



SOLID MODELLING ASSIGNMENT

Romain PINQUIE

This document corresponds to the assignment report for the 3D solid modelling course. It includes the design step by step of the Slider component using CATIA.V5 software package, some Slider's drawings and a short account of the benefits of solid modelling for mechanical design.

12/12/2011

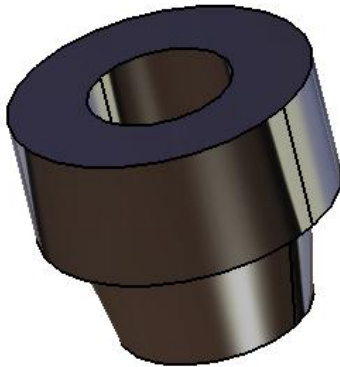
TABLE OF CONTENTS

DESIGN OF THE SLIDER COMPONENT	3
1. FIRST STAGE	3
2. SECOND STAGE.....	5
3. THIRD STAGE	11
I. SOME OF THE ALTERNATIVE DESIGNS	12
1. MAIN STEPS OF THE FIRST ALTERNATIVE DESIGN	12
2. MAIN STEPS OF THE SECOND ALTERNATIVE DESIGN.....	13
II. SOLID ANALYSIS.....	14
1. MATERIAL DEFINITION	14
2. SLIDER'S MASS PROPERTIES	14
3. SLIDER'S CENTRE OF MASS	14
III.DRAWING	16
1. ISOMETRIC VIEW WITH HIDDEN EDGES INVISIBLE.....	16
2. PLOT OR PLOTS SHOWING ALL SKETCHES	17
3. ISOMETRIC VIEW WITH HIDDEN EDGES DISPLAYED AS DASHED	18
4. DIMENSIONED ISOMETRIC VIEW.....	19
5. DRAFTING OF THE FRONT, TOP, RIGHT AND ISOMETRIC VIEWS.....	20
IV.THE BENEFITS OF 3D SOLID MODELLING.....	21

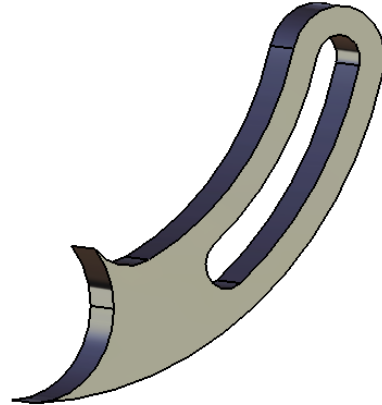
Design of the Slider component

The design of the part can be divided into three different stages as following:

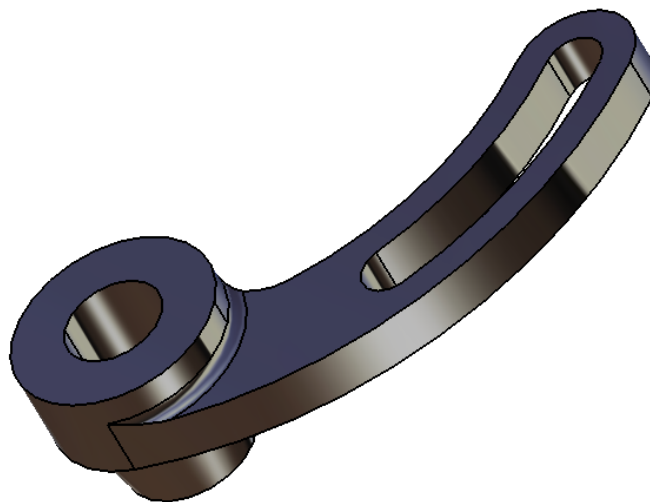
1st Stage



2nd Stage

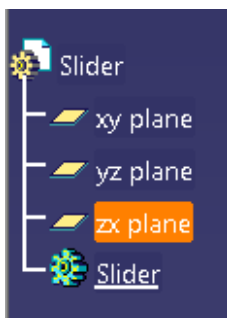


3rd Stage: Final part



1. First Stage

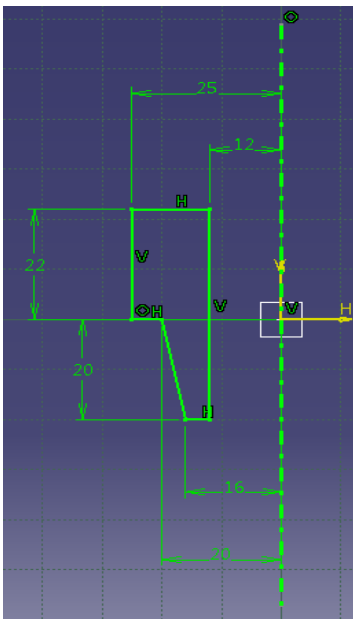
- Selection of the “zx plane” to sketch the first stage



Firstly we define the “zx plane” as reference to start the sketch, which will correspond at the first stage. The “zx plane” is preferred at the others in order to respect the co-ordinate system set up in the assignment sheet and to have the good coordinates when we will calculate the Slider’s centre of mass.

▪ **Sketch:**

○ **Step 1: Design of the cylinder + cone profile**



To design the cylinder and the cone we have to draw the sketch

using the “Profile” tool



We can further insert the dimension values by going to the

“Constraint” icon

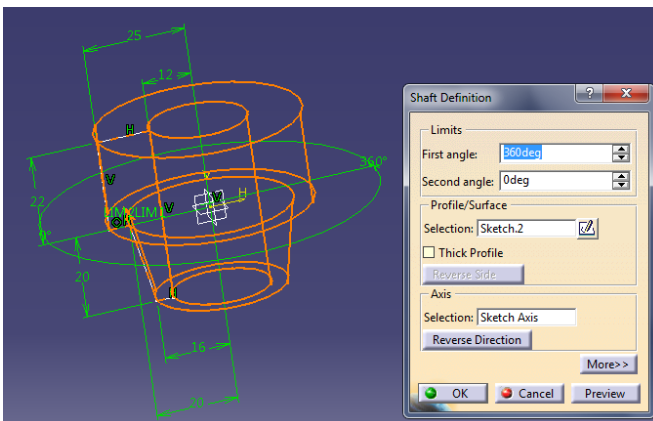


Both parts cylinder and cone are revolution shapes therefore we will apply the shaft technique to design them. But to extrude the

sketch it is necessary to define an axis as reference in order to revolve the profile around itself, besides this one must be coincident with the y axis to respect the axis system given.



○ **Step 2: Design of the shaft**



The extrusion is generated by selecting the

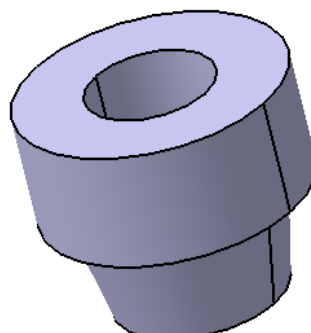
“shaft” item



. Three boxes have to be completed:

- *First angle*: we want a revolution equal to 360°
- *Profile selection*: we choose the profile designed previously
- *Axis election*: the axis allowing the revolution was defined in the sketch, therefore it is automatically chosen.

Stage 1 finished



2. Second Stage

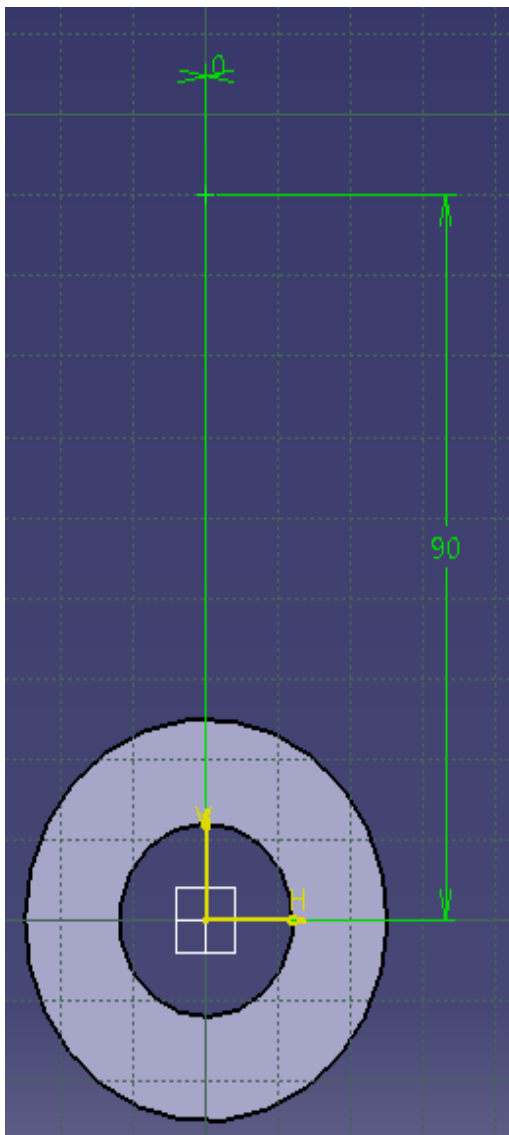
- Selection of the “xy plane” to sketch the second stage



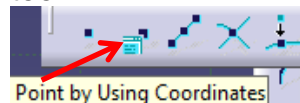
Enter sketch on the “xy plane”, which constitutes the reference always to respect the co-ordinate system.

- Definition of the construction elements

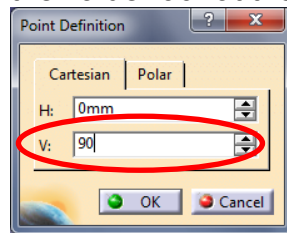
- Step 1: Creating of the reference point



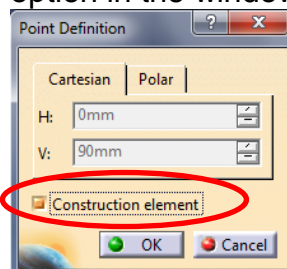
We can realize that a majority of the edges are arcs designed with the point of co-ordinates (0;90;0) as centre, so it will be useful for the next steps. To create the point we click on the “Point by Using Coordinates” tool.



