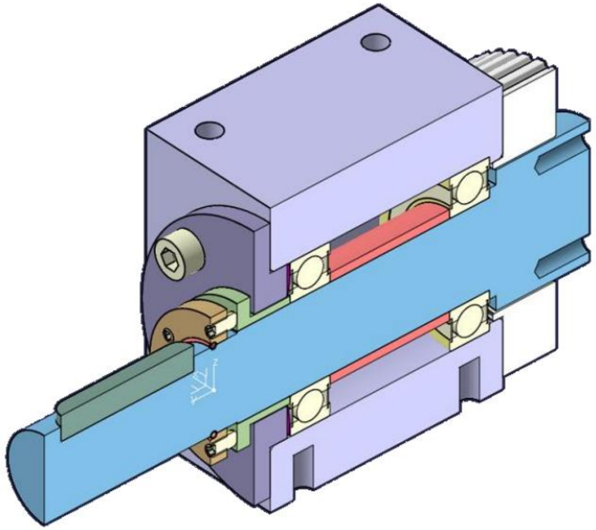
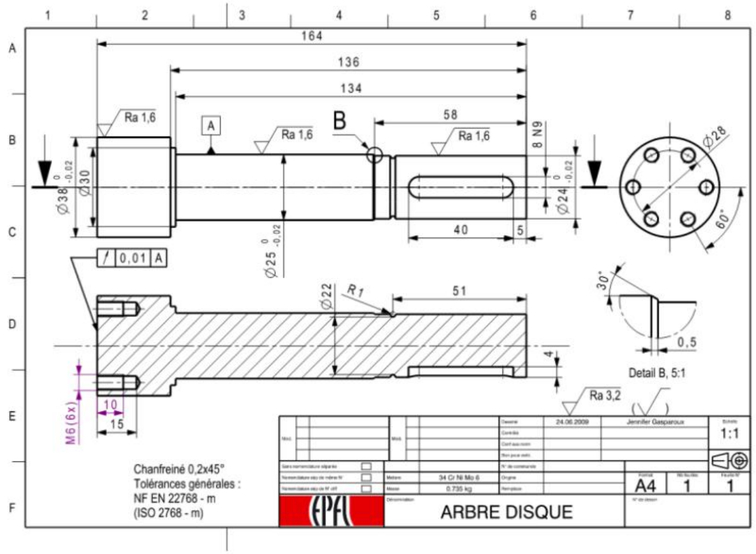
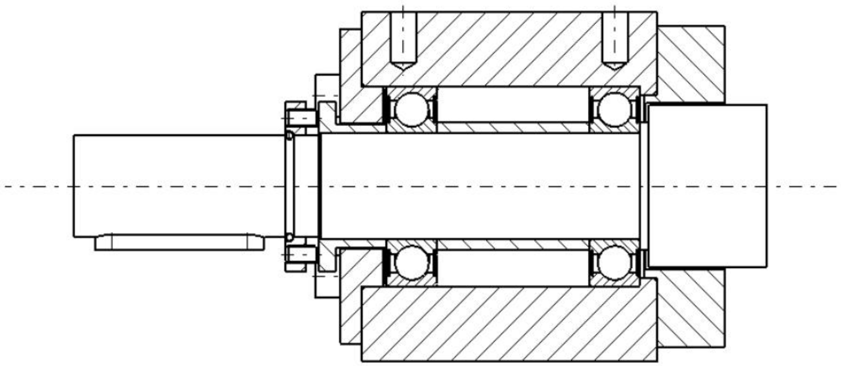
Introduction to Computer Aided Design (CAD) with Catia® V5 software

Exercises for students in: Mechanical Engineering 1st year
2nd year Materials Science and Engineering







Contents :

Theory :

Complementary exercises:

Guidelines for Assembly Management:

1. Introduction

This handout is a collection of guided exercises that allow the student to acquire the practical bases of the graphic representation of parts and Computer Aided Design (CAD). The work is done using *Catia® V5 software.* Based on a power transmission mechanism, it gives a procedure for drawing 3D parts, drawing 2D details for manufacturing, assembling 3D parts and drawing it for assembly. Some notions of methodologies are also presented on the design of parts as well as on the management of assemblies.

Thanks :

Benoît Carton, Mathieu Benoît, Thomas Göte McCarthy, Christophe Mattheeuws, Claude Ramseyer.

Writing – last update: Jennifer Gasparoux – August 2010

Contact: Jennifer Gasparoux, jennifer.gasparoux@epfl.ch
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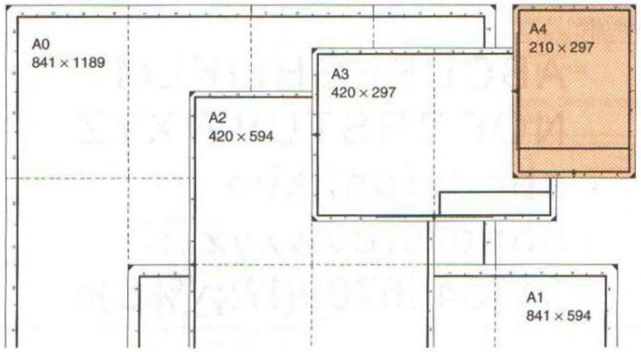
2. Graphic representation of the parts (source: Memotechplus: design and drawings)

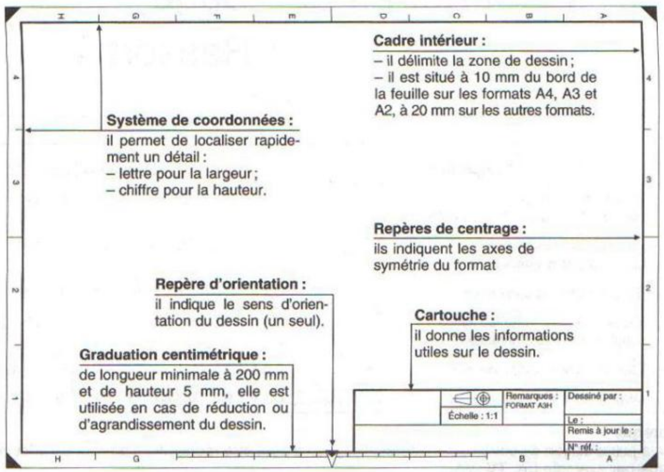
2.1 Drawing format:

Formats usuels

Les formats se déduisent les uns des autres à partir du format A0 (1 m²) en divisant le plus grand côté par deux.
Le rapport de la longueur sur la largeur est de $\sqrt{2}$.
Les formats peuvent être utilisés horizontalement ou verticalement.



Indications portées sur les formats



2.2 Inscription cartridge:

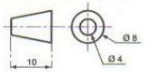
Emplacement

Lorsque la feuille support du dessin est examinée en hauteur pour les formats pairs (**A0**, **A2**, **A4**) et en largeur pour les formats impairs (**A1**, **A3**), le cartouche d'inscription doit toujours se trouver, en sa position de lecture, en bas et à droite, accolé au cadre extérieur du dessin (NF E 04-502).

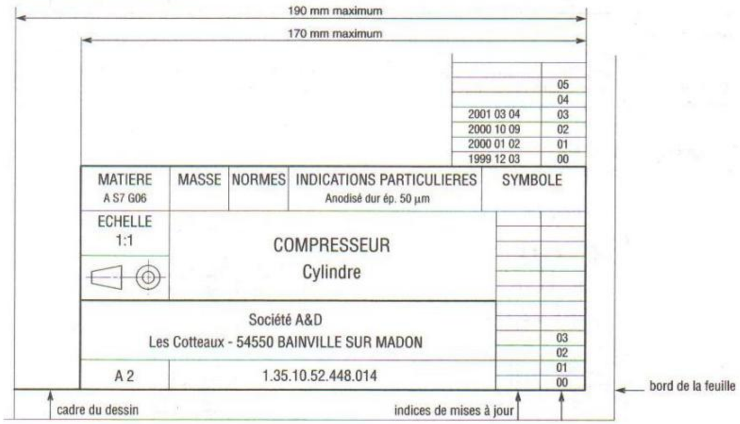
Dimensions

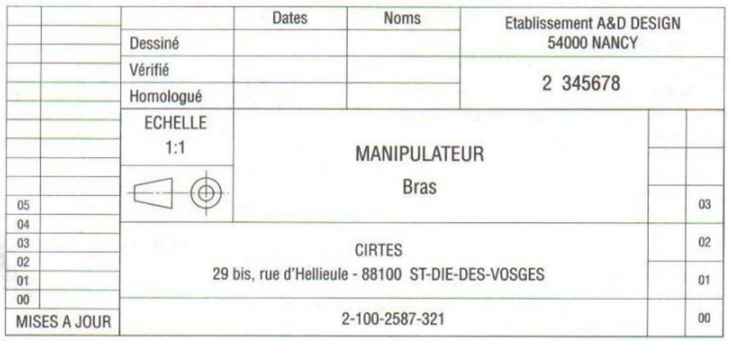
Le cartouche ne doit jamais dépasser en largeur 190 mm et en hauteur 277 mm.

Symbole de la disposition des vues



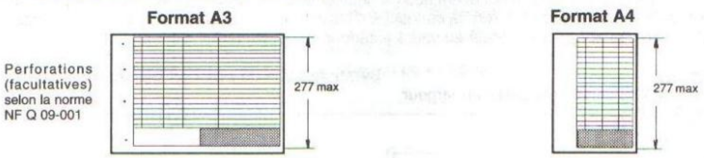
Dispositions et contenu





2.3 Definition nomenclature: (memotech)

Elle peut être disposée sur une feuille indépendante ou sur le dessin lui-même. Elle peut contenir autant de renseignements qu'il est jugé utile d'y porter.



2.4 Scales: (memotech)

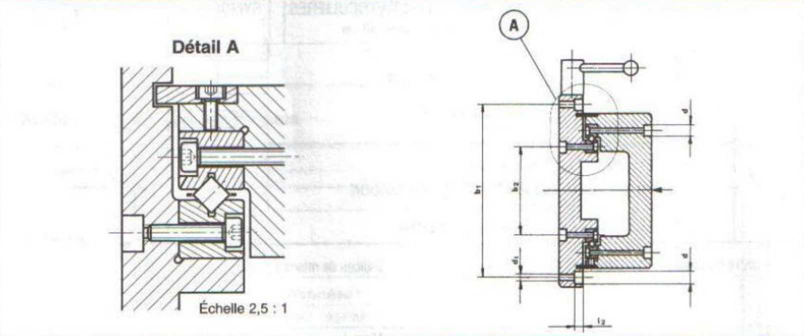
Désignation

Échelle 1 : 1 pour la vraie grandeur
Échelle X : 1 pour l'agrandissement
Échelle 1 : X pour la réduction

Inscription

Indiquer toujours l'échelle à l'emplacement prévu dans le cartouche.

Désignation particulière



Échelles recommandées

Catégories	Indications			
Échelles d'agrandissement	200 : 1	250 : 1	500 : 1	1000 : 1
	20 : 1	25 : 1	**50 : 1**	100 : 1
	2 : 1 (*)	2,5 : 1	**5 : 1**	**10 : 1**
Vraie grandeur (échelle recommandée)	**1 : 1**			
Échelles de réduction	**1 : 2** (*)	1 : 2,5	**1 : 5**	**1 : 10**
	1 : 20	1 : 25	**1 : 50**	**1 : 100**
	1 : 200	1 : 250	**1 : 500**	**1 : 1000**
	1 : 2000	1 : 2500	**1 : 5000**	**1 : 10000**

(*) Ces échelles ne sont pas recommandées car elles peuvent donner lieu à des impressions trompeuses à la conception.

Nota :
– Seules les échelles en caractères gras ont été retenues à l'ISO.
– Pour les dessins s'incorporant à des bâtiments, se référer à la norme NF P 02-002.

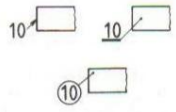
2.5 Element markers: (memotech)

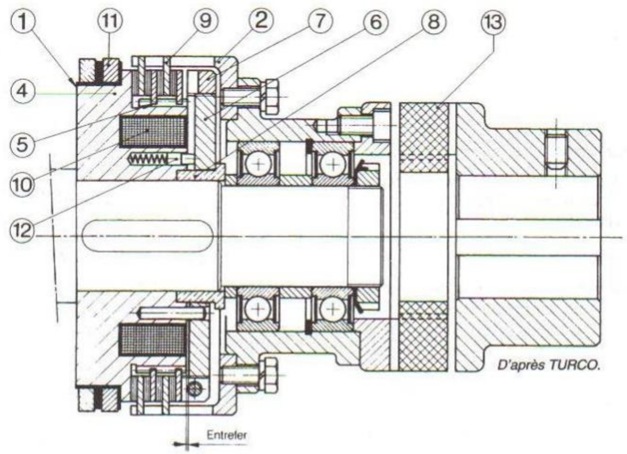
Spécifications générales

– Les repères sont attribués de façon successive à chacun des éléments composant un ensemble.
– Tous les éléments identiques d'un même ensemble doivent être identifiés par un même repère.

Représentation

– Les repères sont composés de chiffres arabes. Ils peuvent être complétés par une lettre majuscule (8A, 8B, 8C...);
– utiliser des caractères de plus grande hauteur d'écriture que celle utilisée pour la cotation par exemple;
– inscrire chaque repère à l'intérieur d'un cercle (ou comme indiqué ci-contre);
– disposer les repères en dehors du tracé général des éléments concernés;
– adopter un ordre déterminant :
- ordre numérique,
- ordre de montage possible,
- ordre d'importance (sous-ensemble, pièces principales, pièces secondaires...),
- tout autre ordre logique.



