obtain the filling.1 rename this filling **base area**, hide the surface and its profile

- Creation of the cylinder which materializes the 10 mm diameter hole

Show the **Profile hole 1**

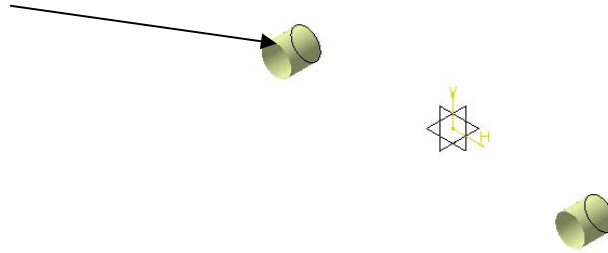
Choose the extrusion icon



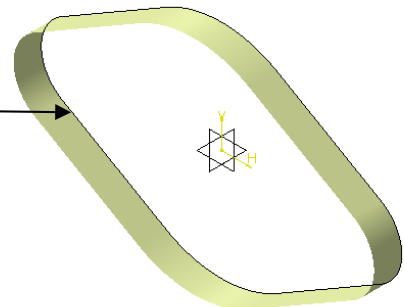
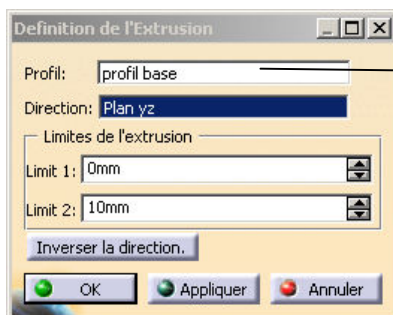
We obtain the extrusion.1 rename this extrusion **Hole 1**, hide cylindrical surface and profile circle



Do the same for the **Hole 2**



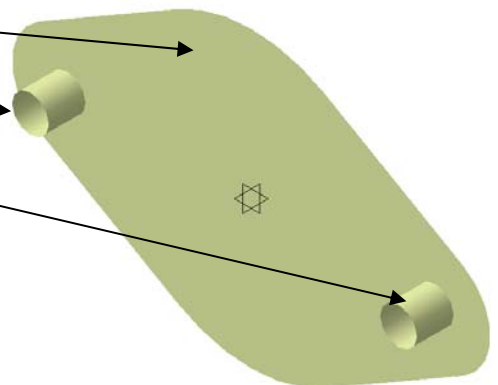
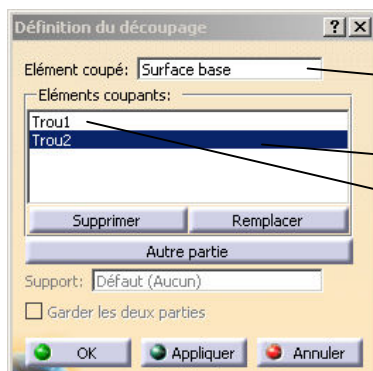
- Creation of the side face of the base
Show the **Profile base1** Choose the extrusion icon



We obtain the extrusion.3 rename this extrusion **Base side surface**, hide the surface

- Cut holes on the **Base area**

Show the **Base surface** and the **Hole 1** and **Hole 2**
Choose the create a cut icon



We get the **cutout.1**

Rename Cutout.1 **Perforated base surface 1**

- Obtaining the **Base area holed 2** from the **Base area with holes 1** Choose the create offset surface icon



We obtain the two base surfaces opposite. Rename Offset Surface **Perforated base surface 2**

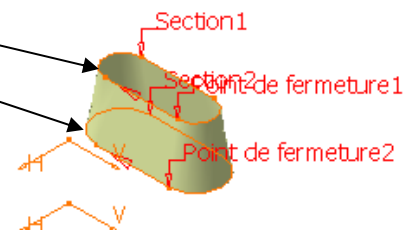
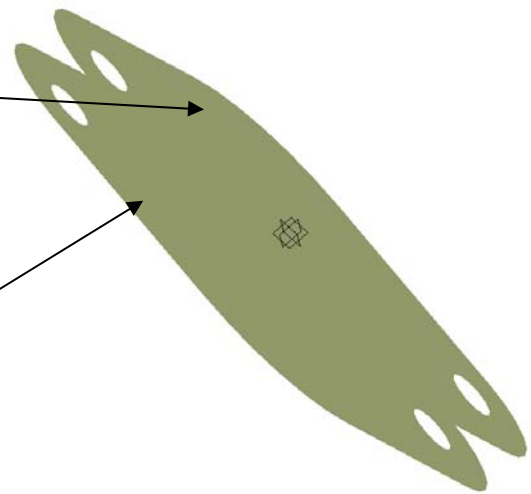
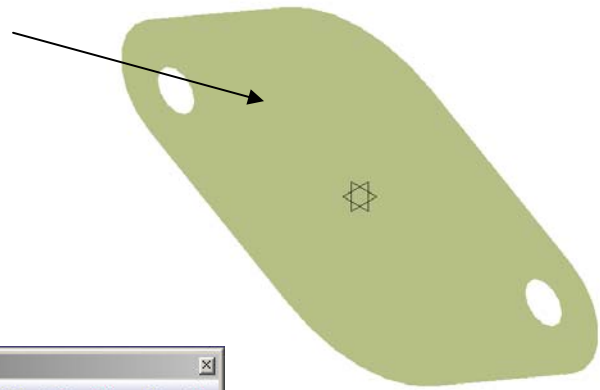
Hide both surfaces