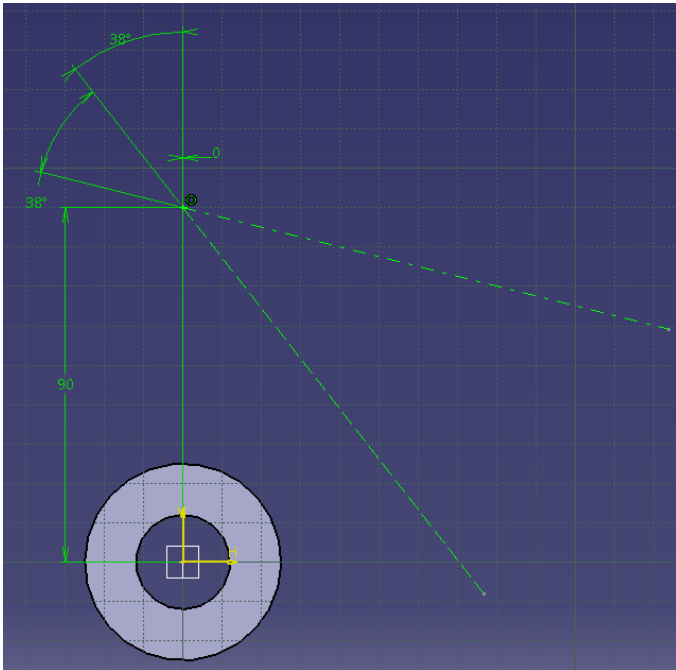
Then the window allows the user to define the Cartesian co-ordinates appears and we have to enter the value “90” at the “v” label.



Attention! The point is not a construction element it should not be present in the construction sketch, so I will switch it to construction element selecting the option in the window “point definition”.



○ Step 2: Design of the axes construction elements



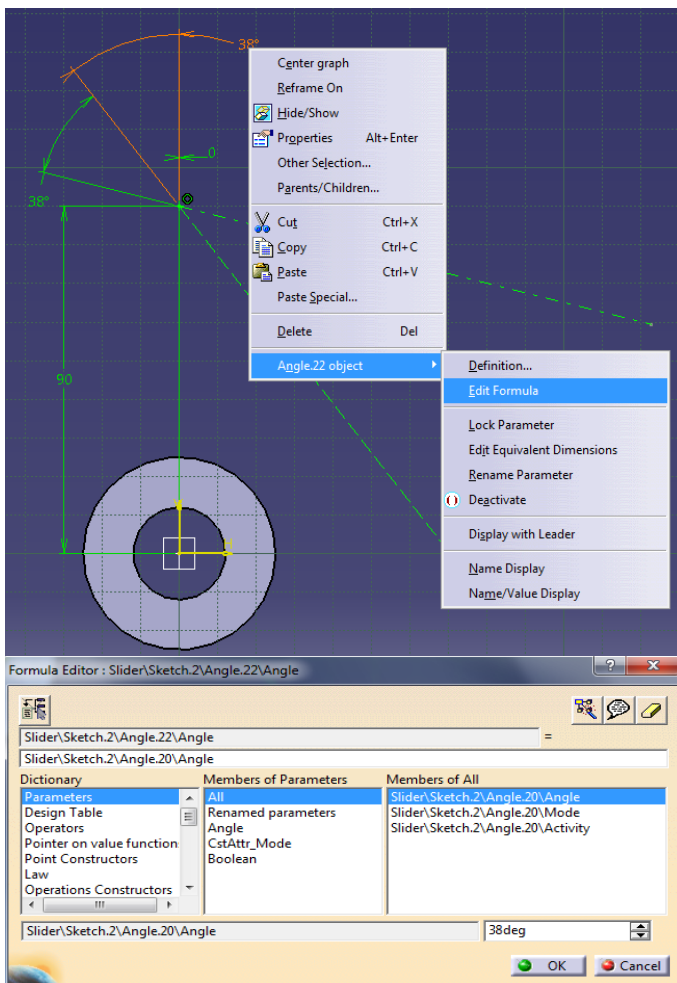
In order to define the pocket presents in the part, I design the both axes using the



“Axis” item, and for the same reason as the step one, I switch it to construction element.

The dimensions are added like at the step one using the “Constraint” tool.

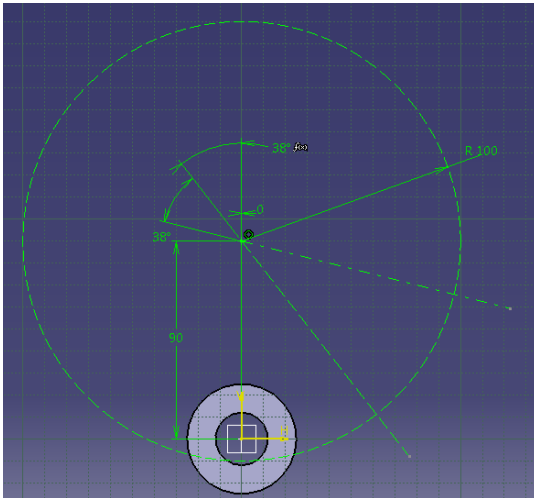
○ Step 3: Define a function to pilot the angle's constraint



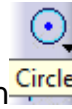
Reading the definition drafting we realize that the angle specifying the position of the two axes is the same. Therefore we can use a formula to handle them. To edit the formula we highlight one of the two constraints, click right and select “*Edit formula*” tool, then we click on the second angle constraint and fill in the value of the angle that we want, so here it is 38 degrees and validate clicking OK button.

The advantage of this solution is that if we want to change the angle's value of each one we just have to change one of them.

○ **Step 4: Design of the circle “R100mm” as construction element**



I design the circle of radius 100mm using the

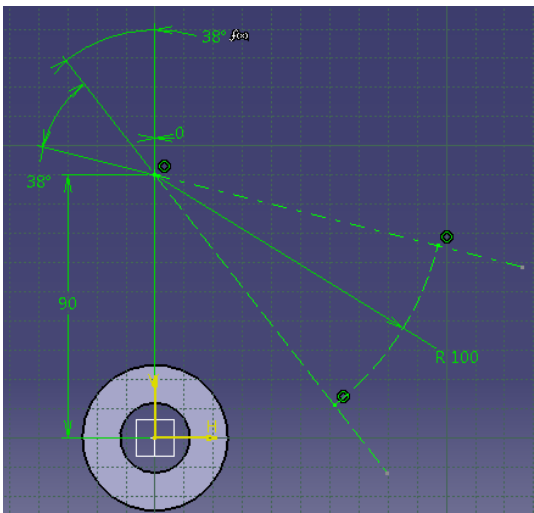


“Circle” icon

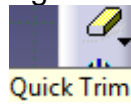
As in the drafting sheet, I centre the circle on the reference construction point made before.

Moreover, it should not be part of the final sketch, so another time again, we have to switch the feature to construction element.

○ **Step 5: Erasing of the edges useless**

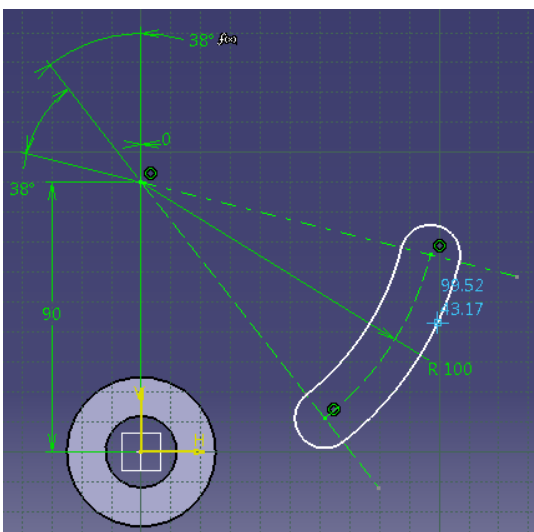


To make the drawing more precise we can erase the edges useless by selecting the “Quick Trim”

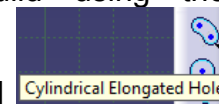


item **Quick Trim** and clicking on the edges that we want to delete. In the end, we keep only the arc that will allow us in the next steps to sketch the pocket and the boundary of the part.