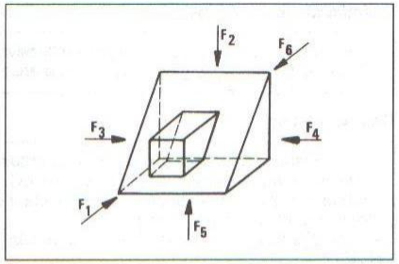


1. Partie motrice
2. Cloche d'embrayage
4. Électro-aimant
5. Disques intérieurs
6. Armature mobile
7. Écrou de réglage
8. Fourrure amagnétique
9. Disques extérieurs
10. Bobine fixe
11. Bagues collectrices
12. Poussoirs de rappel
13. Accouplement élastique

2.6 Arrangement of views: (memotech)

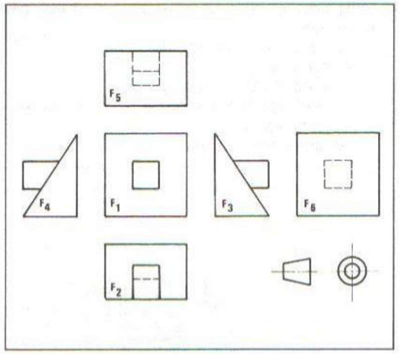
Dénomination des vues

Méthode de projection du premier dièdre :
- Vue suivant F_1 = vue de face
- Vue suivant F_2 = vue de dessus
- Vue suivant F_3 = vue de gauche
- Vue suivant F_4 = vue de droite
- Vue suivant F_5 = vue de dessous
- Vue suivant F_6 = vue d'arrière



Positions relatives des vues

Par rapport à la vue de face (F_1), les autres vues sont disposées comme suit :
- celle de dessus (F_2), au-dessous
- celle de dessous (F_5), au-dessus
- celle de gauche (F_3), à droite
- celle de droite (F_4), à gauche
- celle d'arrière (F_6) peut être disposée à droite de (F_3) ou à gauche de (F_4), indifféremment.

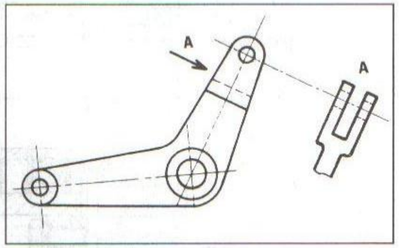


Vues particulières

La flèche indique le sens d'observation du dessin, par exemple lorsqu'une vue ne peut être disposée dans sa position normale.

Vues partielles

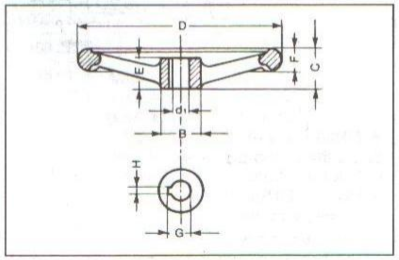
Si, dans une vue, la représentation de la totalité d'un élément n'est pas indispensable à la compréhension du dessin, la vue entière peut être remplacée par une vue incomplète (voir dessin de la biellette ci-contre).



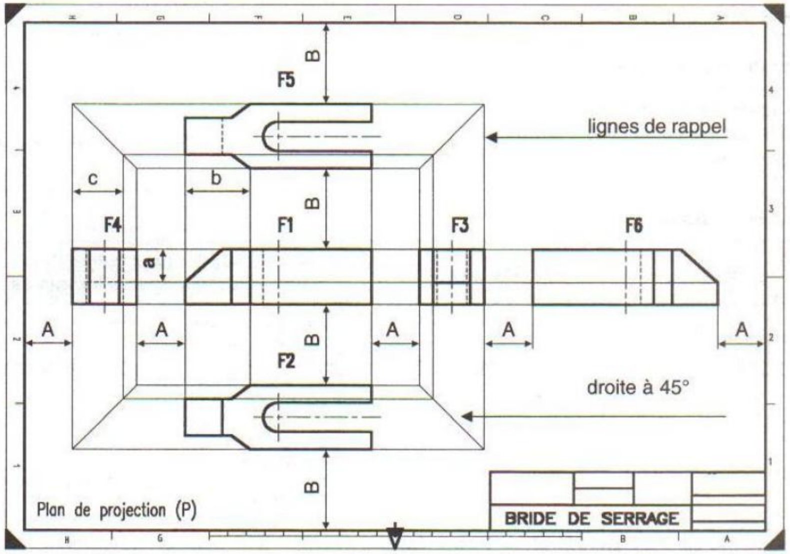
Vues locales

À condition que la représentation ne soit pas ambiguë, il est permis de se limiter à une vue locale à la place d'une vue complète.

Les vues locales doivent être dessinées en trait continu fort et doivent être reliées à la vue principale au moyen d'un trait mixte fin.



Correspondance entre les vues



Les vues, construites à partir des plans de projection perpendiculaires entre eux, sont alignées les unes par rapport aux autres.

On définit les trois règles de correspondances suivantes :

– Correspondances horizontales
Une dimension verticale sur la vue de face (exemple **a**) se retrouve verticale sur les vues de droite, de gauche et d'arrière.

– Correspondances verticales
Une dimension horizontale sur la vue de face (exemple **b**) se retrouve horizontale sur les vues de dessus et de dessous.

– Correspondances en équerre ou à 90°
Une dimension horizontale sur la vue de gauche ou de droite (exemple **c**) se retrouve verticale sur les vues de dessus ou de dessous.

Remarques
- Les lignes de rappel et les droites à 45° sont des aides efficaces lors de la construction de l'esquisse du dessin.
- Les cotes **A** et **B** indiquent le positionnement des vues dans le format. Elles se déduisent des dimensions « hors tout » de la pièce.

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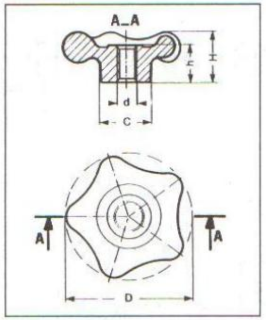
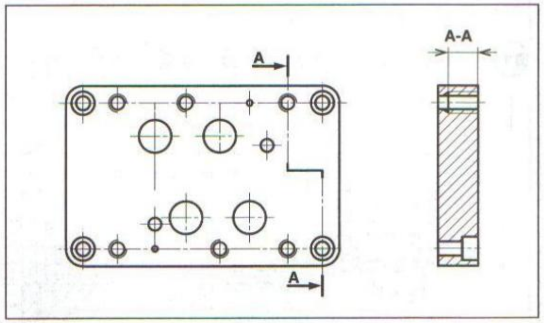
2.7 Cuts and sections: (mémotech)

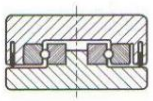
Coupe par un seul plan



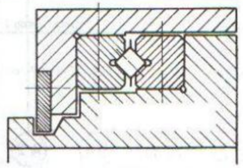
Coupe brisée à plans parallèles



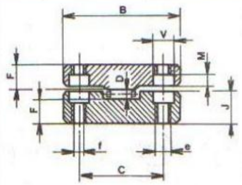
Généralités sur les hachures



L'intervalle entre les traits des hachures est fonction de la grandeur de la surface à hachurer.



Pour les surfaces importantes, les hachures peuvent être réduites.

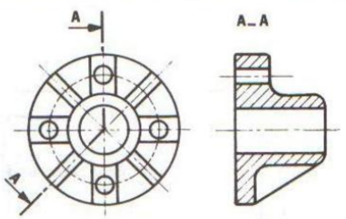


Interrompre les hachures pour placer une cotation.

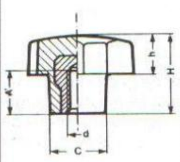
Sections de faible épaisseur

Les sections de faible épaisseur peuvent être noircies ou grisées entièrement. Ménager entre plusieurs sections contiguës noircies un espace blanc.

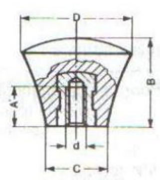
Coupe brisée à deux plans concourants



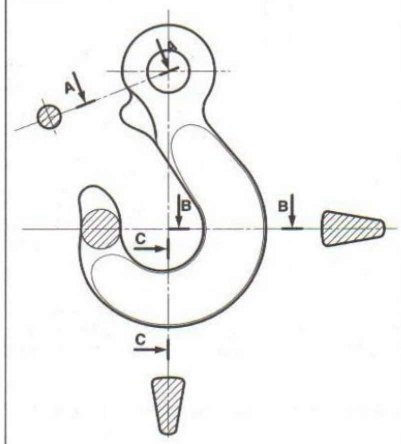
Demi-coupe



Coupe locale

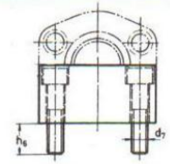


Sections rabattues sur place ou sorties



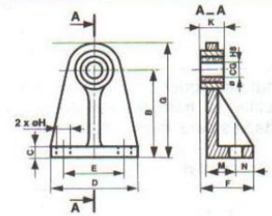
La section rabattue sur place doit être tracée en trait fin continu.

Demi-rabattement



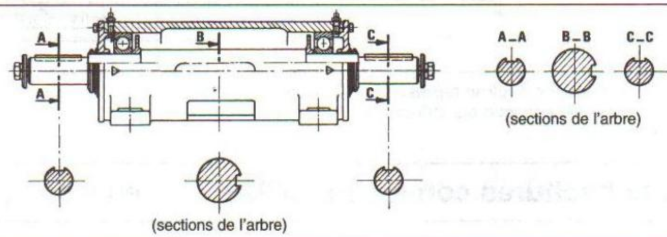
Exécuté en trait mixte fin à deux tirets.

Pièce nervurée

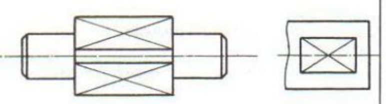


Coupe dans l'épaisseur de la nervure mais non hachurée.

Sections sorties successives (deux localisations possibles)

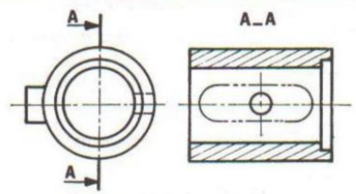


Surfaces planes



Surfaces planes en général Méplat sur cylindre

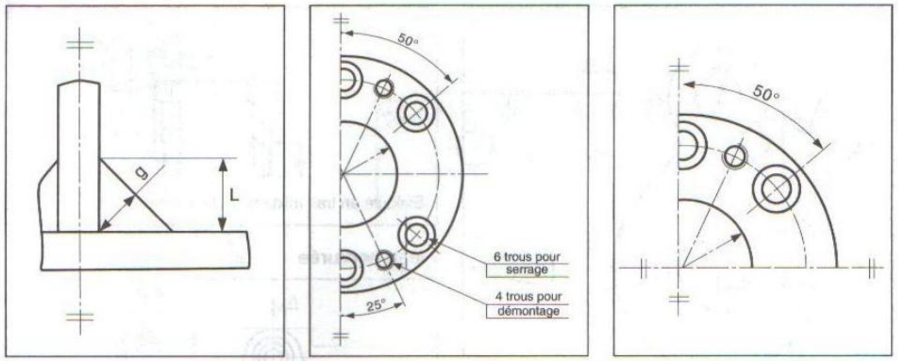
Parties situées en avant du plan de coupe



Si la représentation de parties se trouvant en avant du plan de coupe est nécessaire, les dessiner en trait mixte fin à deux tirets.

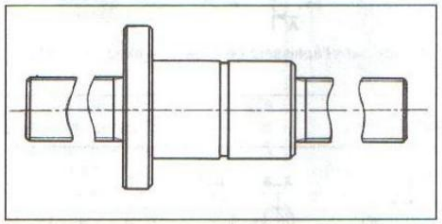
Cuts and sections: (continued)

Vues de pièces symétriques



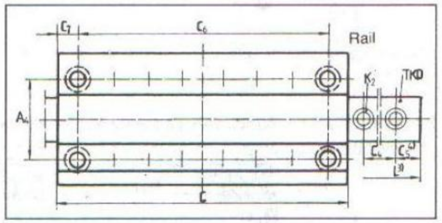
Dans le but de gagner du temps et de la place, on peut représenter les pièces par une fraction de leur vue complète. La trace du plan de symétrie doit être repérée à chacune de ses extrémités par deux petits traits fins parallèles perpendiculaires à l'axe.

Vues interrompues



Pour gagner de la place, on peut ne représenter que les parties d'une pièce longue qui suffisent à la définir.

Représentation simplifiée d'éléments répétitifs

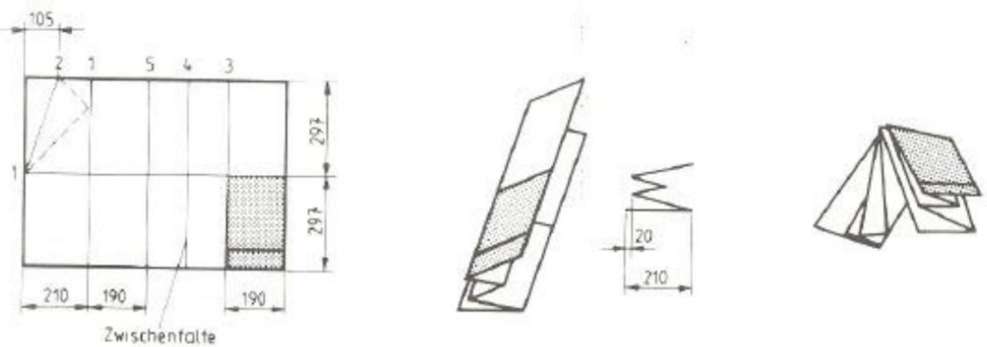


Position des usinages

2.8 Conventional hatching: (memotech)

Pour la représentation notamment des liquides, du sol naturel (roche), du sol aménagé, de la mousse de calfeutrement, voir NF P 02-001.

2.9 Folding of drawings (A0-A2):



3. Introduction to the different Catia software tools

F1 **key :** ***Access to Catia help*** (you need an internet connection to access it)

The main workshops are:

Part Design
Design of 3D parts.