- Creation of the boss surface Show profiles relative to the boss a) creation of the lateral surface Choose the guided surface icon



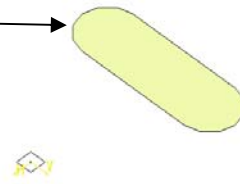
Rename Guided Surface.1 **Surface**

the your boss So.



b) Creation of the top surface of the boss

Mount profile **base sup bos** Choose the fill icon Select the **base sup bos** Rename Fill.2 **Upper surface boss** in the tree.



3 - Creation of surface assemblies with placement of fillets.

- Connecting the top surface of the boss to the side surface Choose the icon create a fillet between two surfaces



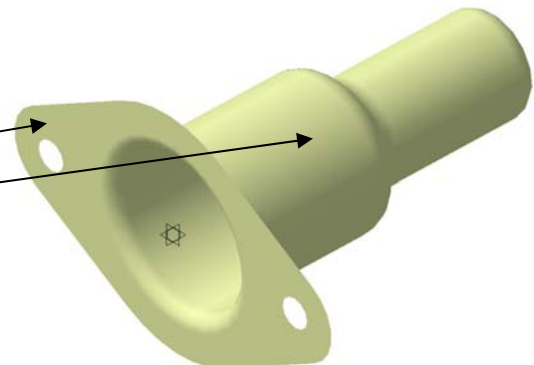
Check that the "cutting and assembly of supports" box is active **WARNING :**

Check that the two red arrows (normal to the surfaces) are oriented towards the kept material

A connection.1 surface is obtained in the tree structure.

Rename this surface **boss** Hide surface **boss**

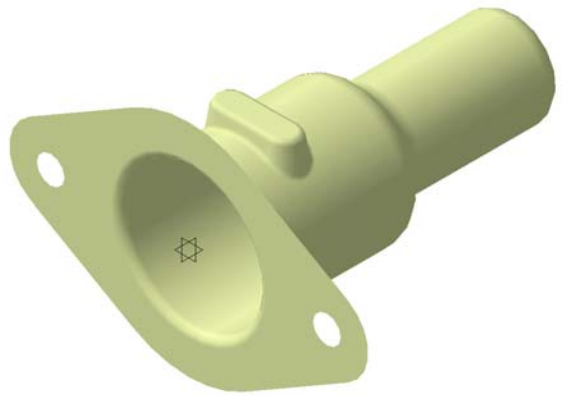
- Connecting the **Outer revolution surface** with the **Perforated base surface1** Choose the icon create a fillet between two surfaces



- Connection of **connection.2** with the surface **boss** Show area **boss** and proceed as before.

Choose a radius of 2 mm

We get the surface of **connection.3** opposite

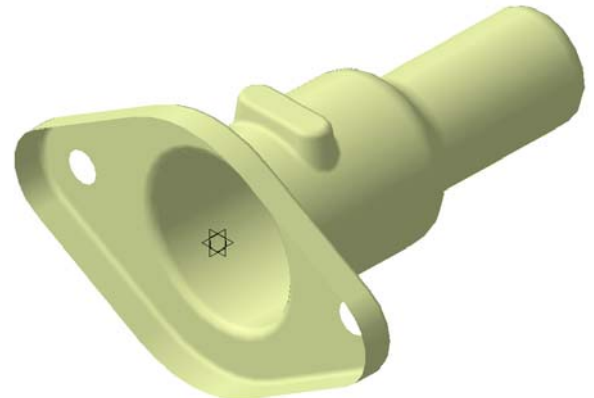


- Connection of **connection.3** with the surface **Base side surface** Show area **Base side surface** and proceed as before.

Choose a radius of 2 mm.