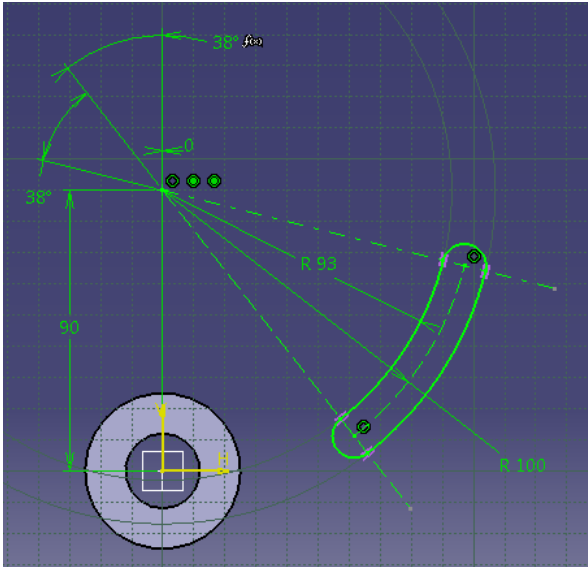
○ **Step 6: Design of the pocket**



The pocket is build using the “Cylindrical

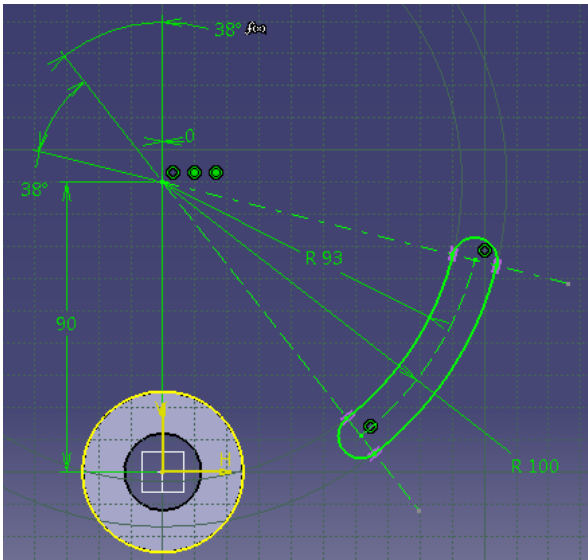


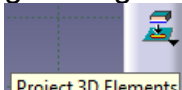
Elongated Hole” tool **Cylindrical Elongated Hole**. To define the pocket’s sketch three points need to be selected. The first one corresponds at the centre, which is the construction point (0;90;0), the second point is on one of the two axes and the last one is on the other axis.

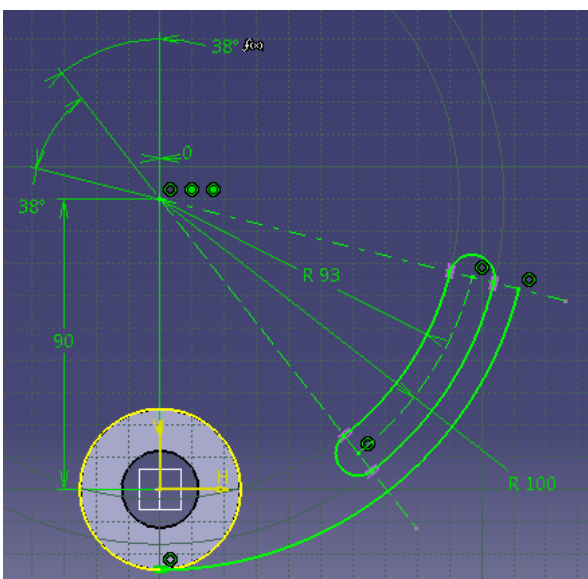



To finalise the sketch we have to add the radius constraint “R93mm” given by the definition drawing and put a tangency constraint between the arcs to be sure that it is well constrained.

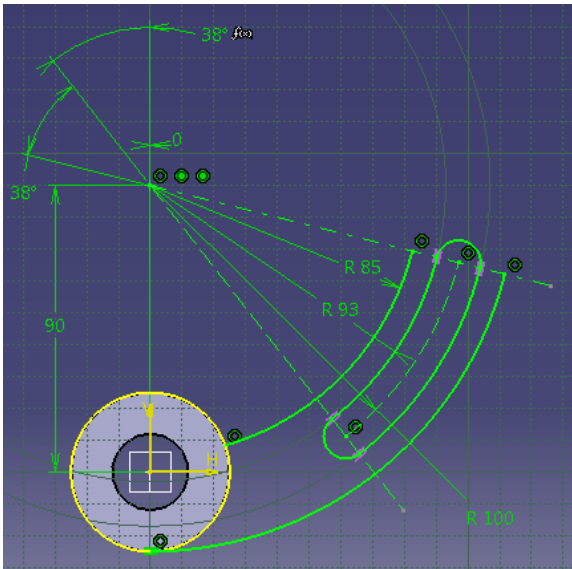
○ Step 7: Design of the curve part



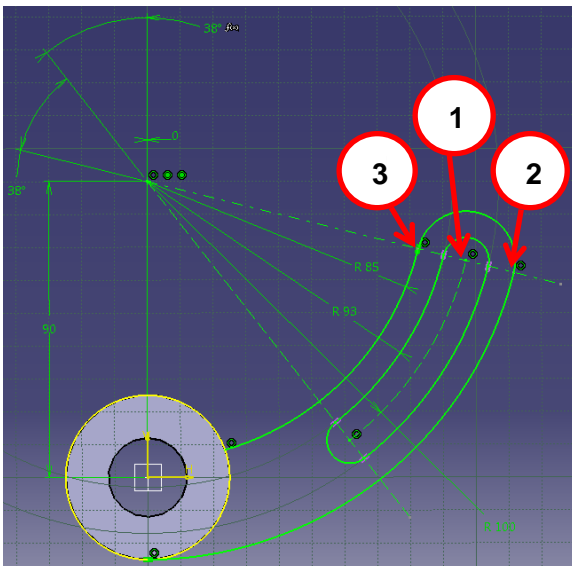
To design the end of the second stage we will reuse the circle “R25” by projecting its edge with the “Project 3D Elements” function .



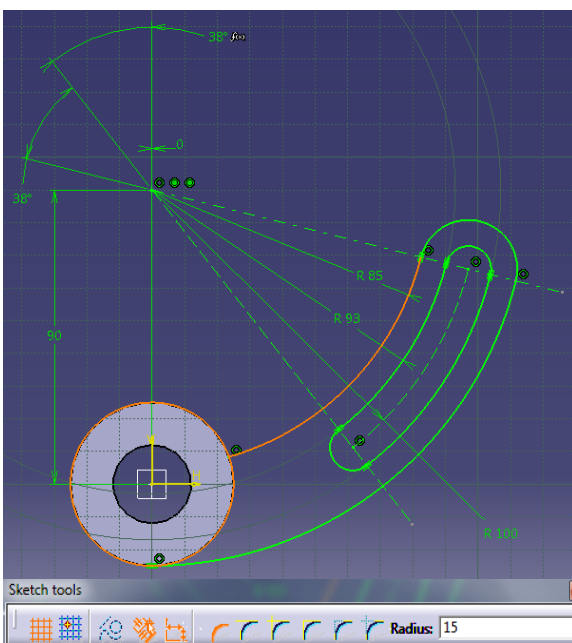
To design the curve part we use the “Arc” tool  by choosing the construction point as a reference again and the trick is to use the tangency property between the edge projected at the last step and the arc that we are sketching. Furthermore, the limits points of the arc are on the projection for one of them and on the axis construction element for the other.



The arc radius “R85mm” is drawn with the same method as before.



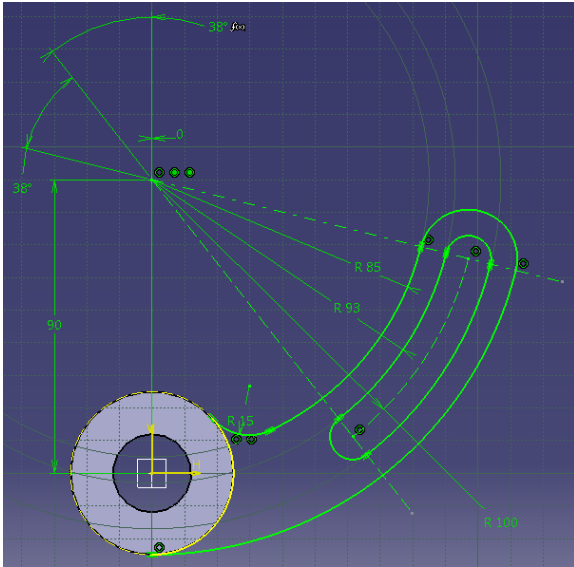
By choosing the “Arc” tool we can close the end of the sketch. We need to select the centre of the arc, which is the point (1), then we choose one of the limit points (2) or (3) and to finish we click on the remaining point in order to close the sketch. Thanks to the intersection points and the “Arc” tool, we can obtain a concentric arc with the end of the pocket.



One of the solutions to design the edge fillet is to



use the « Corner » function. Then, we need to highlight the projection of the circle and the arc “R85mm”, afterwards a toolbar called “Sketch tools” appears containing the radius label and one box, which allows entering the value of the radius (15mm in our case).

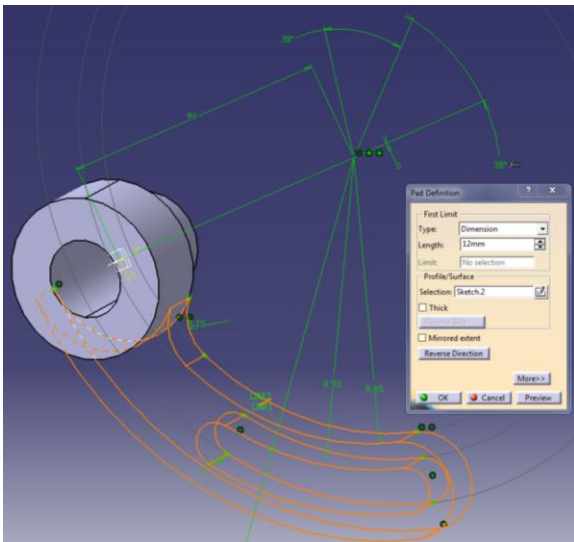


But as we can see above we have two closed boundaries but it is not possible to extrude them, we must have only one. In the same way in step 6, I erase the edge concerned by picking “Quick



time” item **Quick Trim** and clicking on the part that I want switch to dashed line.