Sketcher
2D sketch drawing

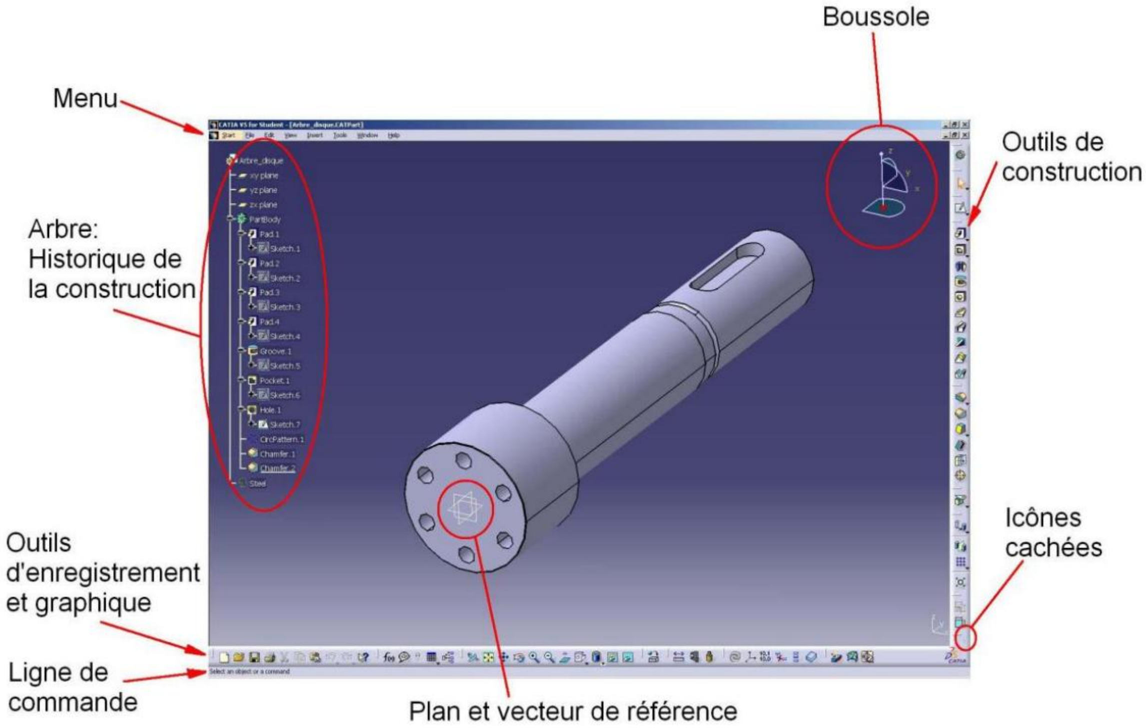
Drafting
2D drawing

Assembly Design
Assembly of parts.

3.1 Part Design

The Part Design workbench allows you to create 3D parts. The generated files are products with .CATPart extensions.

Presentation :



Handling :

Translate the piece: 

Rotate the part :  + 

Zoom :  +  + 

Toolbar: *(Only the most used tools are shown)*

Sketch

Components from a sketch:

Extrusion
Extrude with drafts and fillets
Multi-extrusion
Revolution
Hole
Groove
Multi-section solid

Poached
Pocket with drafts and fillets
Multi-pocket
Throat
Rib
Back smoothing

Skin components:

Edge fillet
Variable leave

Chamfer
Tapping

Processing Components:

Translation
Spin
Symmetry

Mirror
Rectangular repeat
Circular repeat

Miscellaneous components

Update
Catalog

Point
Right

Plan
Inserting a marker

3.2  Sketcher

The sketching workshop is inserted in the **part design workbench,** it is selected automatically when a sketch is made.

Outline tools:

Outlines
Rectangles
Oriented rectangles
Parallelograms
Oblong contours
Oblong arches
Keyholes
Hexagons
Points

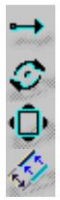
Basic circles
Circles by three points
Circles with coordinates
Tri-tangent circles
Arcs by three points
Three-point arcs using bounds
Basic Bows
Curves
Intersection

Ellipses
Parable
Hyperbola
Conical
Right
Infinite line
Axes

Operations tools:

Round
Chamfer
Cutting elements
Cutting elements
Eraser

Projection of 3D elements
Intersection of 3D elements
Projection of 3D silhouette lines
Symmetry



Translation
Spin
Scaling
Gap

Constraints tools:

Constraints from a Dialog
Dimensional constraints

Animating Constraints

Visualization of constraints:

Perpendicular
coincide
Concentric
Vertical
Horizontal

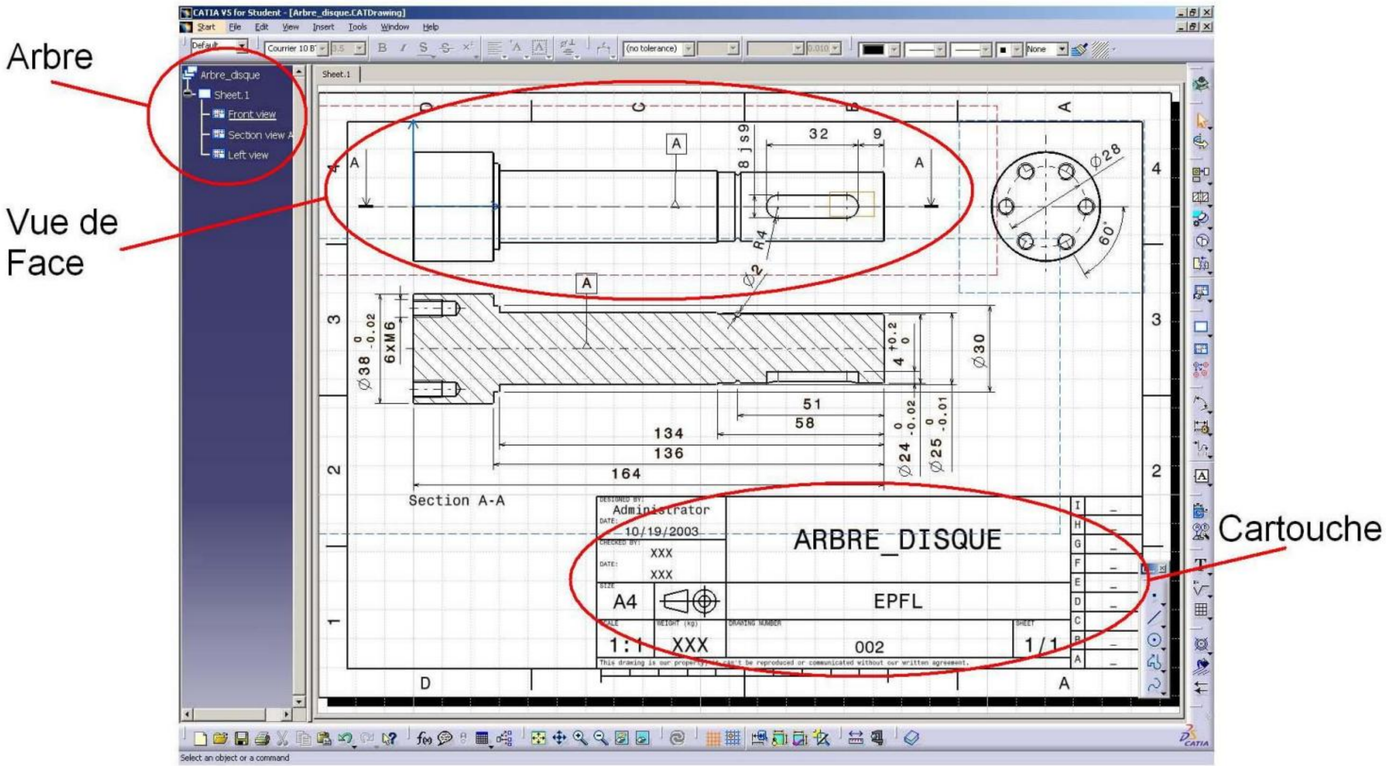
Fixed
Parallel
R 25 Ray
D 50 Diameter

Element colors in sketches:

White	Fluent
Red	Selected
Green	Constrained
Purple	Over-constrained
Brown	Incoherent

3.3  Drafting

The Drafting workbench allows you to produce 2D parts. Generated files are products with .CATdrawing extensions



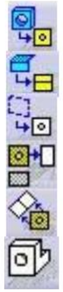
Toolbars: *(Only the most used tools are shown)*

Cartridge

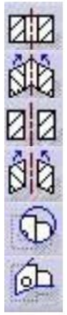
Creating a frame and title block

Creating a parts list

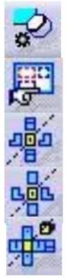
Views :



Front view
Unfolded view
View from 3D
Projected view
Auxiliary view
Isometric view



broken cup
Unfolded cup
Broken section
Unfolded section
Clipped view
Clipped view with profile



Detail view with profile
Setting up views with the wizard
Front, top, left view
Front, bottom, right view
All Views

Quotes:

Odds
Cumulative quotes
Stacked dimensions

Length dimensions
Angle dimensions
Radius dimensions

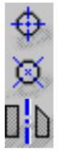
Chamfer dimension
Tapping dimensions
Diameter dimensions

Tolerances:

Reference

Geometric tolerances

Dressing:



Center line seen from the front without reference
Center line seen from the front with reference
Center line seen from ends

Thread without reference
Thread with reference
Center lines seen from the front and seen from the ends

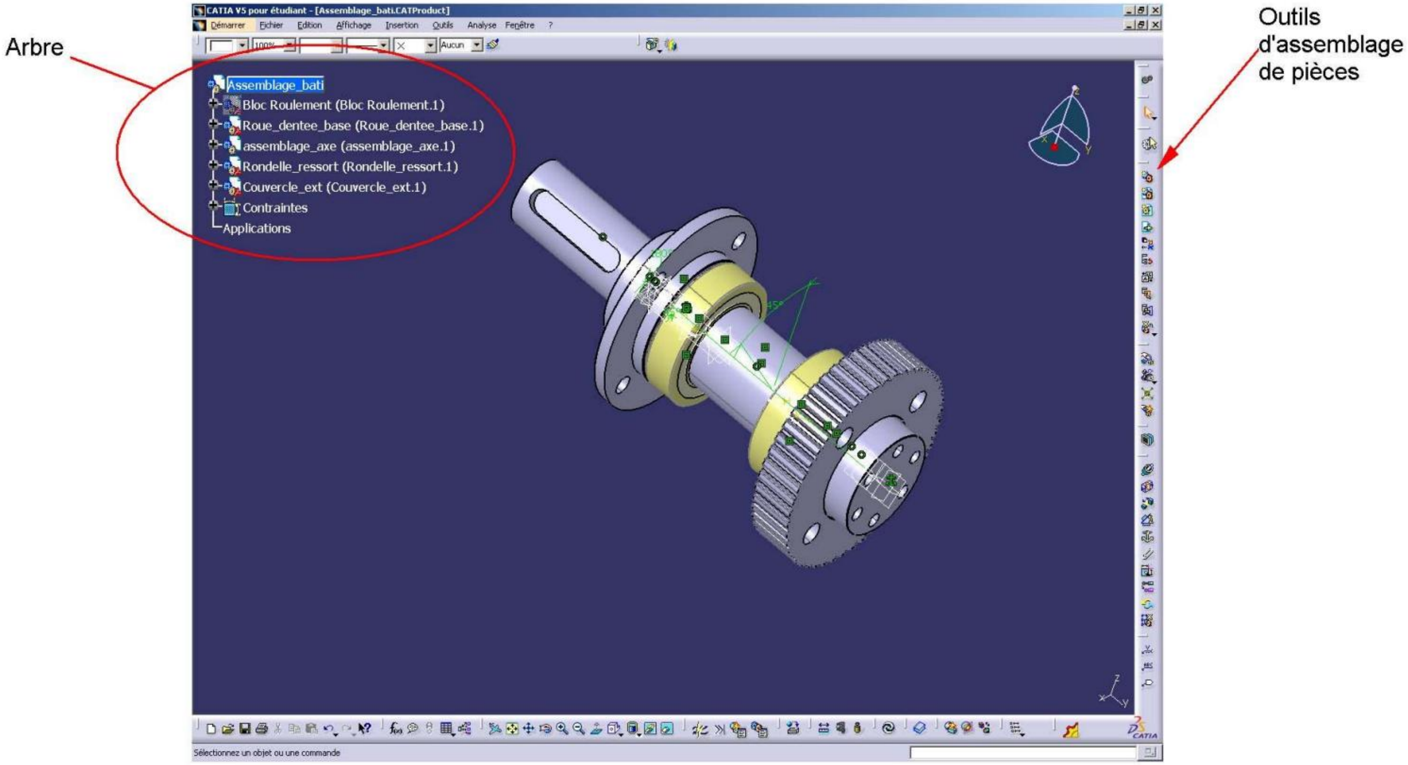
Notes:

Text
Attached text
Copy of text
part number
Partial reference
Table
CSV table

Roughness symbol
Weld symbol
Welding

3.4 Assembly Design

The assembly workshop allows to assemble already existing parts or to create parts from this workshop. The generated files are products with .CATProduct extensions.