We get the surface of **connection .4** opposite

To hide **connection 4**

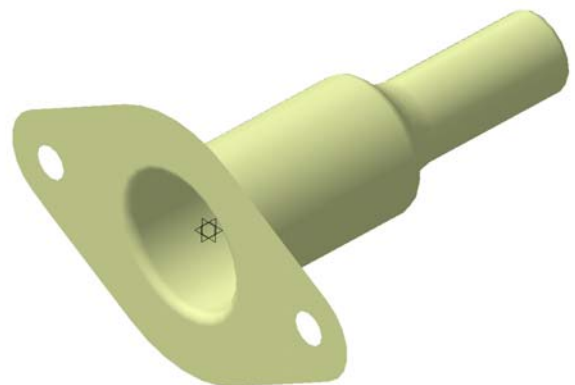


- Connecting the **Interior revolution surface** with the **Perforated base surface 2**

Show Inner Revolution Surface and Hole Base Surface 2

Proceed as before. Choose a radius of 2 mm.

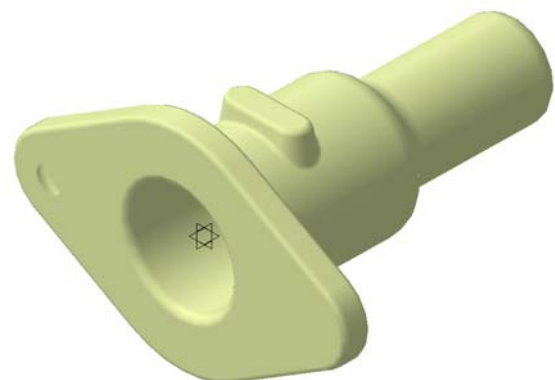
We get the surface of **connection. 5** opposite.



- Connecting the surface of **connection.4** with the area of **connection.5**

Show the **connection.4** Proceed as before. Choose a radius of 2 mm.

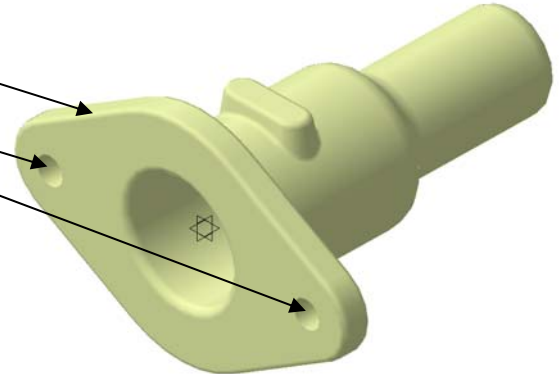
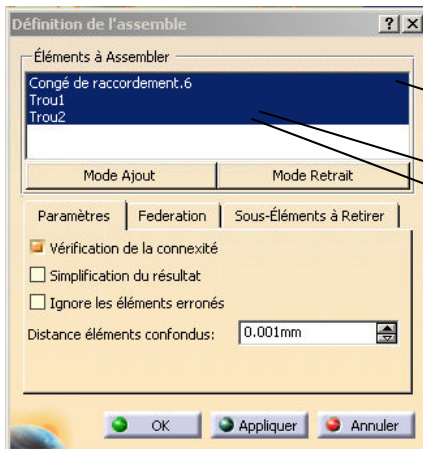
We get the surface of **connection.6** opposite.



- Join the **Hole 1** and **Hole 2** on the surface **connection.6**

To show **hole 1** and **hole 3**

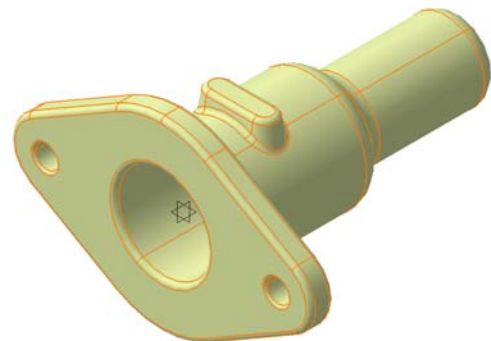
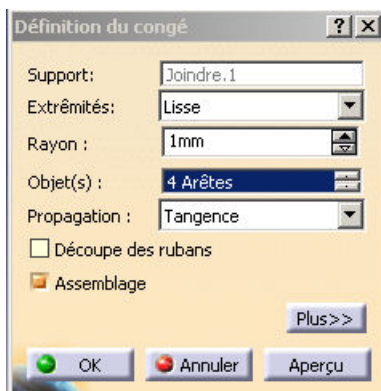
Choose the Attach icon



We get the surface **attach.1** opposite

- Connection of **Hole 1** and **Hole 2** on the surface **join 1**

Choose the icon create a fillet on edge of the surface **attach.1** with a radius of 1 mm



We obtain the finished surface given at the beginning of the lab.

Rename edge fillet.1 **Master cylinder**