○ Step 8: Extrusion of the sketch



To finish we built a 12mm pad by going to “pad”



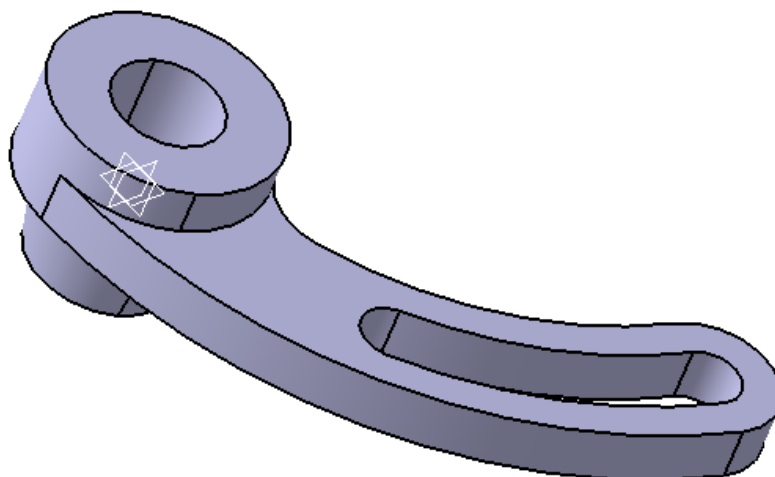
and applying the following parameters:

-*Type*: Dimension.

-*Length*: $S=22\text{mm}$ according to the dimensions allocated.

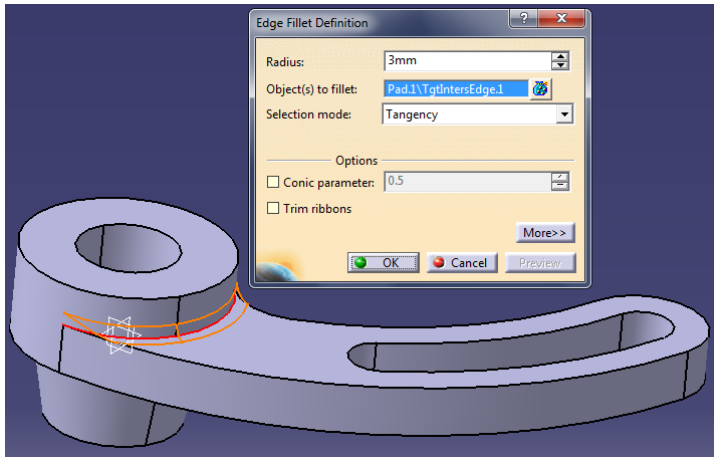
-*Selection of the profile*: We have to highlight the sketch designed in the last step (see above).

Stage 2 finished

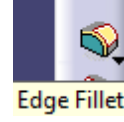


3. Third Stage

- Design of the edge fillet "R3mm"

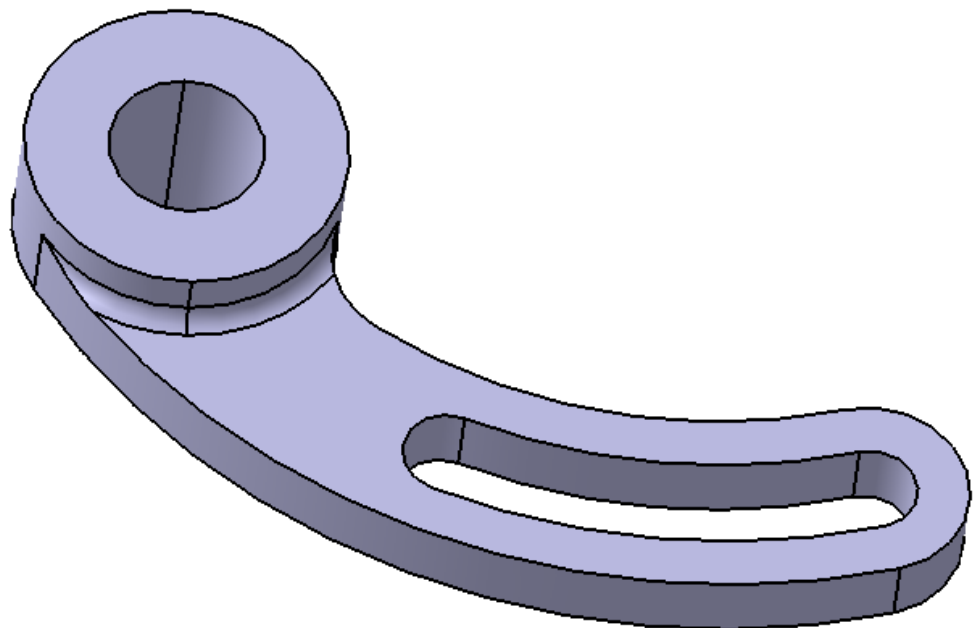
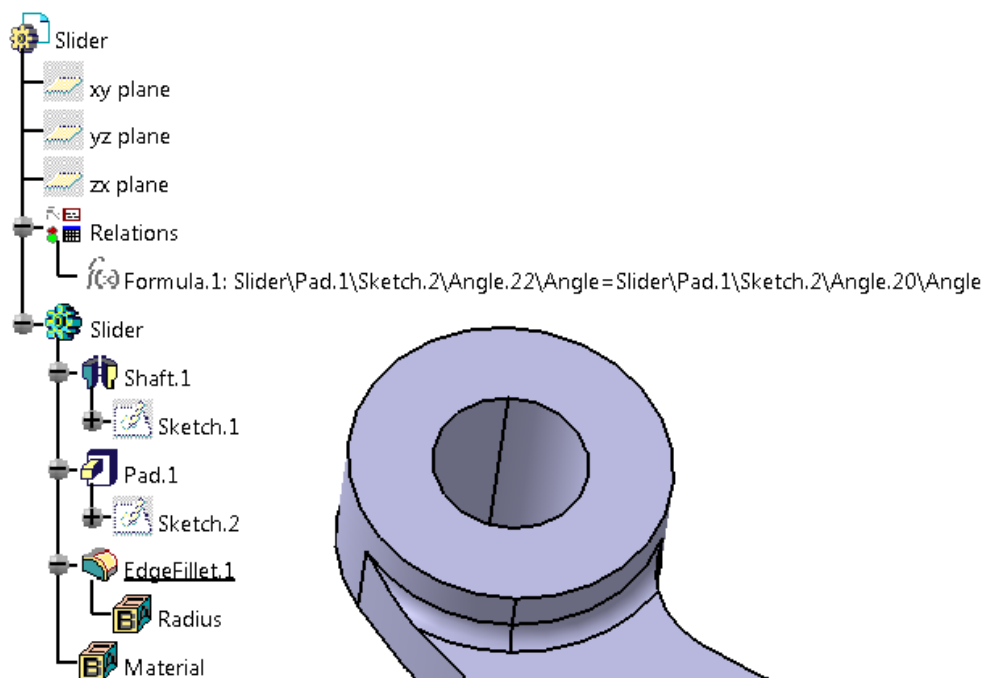


The edge fillet is designed by picking



the "Edge fillet" item and by highlighting the edges that we want to modify. The definition of the radius value can be changed in the "Radius" box and as in most cases the "Tangency" mode is conserved to have a fillet tangent to the next surfaces.