Toolbars: *(Only the most used tools are shown)*

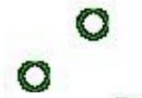
Structure :

Inserting a new component
Inserting a new product
Inserting a new part
Inserting an existing component

Constraints:

Coincidence
Contact
Distance
Angular
Fixed
Relative fixing
Constraint command
Reusing a pattern

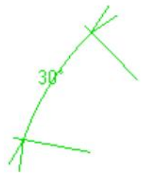
Constraints on the assembly: Symbol in the tree

Coincidence →


Contact →


Gap →

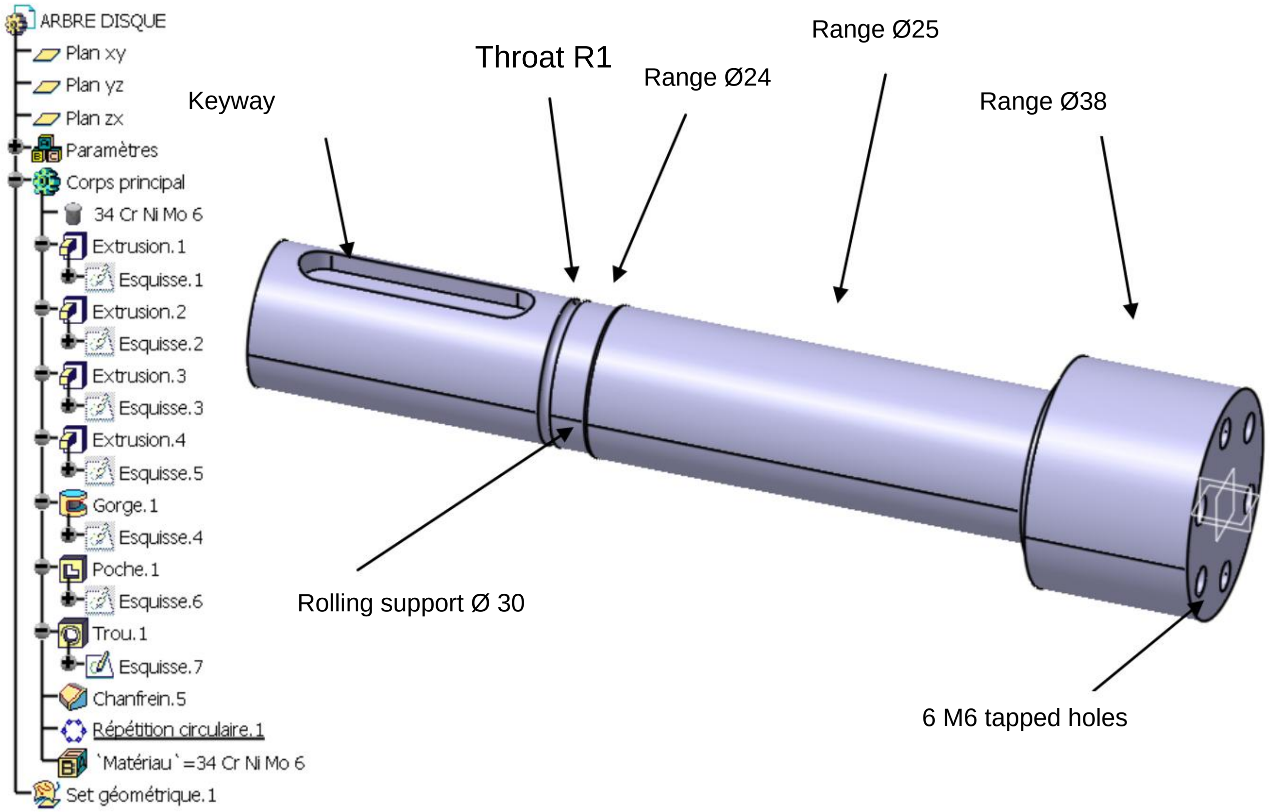

Perpendicularity →


Angle →


Parallelism →


To stare →


4. BASIC EXERCISE 1: 3D DISK TREE



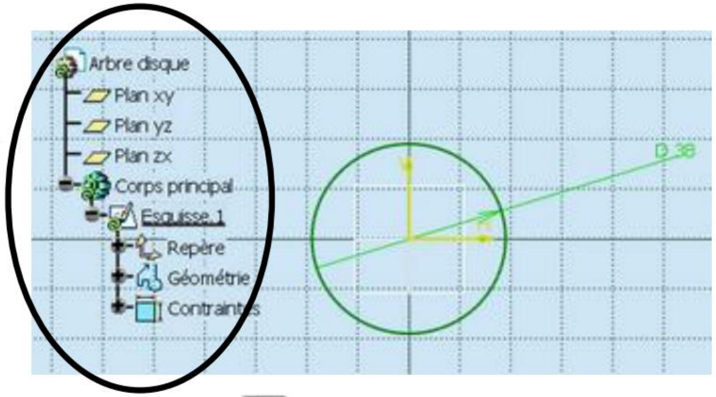
FINAL RESULT

Note : the symbol "double click on the left mouse button" means "click" once with the left mouse button"; **2x** In this document, means

Start Menu > **Part Design** > enter part name: **DISK TREE** > OK
File menu > **Save** > (default location, or as indicated) > file name: **DISK TREE** > Save

Creation of the span Ø 38 mm length 28 mm

ZX plane in tree view > **SKETCH**



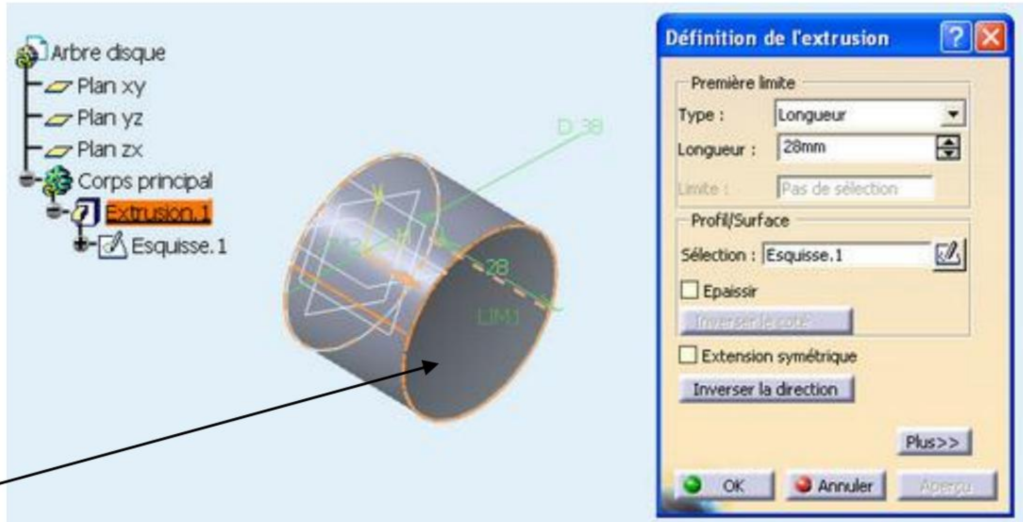
CIRCLE > place the center on the origin = approach the origin with the cursor until the symbol appears (coincidence between two points), to set the center > circle anywhere in the window to put the of any diameter.

The circle is red, indicating that it is selected. Otherwise (white circle), select it : The circle.

CONSTRAINT > set dimension (anywhere in the window) > **2x** on the diameter value, enter **38 >** OK

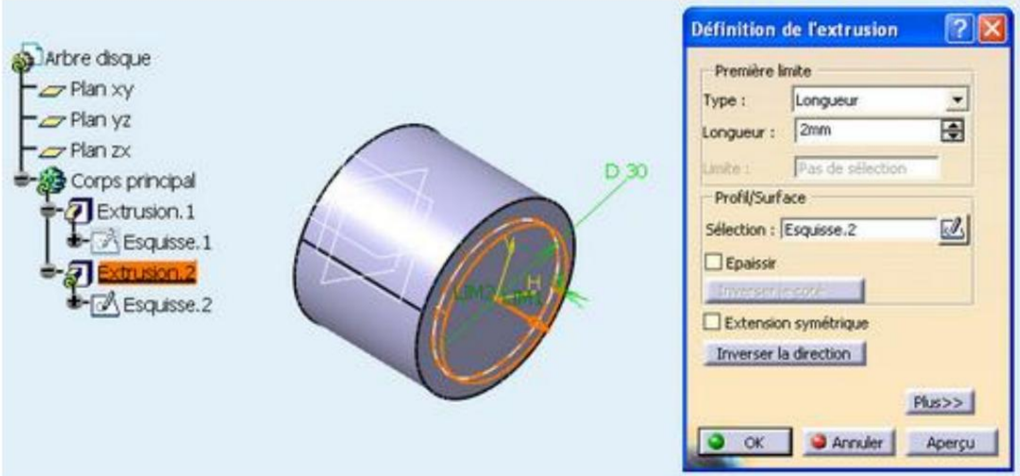
LEAVING THE WORKSHOP

EXTRUSION > Type: Length > **Length:** 28 > **Selection:** select the last sketch created, here **Sketch.1** (if not already selected by default) > OK



Face to select as sketch plane
for next step

Range Ø 30 mm for bearing support



face of the Ø 38 span that we want as **the sketch plane** (see figure p 20). The contours of this surface turn red.

SKETCH

CIRCLE > center on origin > any (circle turns red) diameter

CONSTRAINT > set the dimension > **2x** on the diameter value > enter **30 >** OK

LEAVING THE WORKSHOP

EXTRUSION > Type : Length > enter **2** > select the last sketch created (if it is not already selected by default) > (reverse the direction if necessary, so that the extrusion is indeed created outside the solid already existing) >OK

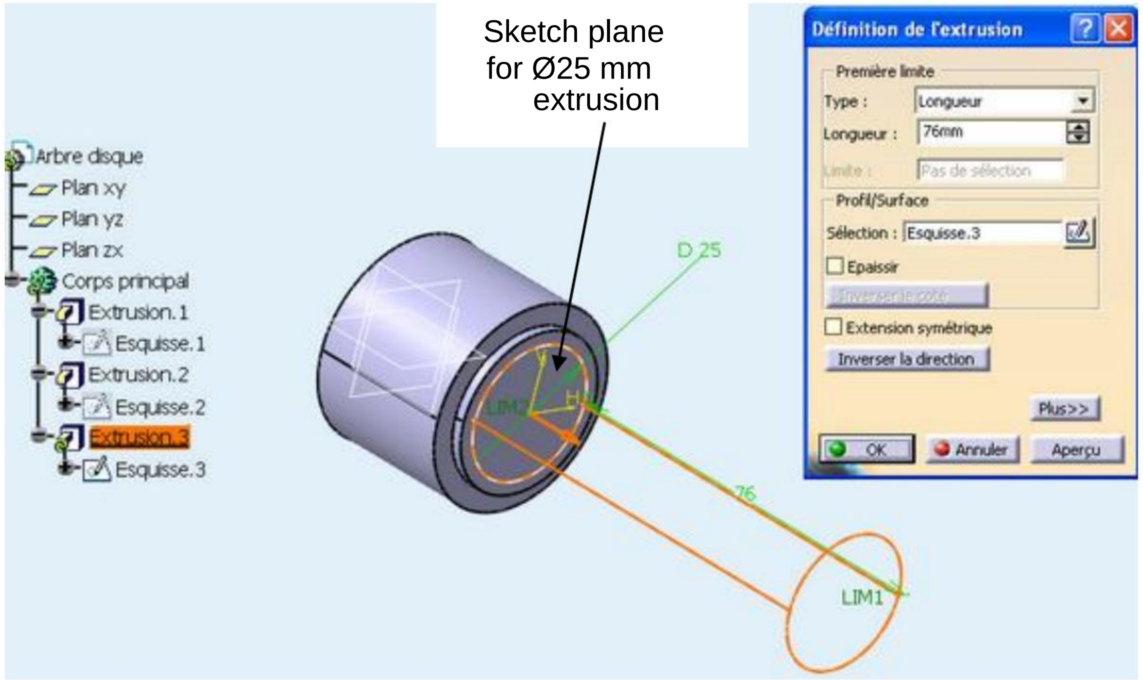
Note : to display the object (either the sketch or the 3D object) centered on the screen, with an optimal zoom, several solutions:

- Press the key Home of the keyboard

- **View** menu > **Center All**

- **CENTER ALL** (bottom of screen)

Range Ø25 mm length 76 mm



the face of the cylinder Ø 30 that we want as **a sketch plane**

SKETCH

CIRCLE > center on origin > any diameter (the circle turns red)

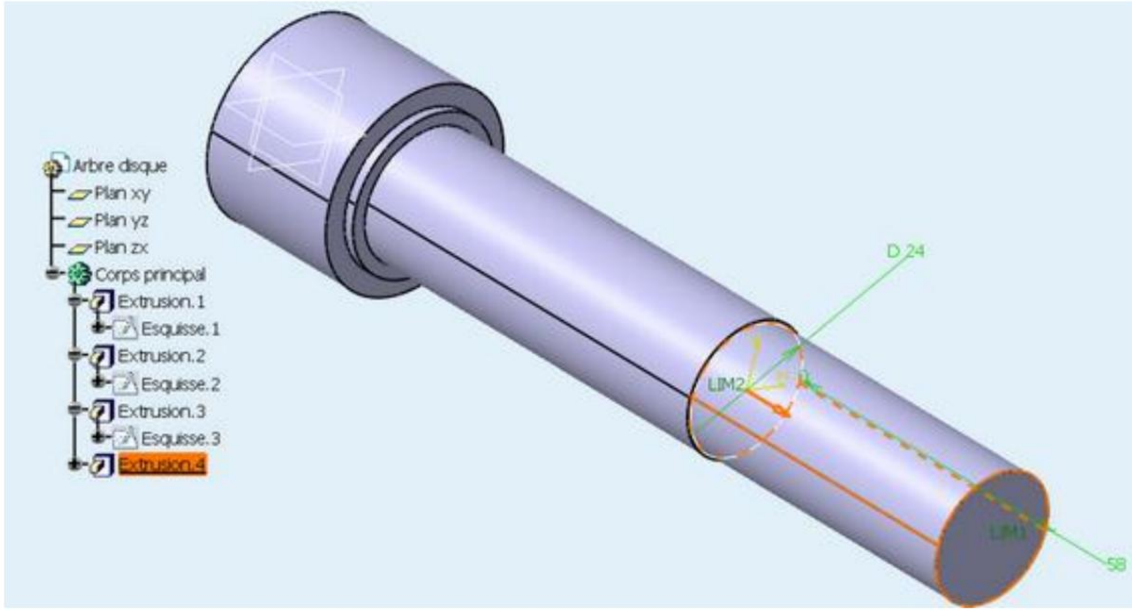
CONSTRAINT > set dimension > **2x** on the diameter value > enter **25 >** OK

LEAVING THE WORKSHOP

EXTRUSION > Type : Length > enter **76** > select the last sketch created, here **Sketch.3** (if it is not already selected by default) > (reverse the direction if necessary, so that the extrusion is well created outside the already existing solid) > OK

Note : Practice moving the piece using mouse shortcuts (see page 10)

Range Ø 24 mm length 58 mm



face of the Ø 25 cylinder that we want as **the sketch plane**

SKETCH

CIRCLE > center on origin > any diameter (the circle turns red)

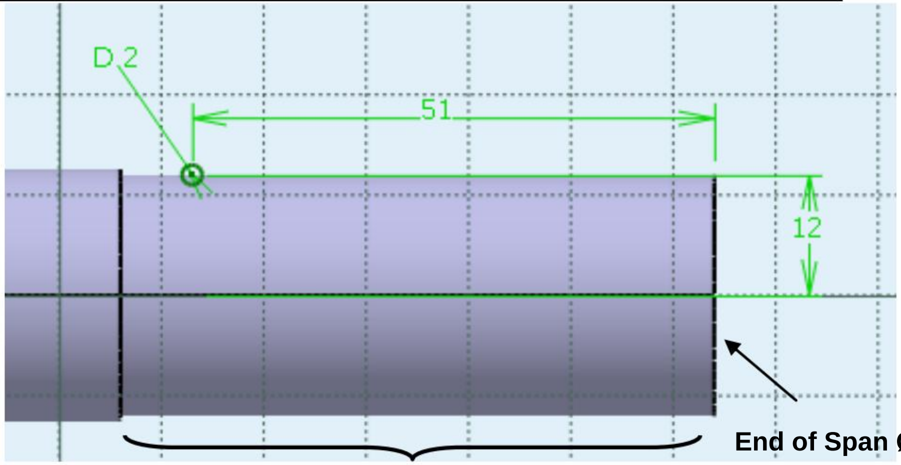
CONSTRAINT > set the dimension > **2x** on the diameter value > enter **24 >** OK

LEAVING THE WORKSHOP

EXTRUSION > Type : Length > enter **58** > select last created sketch (reverse direction if necessary) > OK

Note : remember to save your work regularly **(Ctrl +** S)…

Throat Ø 22 mm; R 1mm



YZ plane in tree view > **SKETCH**

CIRCLE > the center anywhere (for better visibility, outside the room) > any diameter.

next to the circle (the circle turns white)

CONSTRAINT > end of span Ø 24 > circle center > set dimension > **2x** on the odds value > enter **51 >** OK

CONSTRAINT > on the axis of the scope Ø 24 > the center of the circle > set the dimension on the odds value > **2x** enter **12 >** OK

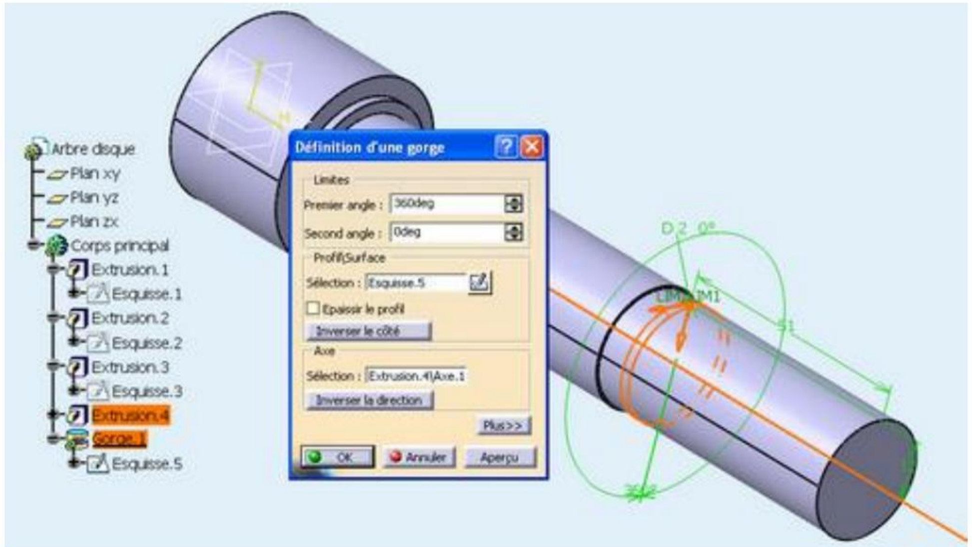
CONSTRAINT > circle > set dimension > **2x** diameter value > on the enter **2 >** OK

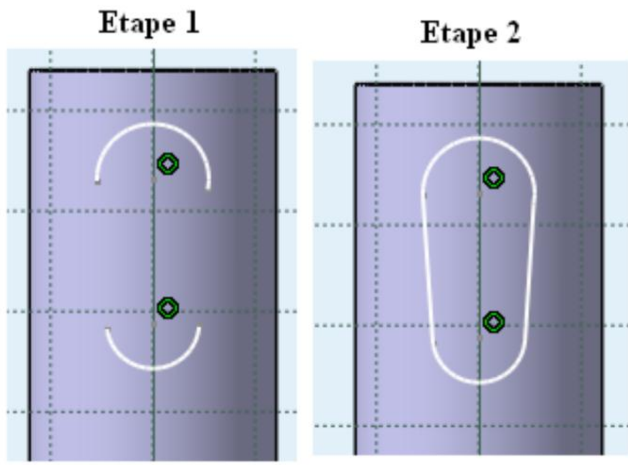
LEAVING THE WORKSHOP

GORGE

First angle: 360 deg, **Second angle :** 0 deg > Profile/Surface, **Selection :** last sketch created **(Sketch.5)** > Axis, **Selection :**

the axis of revolution of the Ø 24 span > OK





Keyway on Ø 24 seat

Creating the key outline using two arcs and two straight lines.

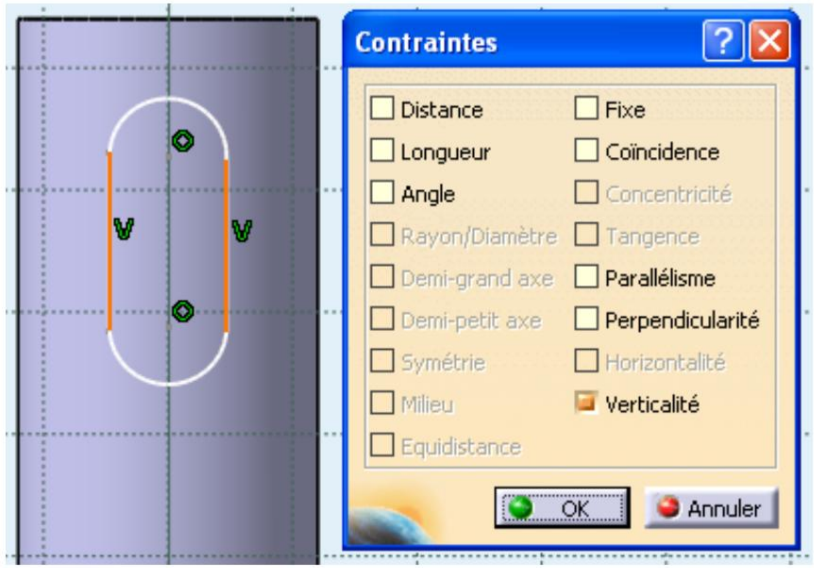
XY plane in tree view > **SKETCH**

ARC (hidden behind the **CIRCLE** symbol) > anything where on the staff axis Ø 24 for the center (the symbol appears when the mouse passes on the axis) and **2** points for the ends of the arc.

CRA > anywhere on the staff axis Ø 24 for the center (symbol) and **2** points for the ends of the arc.

Note : in some cases, you may not want to create automatic constraints such as the coincidence constraint represented by the symbol. .In this case, the **Shift** button must be held down while the sketch is being made.

RIGHT > draw two lines connecting the extremities of each of the 2 bows. To be sure that the extremities of the straight lines coincide well with the extremities of the arcs, bring the mouse close enough to the point to see the symbol appear.
before clicking on the left to set the point on the line.



Keyway Stresses & Dimensioning

on one line > keep the **Ctrl** key pressed and on the other line (the two selected elements turn red) > release the **Ctrl key.**

SELECTED CONSTRAINTS IN A DIALOG BOX
enable **Verticality** > OK

on a straight line > keep the **Ctrl** key pressed and on an arc (the two selected elements turn red) > release the **Ctrl key.**

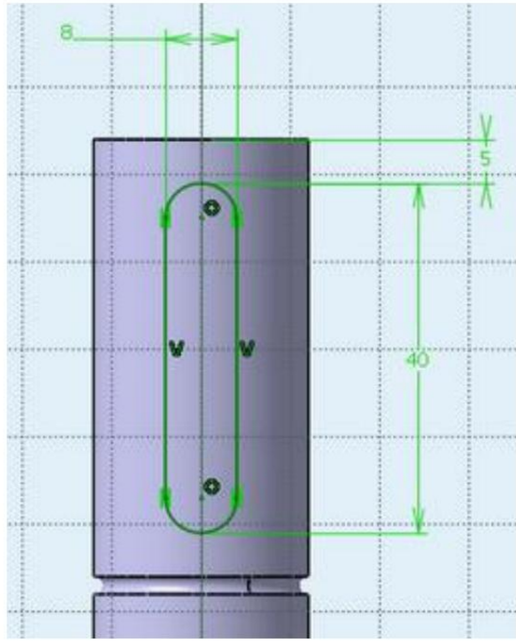
SELECTED CONSTRAINTS IN A DIALOG BOX
enable **Tangency** > OK

Repeat the operation until the 4 extremities of the lines are tangent to the arcs (the symbol must appear for each tangency).

CONSTRAINT > end of span Ø 24 > nearest arc > place the odds > **2x** on the dimension value > enter **5 >** OK

STRESS > dimension a bow > the other arc > set the dimension > **2x** on the value > enter **40 >** OK

CONSTRAINT > a line > the other right > set the dimension > **2x** on the dimension value > enter **8 >** OK



IMPORTANT: If the ends of the lines and the arcs do not coincide:

on the end of the line > keep the **Ctrl** key pressed and (the two selected points turn red) > release the **Ctrl key.** on the end of the arc

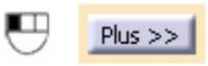
SELECTED CONSTRAINTS IN A DIALOG BOX >
activate **Coincidence** > OK

Repeat the operation until the curve is closed.

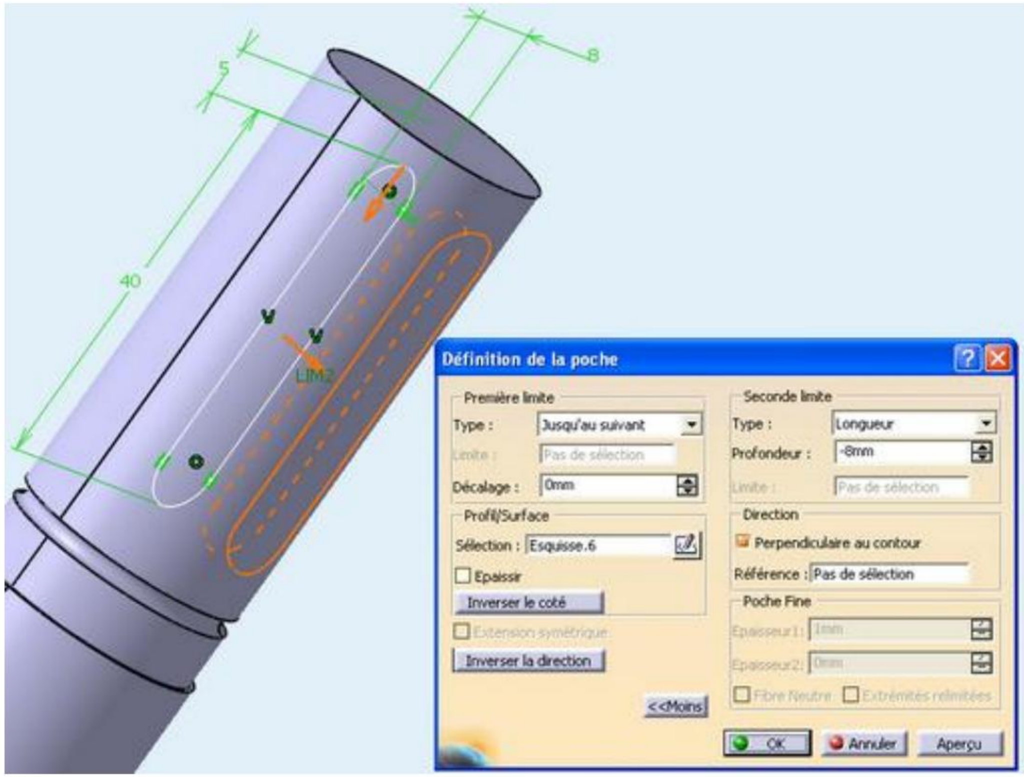
***IMPORTANT:* NOTES** - The
sketch is **fully constrained** if all the lines are **green** - If some lines
are **white,** the sketch is not fully constrained: the entire sketch must then be constrained by setting the missing dimensions and constraints.

- If some lines are in **purple,** the sketch is over-constrained: the over-constraints must be eliminated ~~as **soon as they appear.**~~

 **LEAVING THE WORKSHOP**

 **POCKET >** 

First Limit / Type: Until Next >
Second limit / Type: Length > Depth: enter **- 8** > OK



Note : When no second limit is entered, the pocket starts from the plane chosen to draw the sketch (in this case, the XY plane located at the center of the part) up to the **First limit.**
When you enter a positive value for the **Second limit,** the pocket starts before the sketch plane.
When we enter a negative value (our case) for the **Second limit,** the pocket starts after the sketch plane.

6 M6 tapped blind holes

HOLE

face of the Ø38 span (the hole appears in red), if necessary rotate the part to see the face in question appear (see page 10)

green arrows of the hole (hold down the button) and move the hole so that it is no longer in the center of the room.