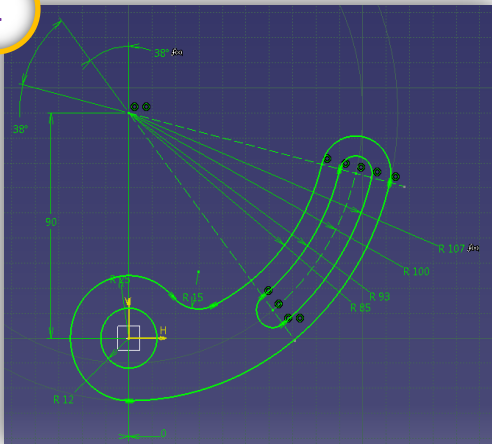
Final part



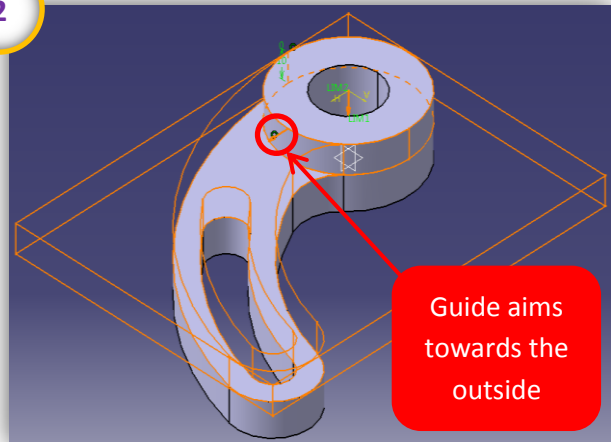
I. Some of the alternative designs

1. Main steps of the first alternative design

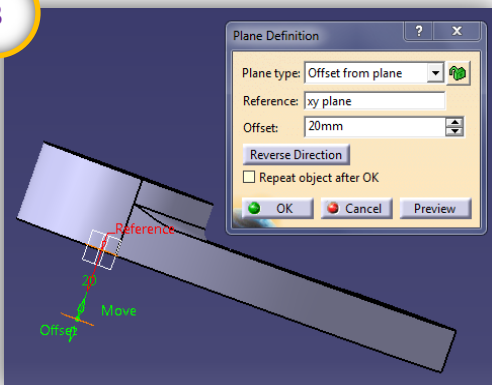
1



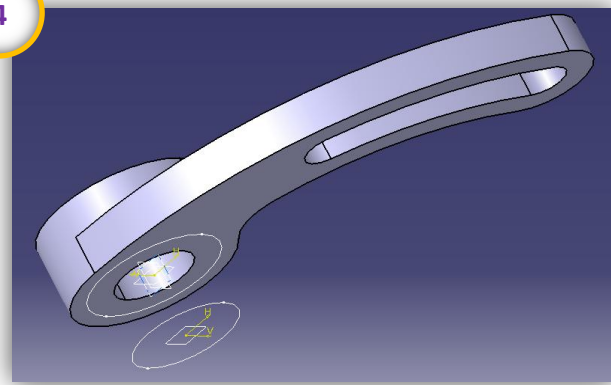
2



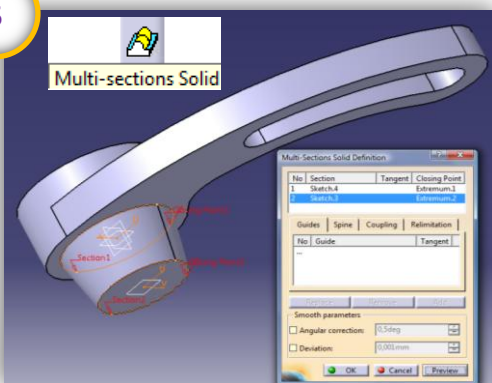
3



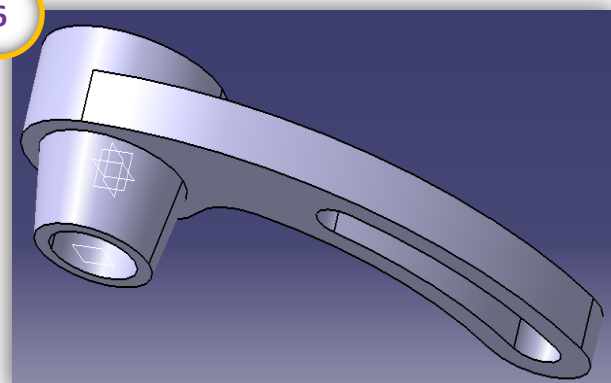
4



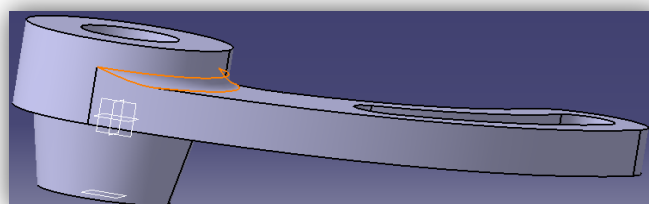
5



6

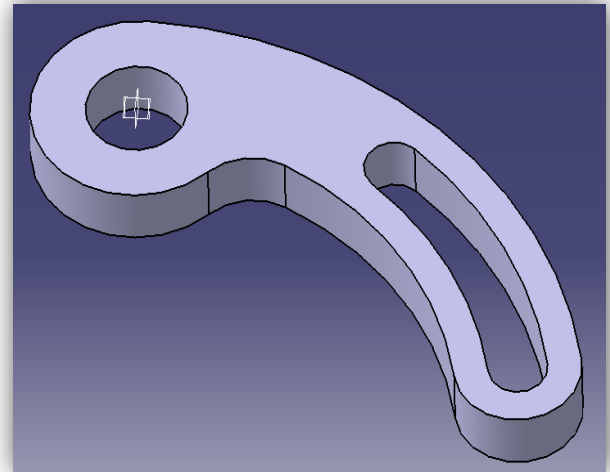
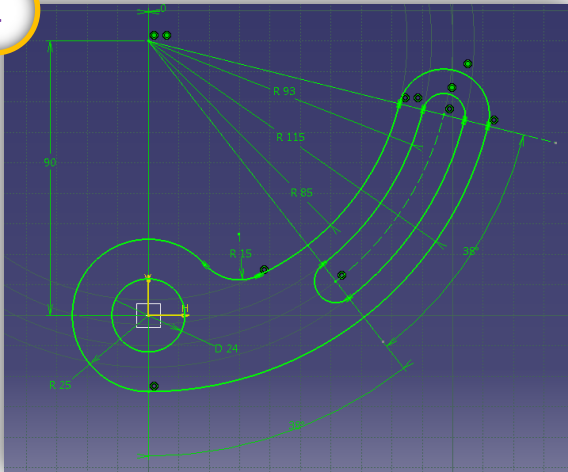


7

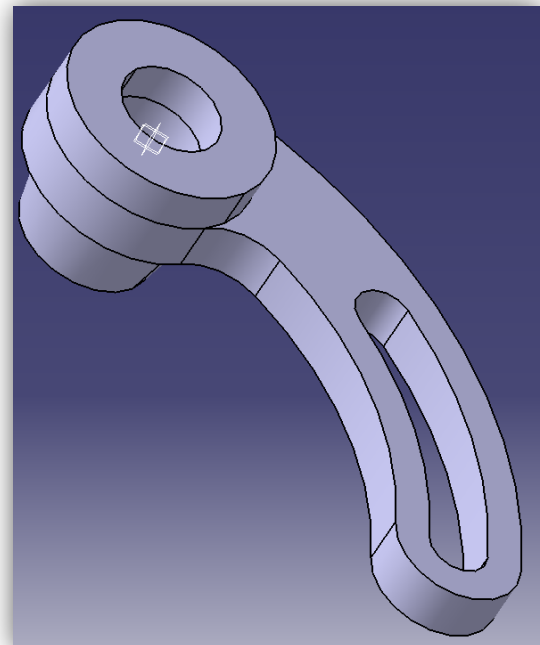
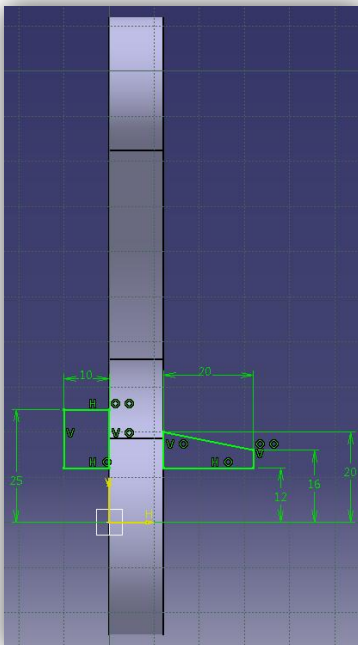


2. Main steps of the second alternative design

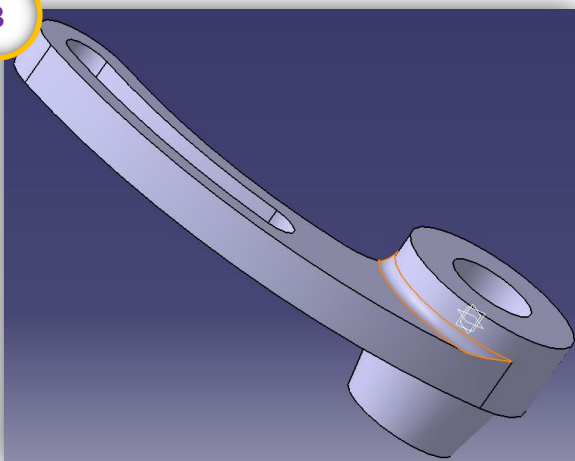
1



2



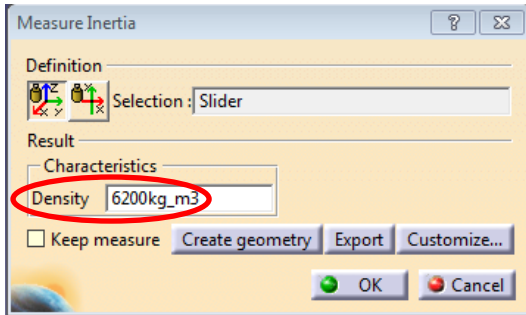
3

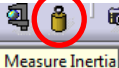


There are many ways that lead to the same result **as long as the sketches are well built and the constraints are well used**. But some of them require less sketches and features, thus both complexity of the design and size of the file are reduced. It is essential to have a design as simple as possible because in many instances the parts are added in an assembly, which can contain thousands of elements. Among the three designs that I have suggested, I have decided to choose the first one as the best solution because it is made up of only two sketches and one operation of edge fillet. Even the biggest sketch, i.e 1, which seems complicated is in reality very clear and easy to modify. **Therefore, neither the simplification of the sketches nor the reduction of their number must be forgotten.**

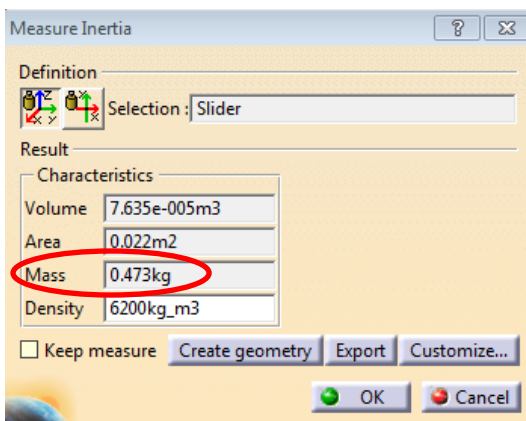
II. Solid analysis

1. Material definition



As requested in the assignment sheet, we need to specify the density of the alloy chosen in order to obtain the good mass properties and coordinates of the centre of mass. The “Measure Inertia” tool  provides a lot of mechanical information including the definition density of the material (6200 kg.m⁻³ for this case).