Extension tab : Blind > Bottom : V -shaped

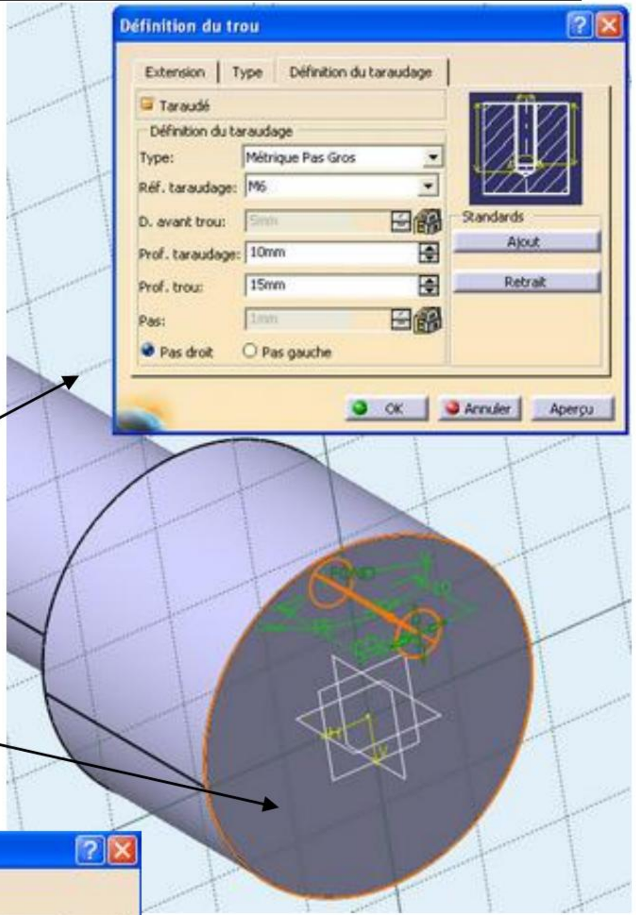
Type tab : Simple

tab : **Thread definition :**
activate:

Threaded Type : Metric
Coarse Pitch **Ref. thread :**
M6 **Prof. thread :** 10 mm
Depth. hole : 15 mm
Enable: **Not straight**

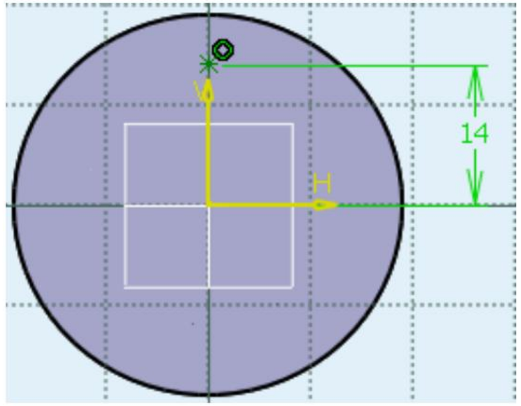
Extension tab >

Editing the sketch



Face chosen to place the hole

vertical axis **V** > hold down the **Ctrl** key > **Ctrl** key dot (both selected items turn red) > drop the



SELECTED CONSTRAINTS IN A DIALOG BOX

Coincidence > OK (the symbol appears)

CONSTRAINTS > the horizontal axis **H** > on the point > set the dimension >

2x on odds value > enter **14** (If needed **Plus** and **Toggle)** **>** okay

LEAVING THE WORKSHOP

>OK

Circular repeat

Hole.1 in tree view (hole turns red)

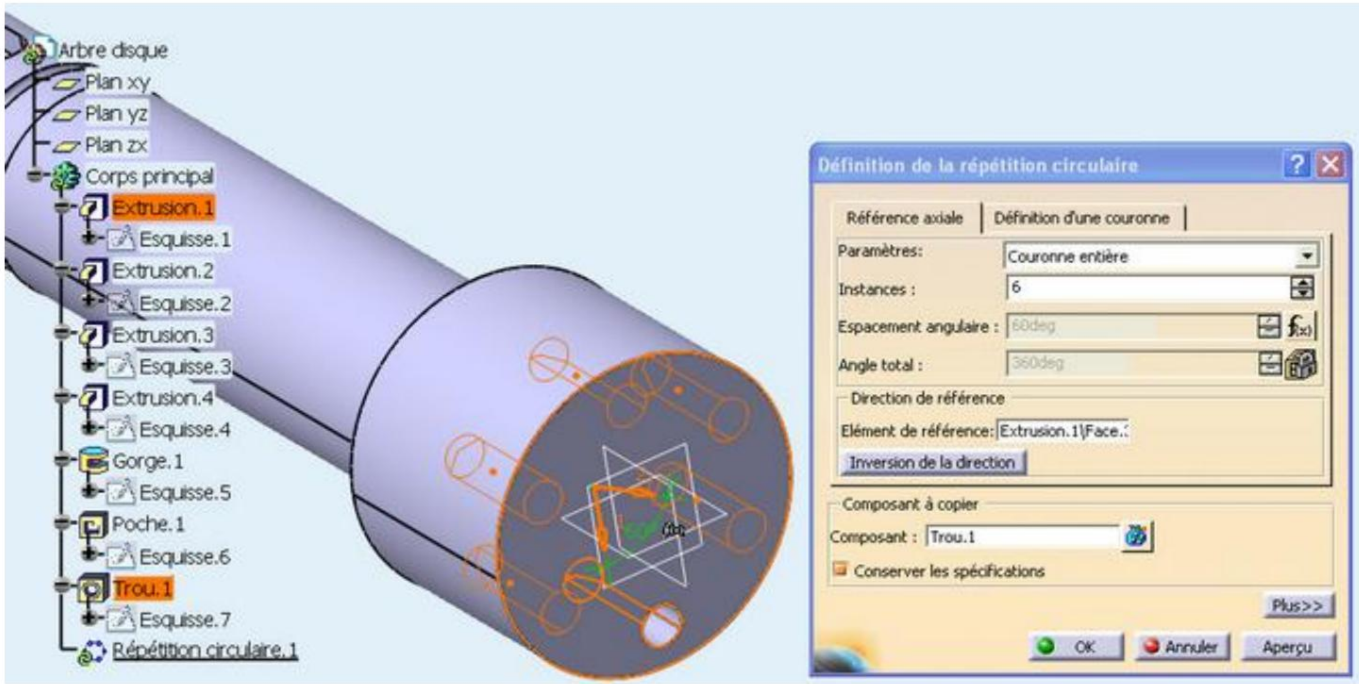
CIRCULAR REPEAT

(rear)

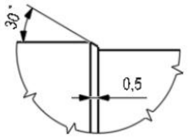
tab: **Axial reference**
Parameters: Whole Crown
Instances : enter **6**

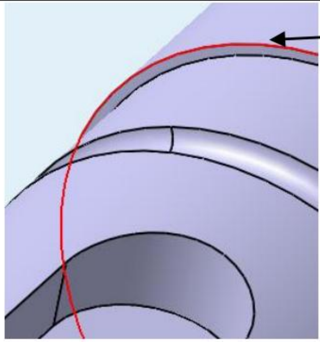
Reference direction : the face of the Ø38 span enable: **Keep specifications**

>OK

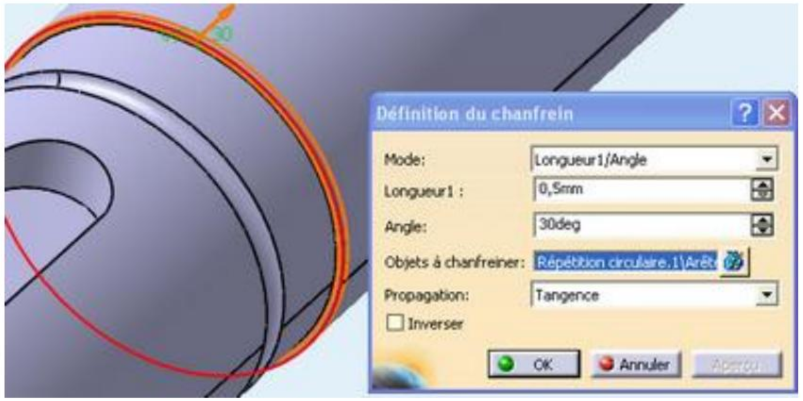


Chamfer 0.5 x 30°





Chamfering edge



edge to be chamfered (the edge turns red)
(do not hesitate to zoom in to select the edge)

CHAMFER

Mode: Length1/Angle
Length 1: enter **0.5**
Angle : between **30 >** (reverse direction if necessary) > OK

APPLICATION OF MATERIALS

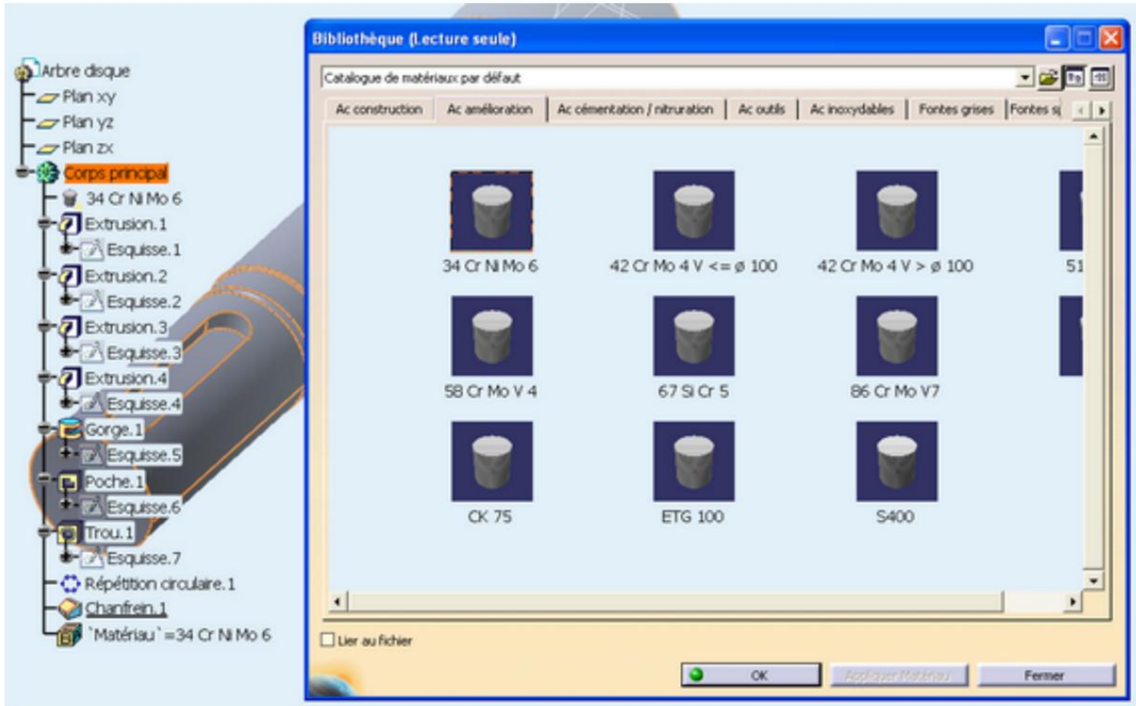
APPLICATION OF MATERIALS

Ac improvement tab > 34Cr Ni Mo 6

Main body in tree view (part outline turns red)

Apply Material > OK

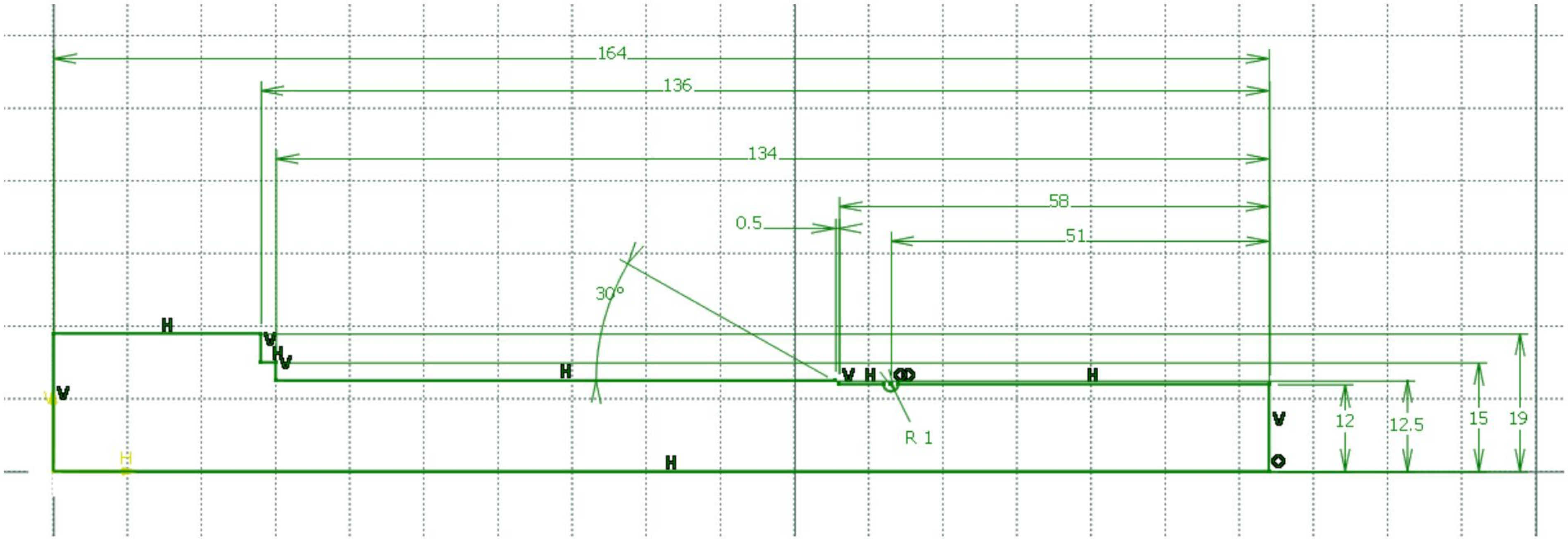
File menu > Save



Note The
tree was made by drawing one staff after another, we could have drawn it in one step using the revolve function .

Start menu > **Part Design** > enter the name of the part: **TREE DISK 2** > OK
File menu > **Save** > (default location, or as indicated) > file name: **TREE DISK 2** > Save

YZ plane in tree view > **SKETCH**



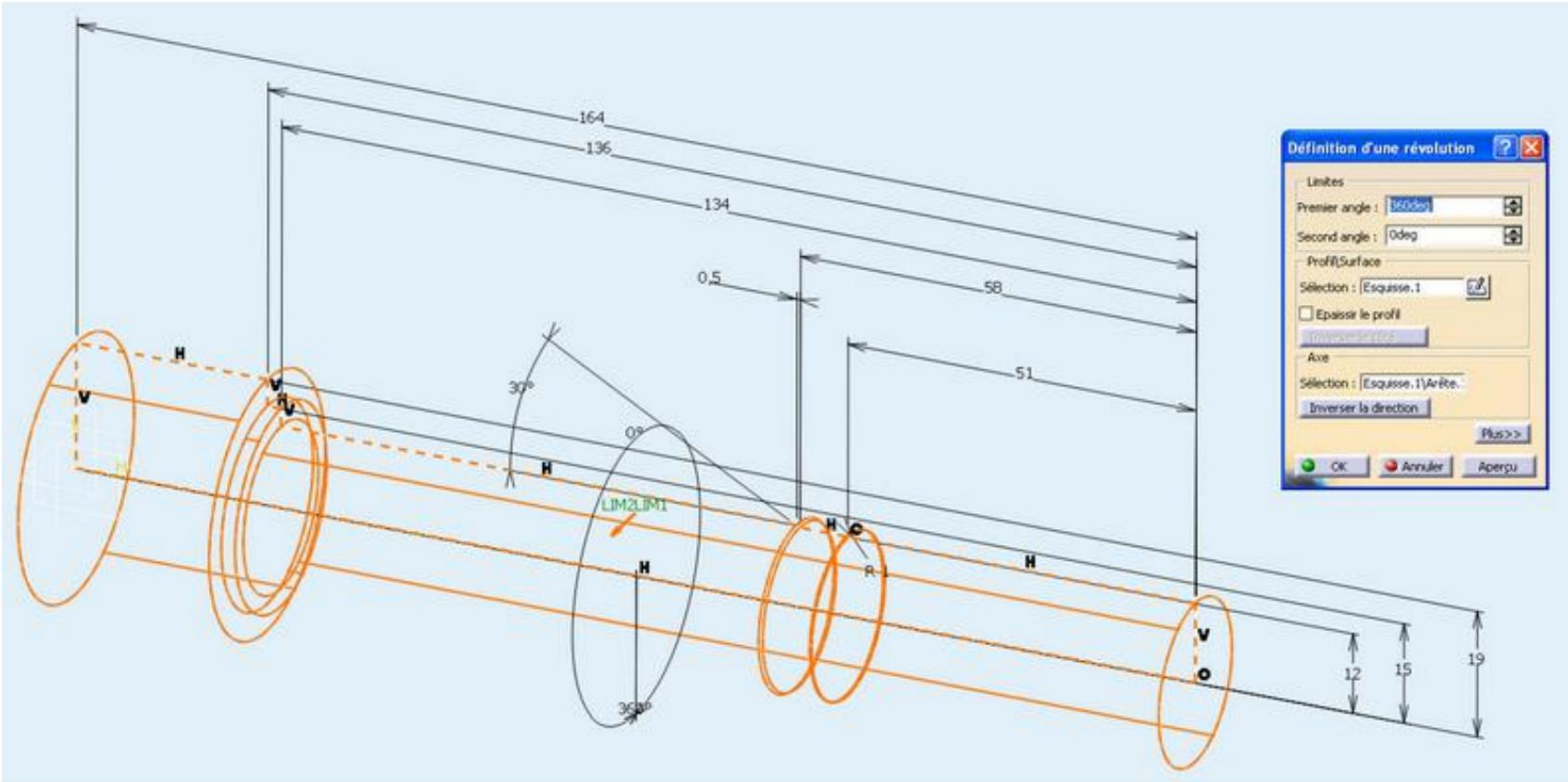
Using the drawing tools, draw the half-profile of the tree and set the necessary dimensional constraints.

Help: the "outline" icon allows you to draw the outline in one go ; **2x** keeps the icon (for the deactivate), this function also works for other icons, such as)

LEAVING THE WORKSHOP

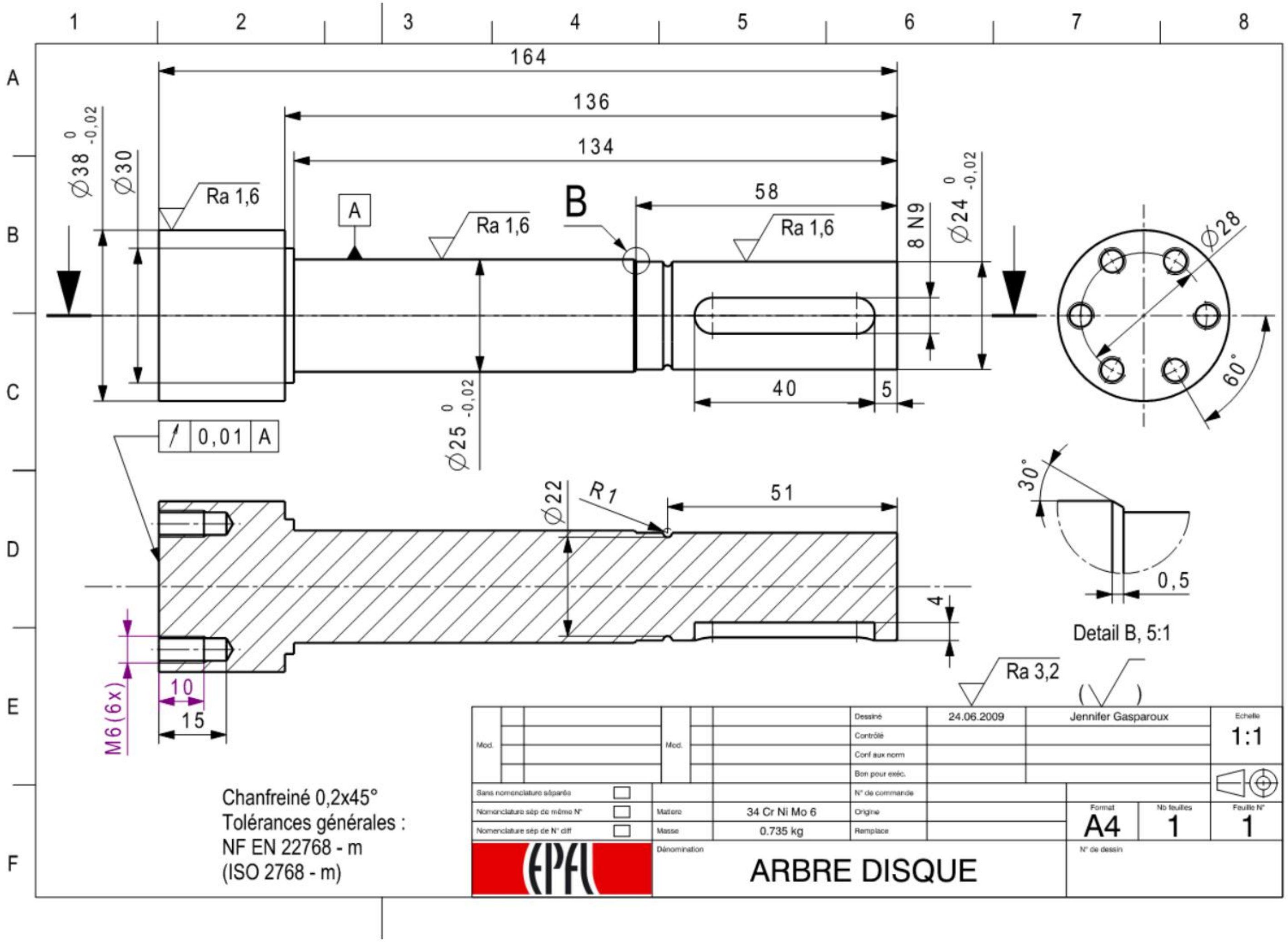
REVOLUTION

Axis > Selection: select the axis of revolution (the H axis) > OK



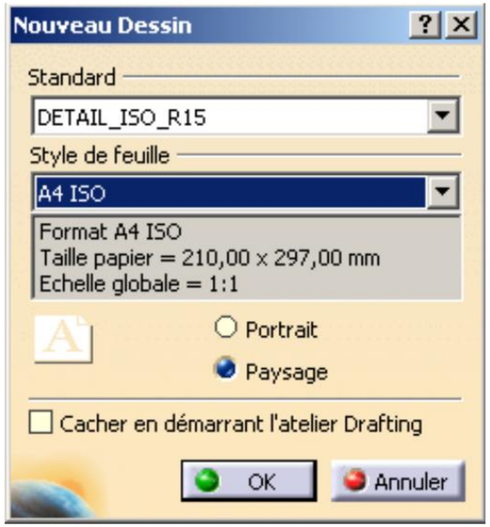
We thus obtain the disk tree in a single step. All that remains is to draw the holes and the groove as before.

5. BASIC EXERCISE 2: 2D DISK TREE



FINAL RESULT

File menu > **New** > **Drawing > OK >** Standard : **DETAIL_ISO_R18 >** Sheet style: **A4 ISO** > **Landscape** > OK
File menu **> Save > L: catia** > file name : DISK **TREE** > OK

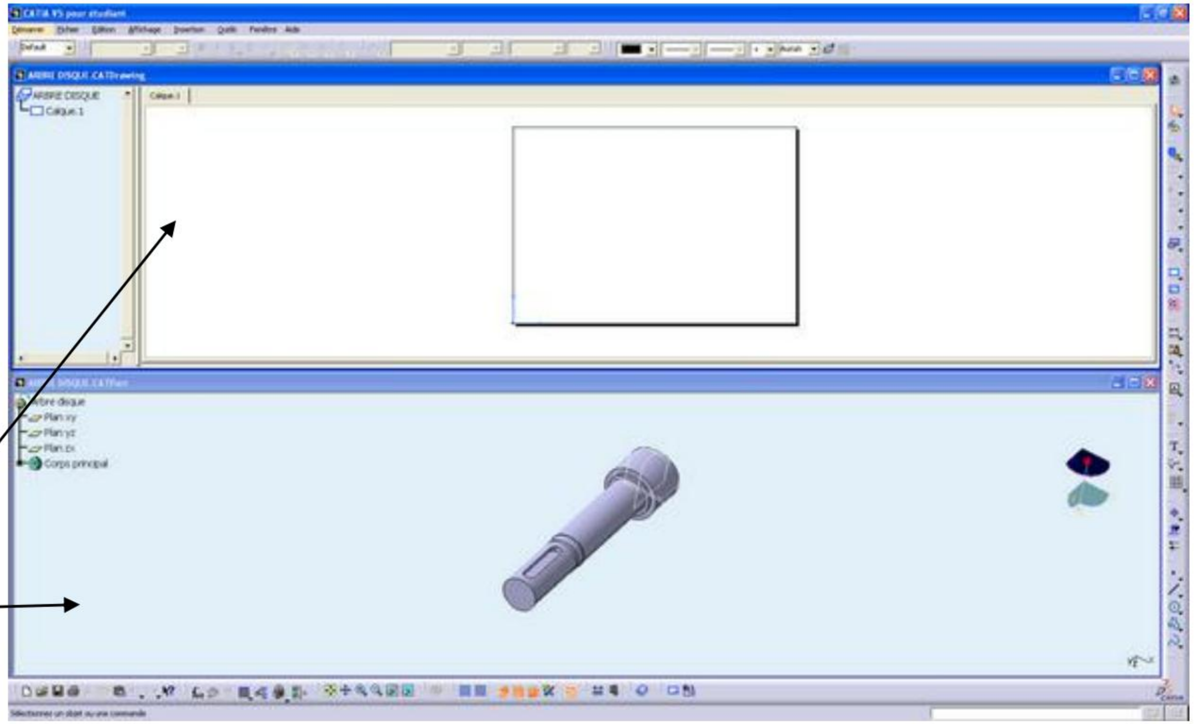


File menu **> Open > L: catia** >

 DISK TREE.CATPart > Open

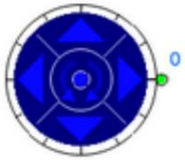
Window menu **> Horizontal Tile**

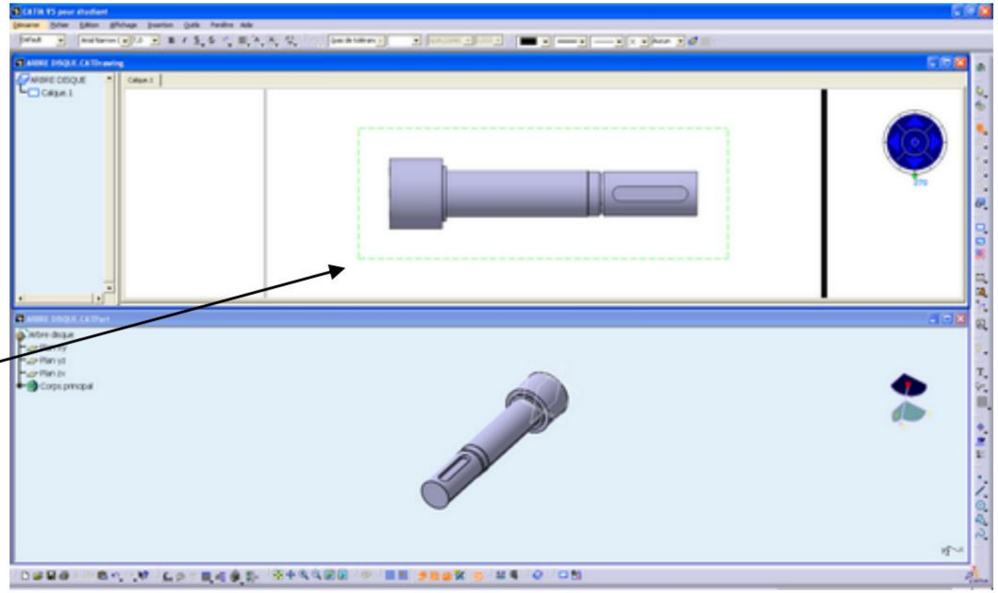
Two windows are then visible on the screen: - the
drawing or ***2D*** window - the ***part*** or ***3D*** window



In the Drawing window and **FRONT VIEW**

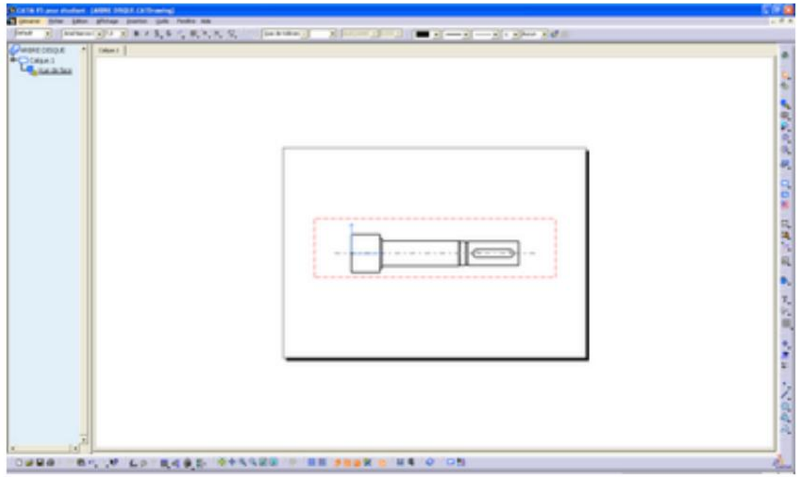
XY plane in the 3D window: the selected view is displayed in the 2D, Using this small element, Rotate the part of the 2D window so as to obtain the orientation of the figure (keyway visible) .