2. Slider's mass properties

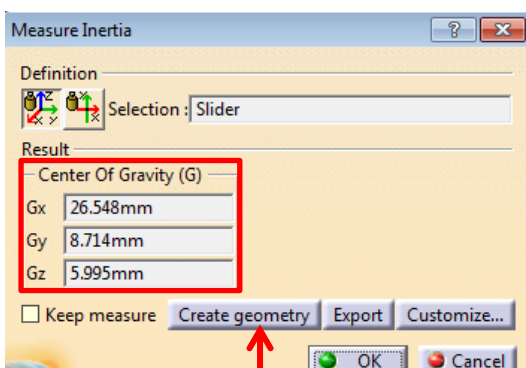


In the same window we can evaluate the Slider's mass in a box called “Mass”, so here the weight of our component is equal to 473 g. Just out of curiosity we can check the truthfulness of the result:

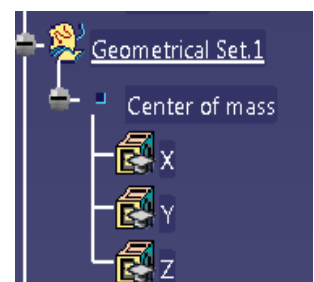
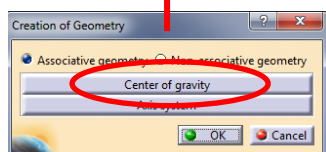
- ✓ $Volume = 7,635 \cdot 10^{-5} m^3$
- ✓ $Density = 6200 kg.m^{-3}$

$$\begin{aligned} \Rightarrow Mass &= Volume \cdot Density \\ &= 7.635 \cdot 10^{-5} \cdot 6200 \\ &= 0.47337 kg = 473.37 g \end{aligned}$$

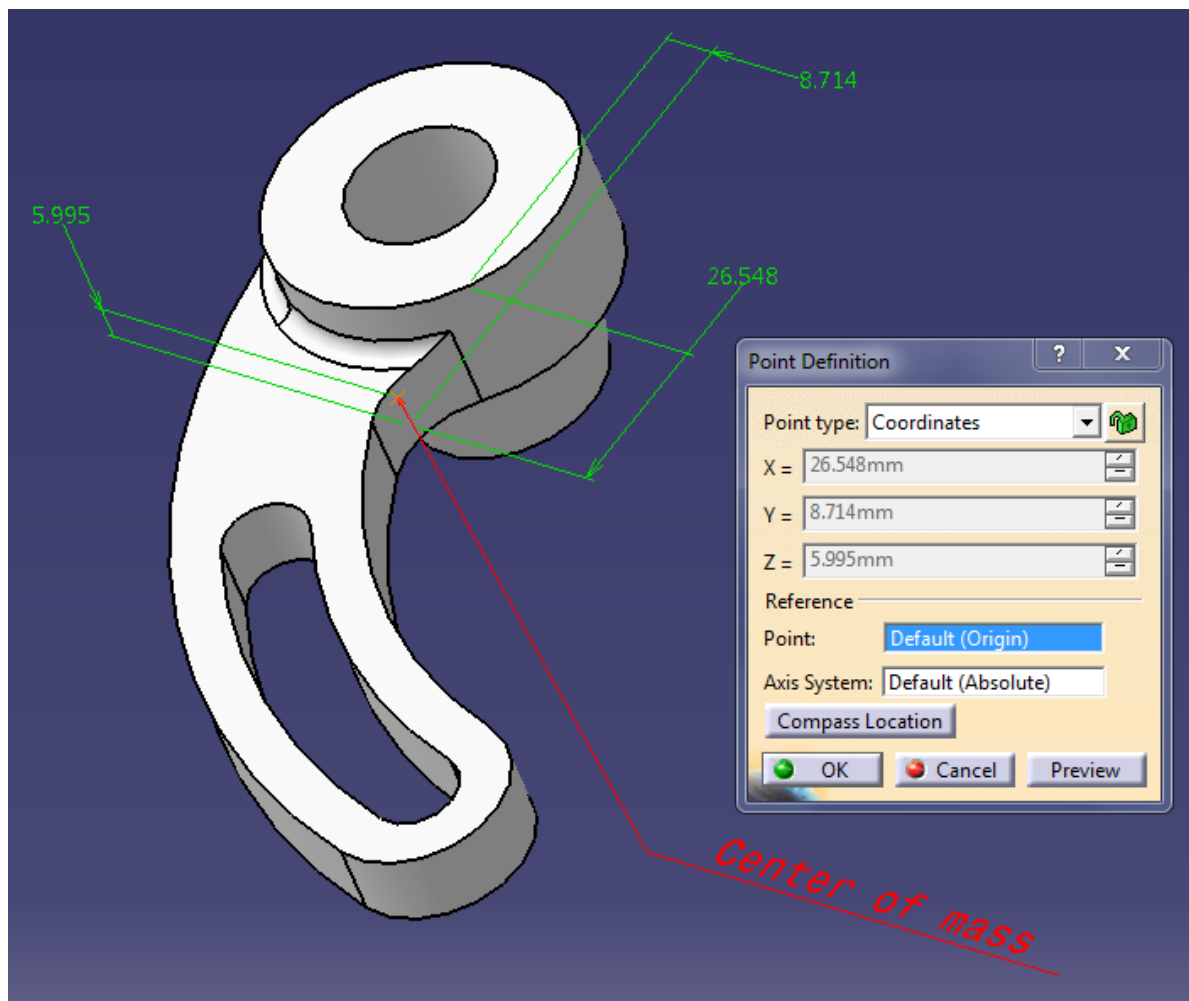
3. Slider's centre of mass



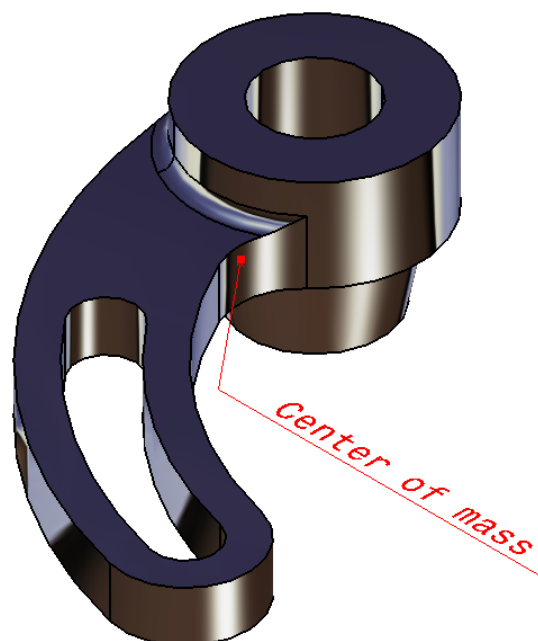
We can also know the coordinates (x;y;z) of the centre of gravity using the “Measure Inertia” tool and clicking on the part's body. Then choosing the option “Create geometry” and “Centre of gravity” we display it on the part with as reference the origin of the part.



To display the coordinates of the centre of mass we have to open the point created in the “Geometrical Set” in the specification tree.

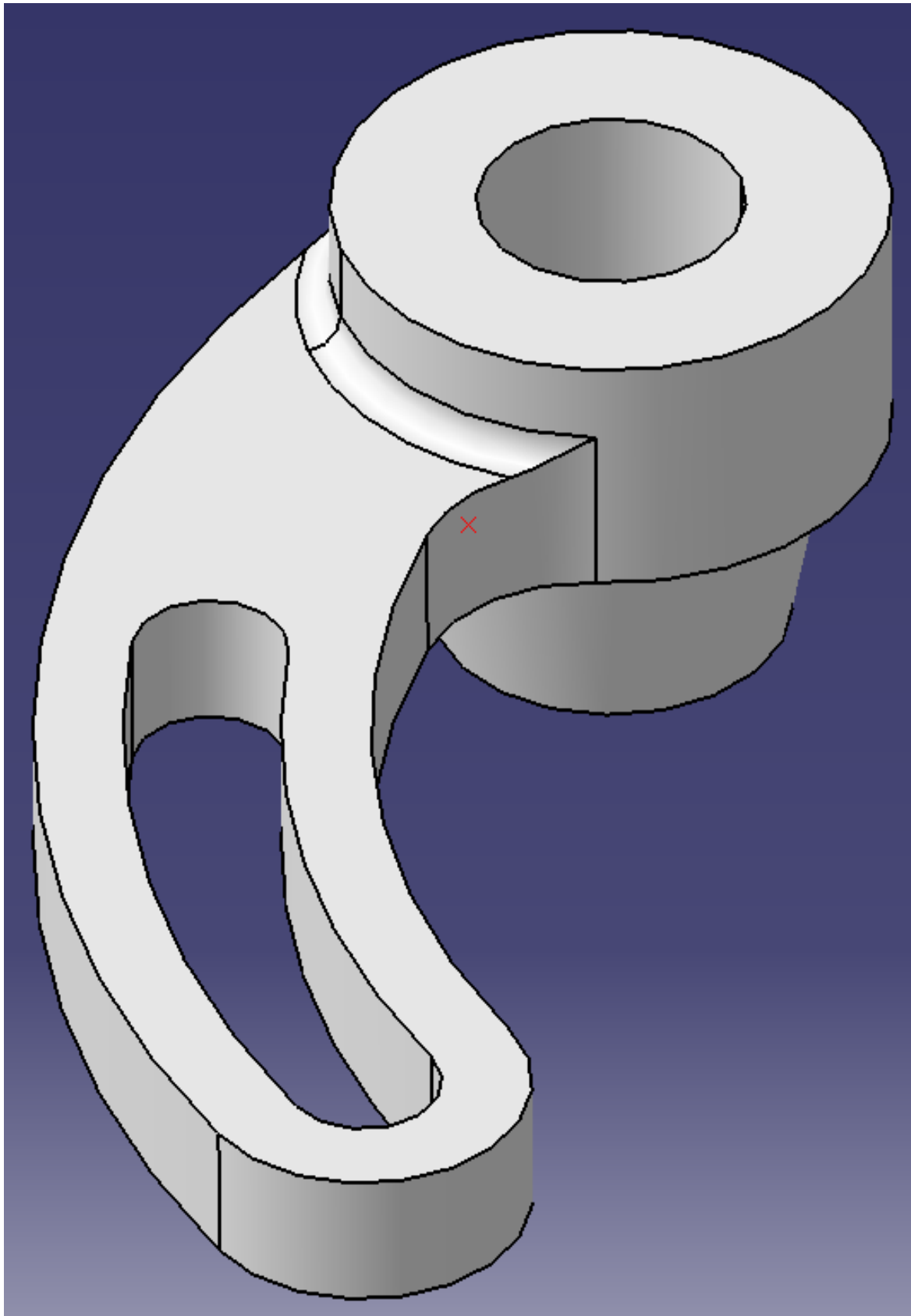


Finally, the centre of mass appeared on the Slider's body with the *point definition* window, which includes the values of the coordinates x, y and z with as reference the origin of the part and as axis system the three-plane datum-system defined at the beginning of the design. In short, the definition of the centre of mass is one of the advantages of CAD software and it is especially true for the complex geometries.

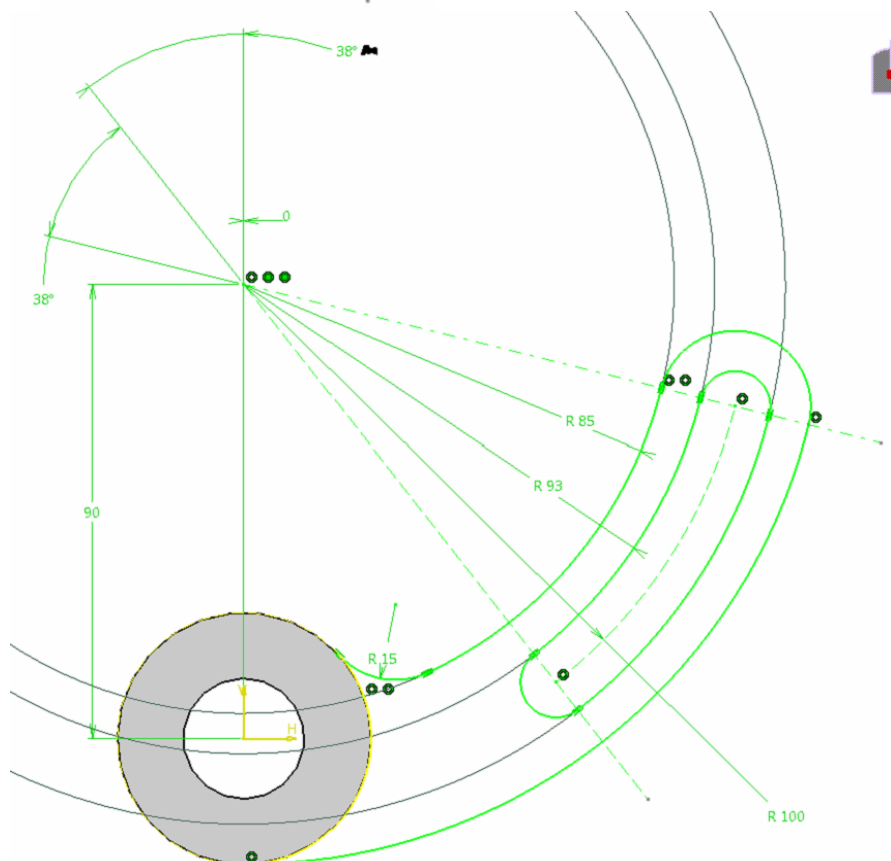
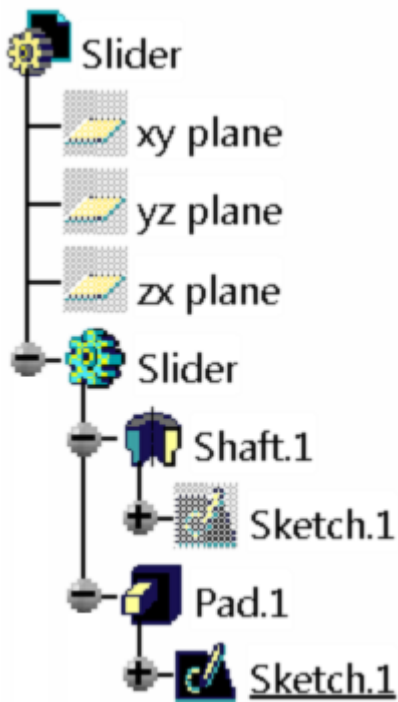
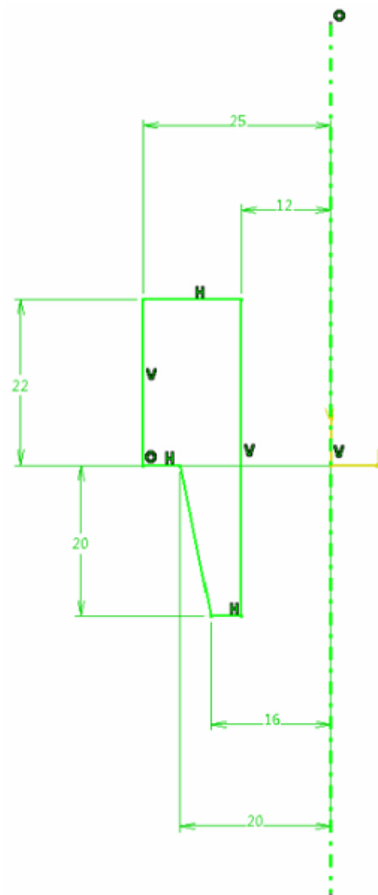


III. Drawing

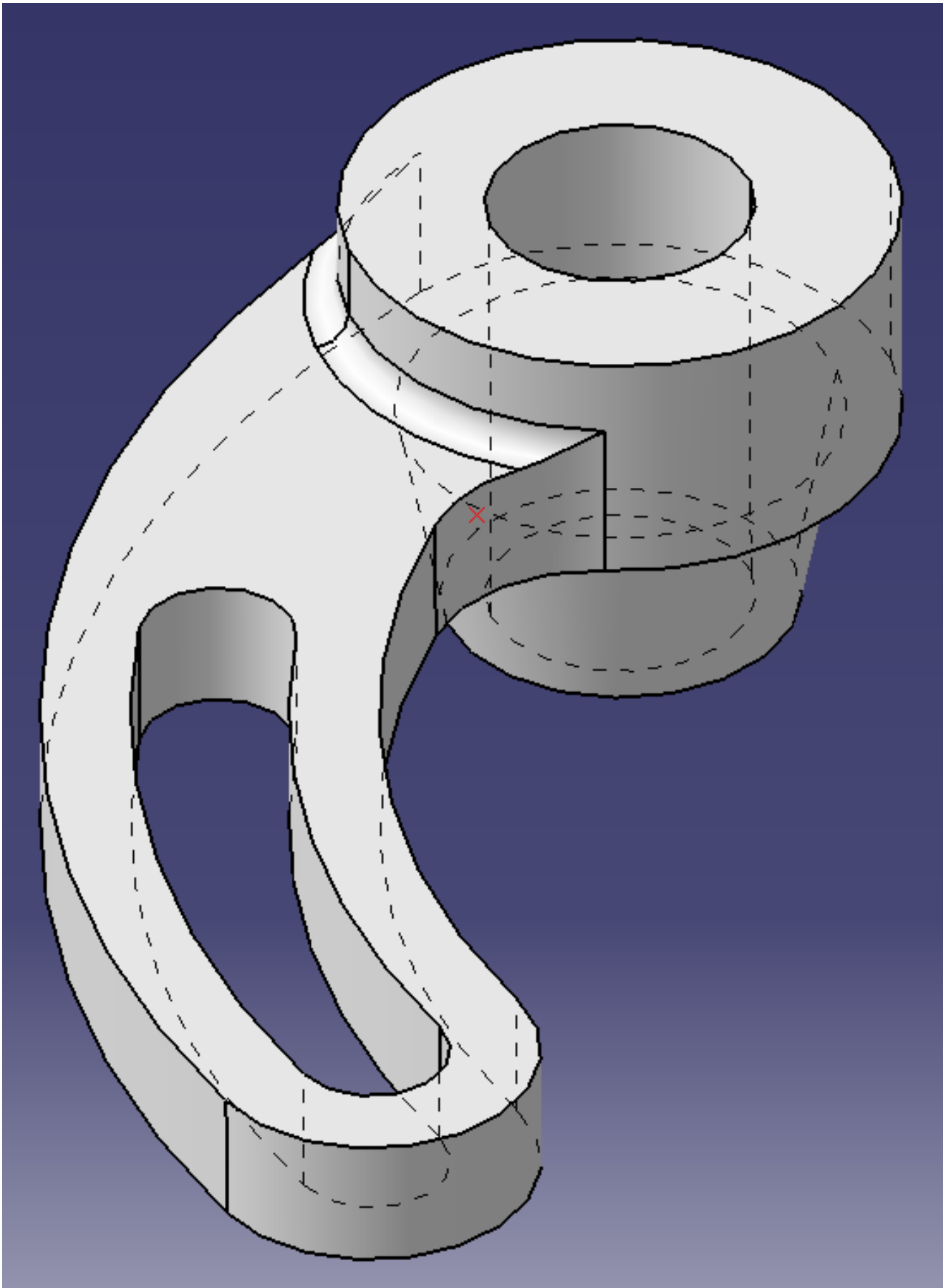
1. Isometric view with hidden edges invisible



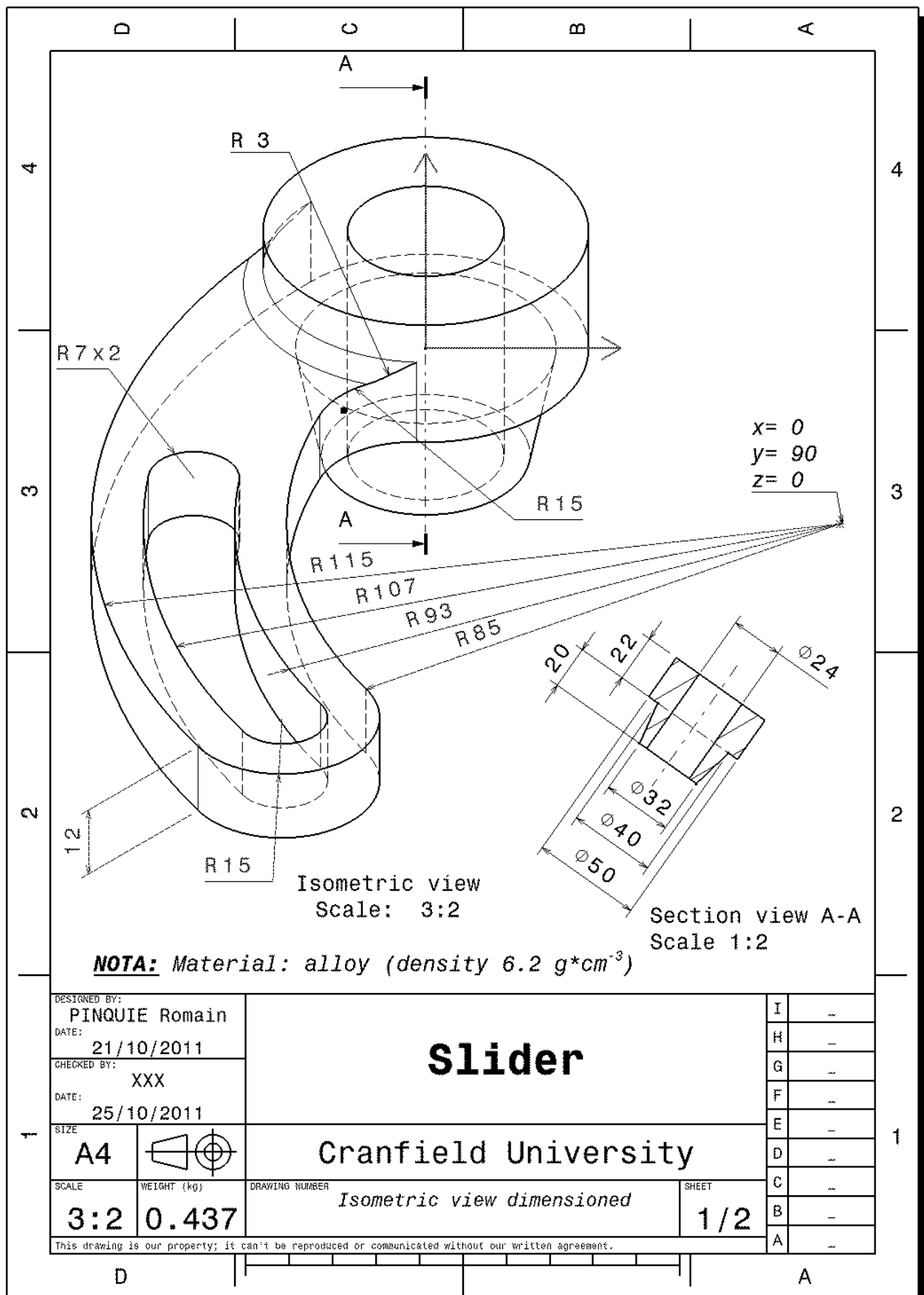
2. Plot or plots showing all sketches



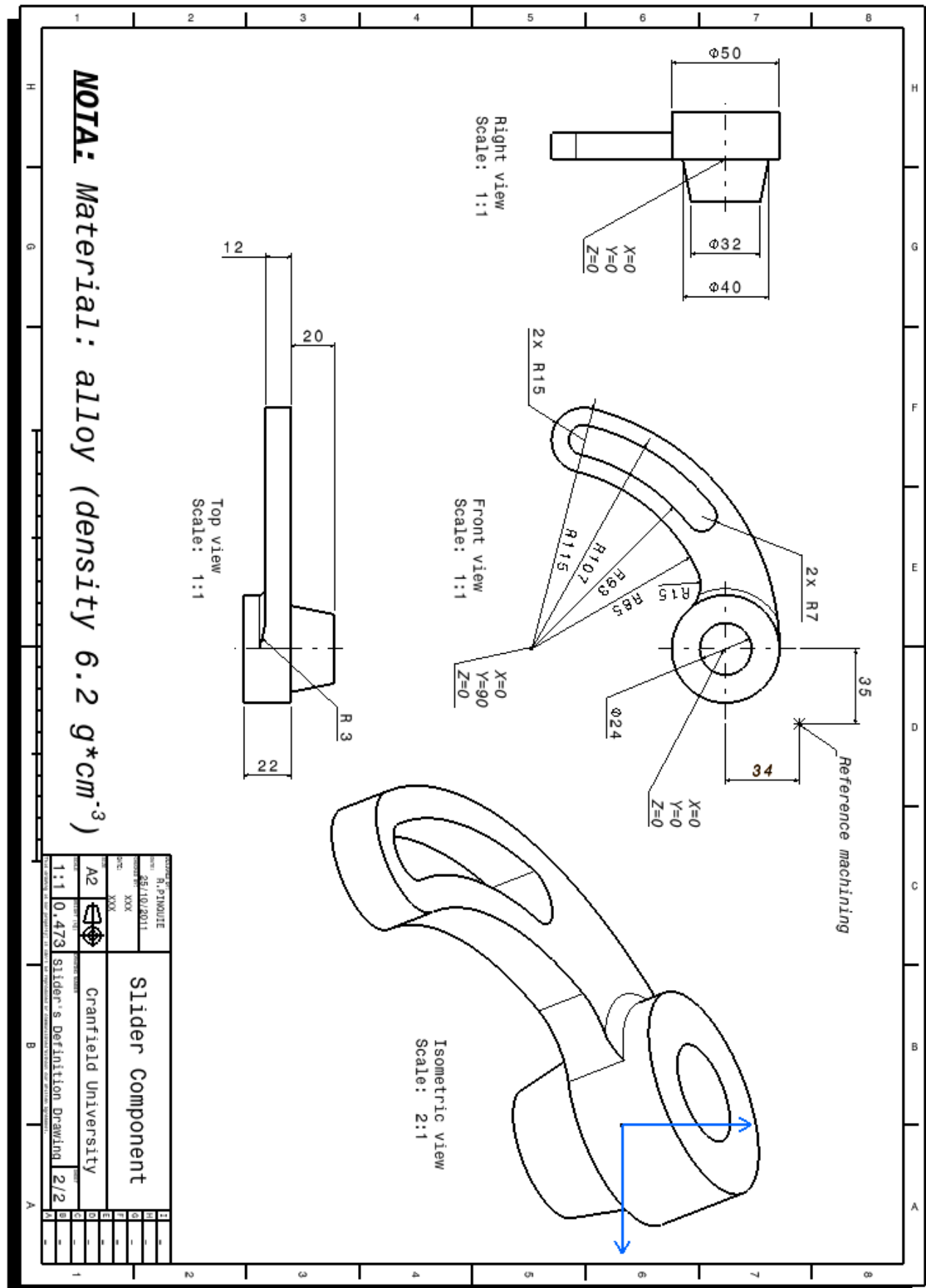
3. Isometric view with hidden edges displayed as dashed



4. Dimensioned isometric view



5. Drafting of the front, top, right and isometric views



IV. The benefits of 3D Solid Modelling

Nowadays the 3D Solid Modelling plays an important role in all areas of mechanics. Each stage of the Product Development Process from the Conceptual Design to the Manufacturing Engineering requires the use of numerical interfaces.

- *Conceptual Design:* In a design environment we generate 3D geometrical models.
- *Analysis/Simulation:* 3D model is used to simulate its physical behaviour and to compute its performances in a virtual environment close to reality, e.g., the structural analysis aims in most cases at reducing the weight of the product by keeping a good structural performance.
- *Detail Design:* After drawing 3D parts we include them in a virtual assembly workbench in order to check that the design details are consistent from one part to another to make sure that the parts can be put together.
- *Documentation:* Most of Solid Modelling software packages include a drafting workbench to achieve definition drawings and bill of materials, which are useful for the manufacturing process, the final assembly of the system, the logistic and filing operations.
- *Production Planning:* Also we can simulate the machining operations and thus estimate the time necessary for the production and hence planning the different tasks.
- *Manufacturing Engineering:* Some of the CAD software enables to do a large number of manufacturing processes (welding, machining, bending, etc.). Thus we can import the part model in these workbenches and recreate the manufacturing operations.

In addition, the use of CAD improves the competitiveness of the company by reducing the lead time for a product, therefore it is a cost effective process that allows amortising investment costs more quickly. The daily evolution of the numerical methods provides tools with which to design complex and innovative products in teamwork. All these things make that the market shares and profits increase significantly.

Moreover, each new product needs to be prototyped and tested on test bench in order to check whether it respects the standards or not and ensure the safety. In this case, the CAE software plays a key role because the structural or fluid analysis allows testing the parts with several parameters and modifying them as many times as wanted without destroying prototypes. Nonetheless, that must not exclude the necessity of making at least one prototype but it reduces their number of them as well as the development costs.

Some of CAD packages are combined with a plug-in which adds advanced imaging functionality to create marketing pictures of 3D model. So, we can offer the customer unusual photos in order to convince him to buy the product.

The lifecycle management requires the prevention of maintenance actions in order to evaluate the time required to check the product and if necessary fix it, ensure its durability and to reduce the cost of down time. Using CAD software we can simulate the maintenance tasks by simulating in a virtual environment the operators repairing the product, thus we check whether they are able to do it or not in the work space.