(Hold) the view frame and move the part in the layer to the desired location.

next to the view to confirm the position.



Now, for better visibility, minimize the **3D** window and enlarge the **2D** window so that it takes up the whole screen.

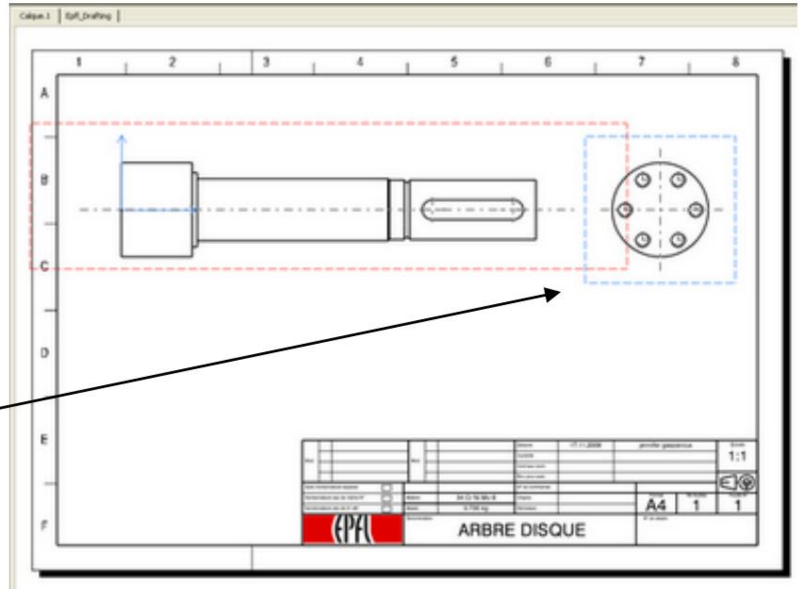
Note : To change the scale of the drawing, > on the red frame of the drawing **Properties** > **View** tab > **Scale >** enter the desired scale > **OK.** (in this case, you have to work on a 1:1 scale, which must already be the case)

To insert the cartridge:

CARTRIDGE CREATION > The **A4** cartridge appears with the name **DISC SHAFT.**

To modify the title block, **EPFL_Drafting** tab , then **twice** on the box of the title block to be modified. To come back, Layer tab.1 .

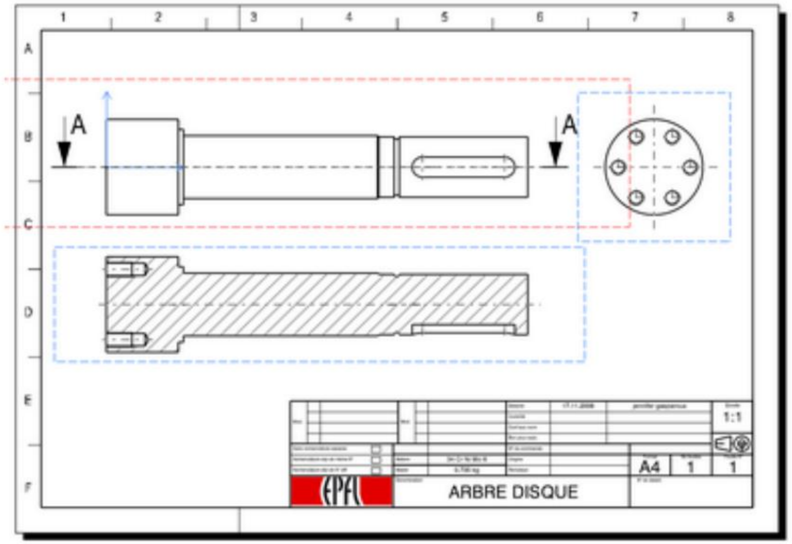
PROJECTED VIEW (behind **FRONT VIEW)** > place the view to the right of the room.



BROKEN CUT

On the horizontal axis of the front view (on the left of the part) > **2x** on the axis (on the right of the part) **>** lay the cut

Hide the letters **A** indicating the cut: on the elements in question > **hide/show** (or on the element > **space bar)**

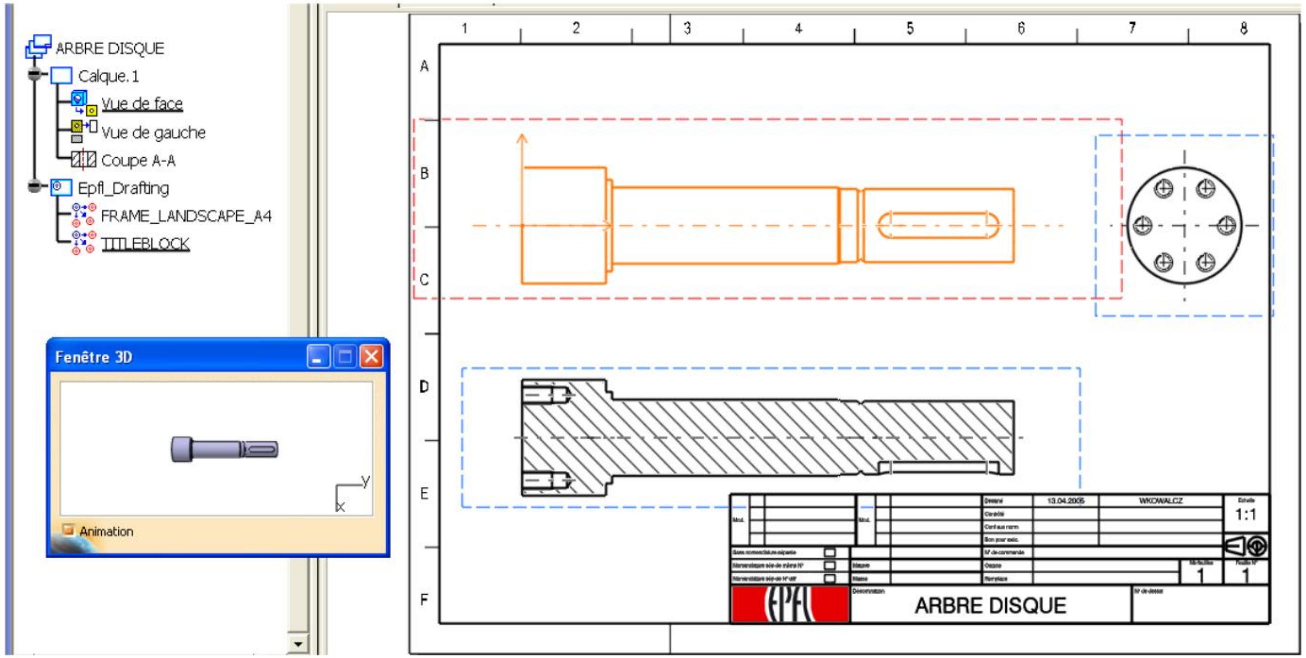


Note: to retrieve hidden items:
View menu > Hide/Show > **Show Hidden/Shown Objects**
Then on the object and hide/show.
To return to the displayed objects: **View** menu > hide/show > **Show hidden/displayed objects**
(Where on the icon located at the bottom of the screen)

Checking links: **File** menu > **Desktop** (the two linked documents are side by side)

close office

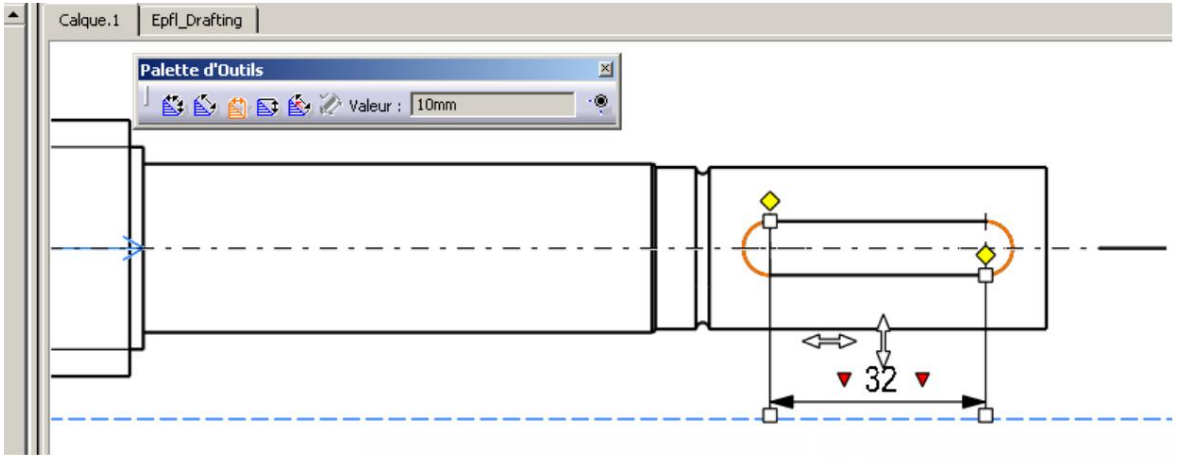
Tools menu > **Analyze** > **Show Geometry in All Viewpoints**



Drag the cursor over the different views of the **2D Window** and observe the highlighting on the **3D Window.**
Close the 3D window.

IMPORTANT : **before dimensioning the figure, select the frame of the view you want to dimension > the working frame turns red.**

FRONT VIEW DIMENSION

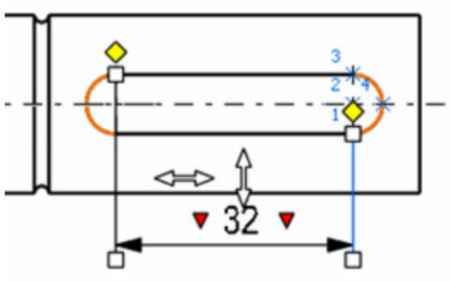


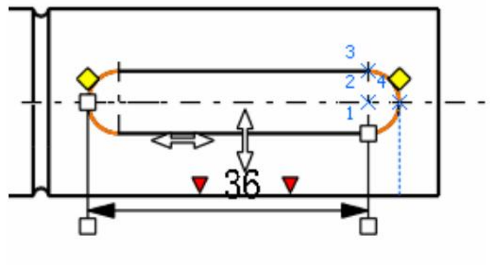
Keyway

  **QUOTATIONS**

 left arch

 right arc



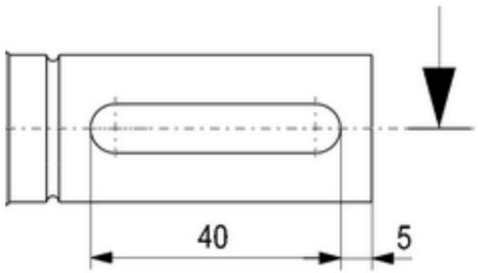


Hold down the **Ctrl** button and point the mouse at one of the yellow diamonds. (The positions **1 2 3 4** indicate the anchoring places of the dimension line).

With **Ctrl** still pressed, you will navigate to #4 (outside the groove)  (and hold) one of the yellow diamonds that

Turn let go and do the same with the other diamond.

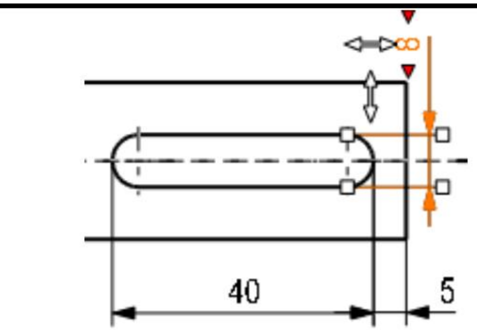
 next to the view to confirm the position of the dimension.



Proceed in the same way to dimension the distance (5) from the key to the end.

Machine Translated by Google

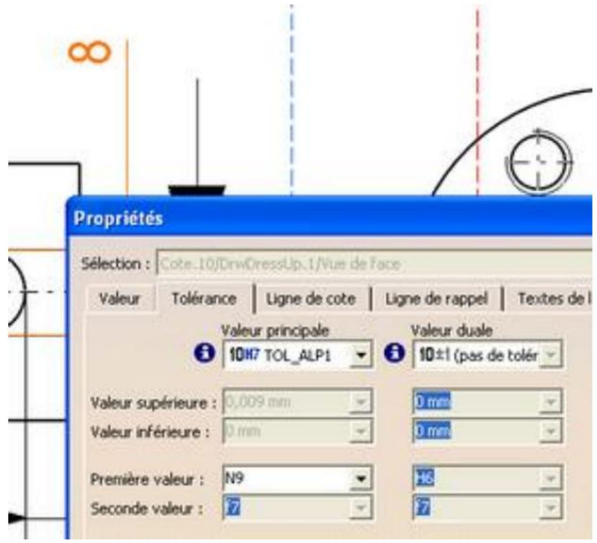
QUOTATIONS



The two parallel lines **> Set** the dimension

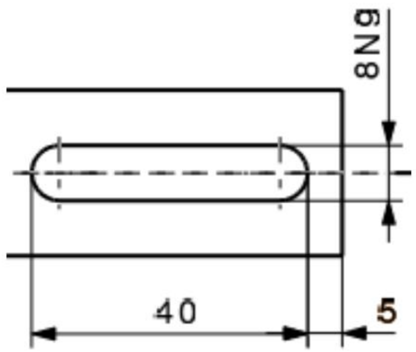
next to the view to confirm the position of the dimension.

Note : when defining the dimension, to prevent the writing from following the mouse cursor, press **Ctrl** when selecting the surfaces associated with the dimension.



on the previous dimension (8) > **Properties** > **Tolerance** tab
> Main value: 10H7 TOL_ALP1
> **First value :** N9 (enter this value manually) > OK

next to the view to confirm the position of the dimension.



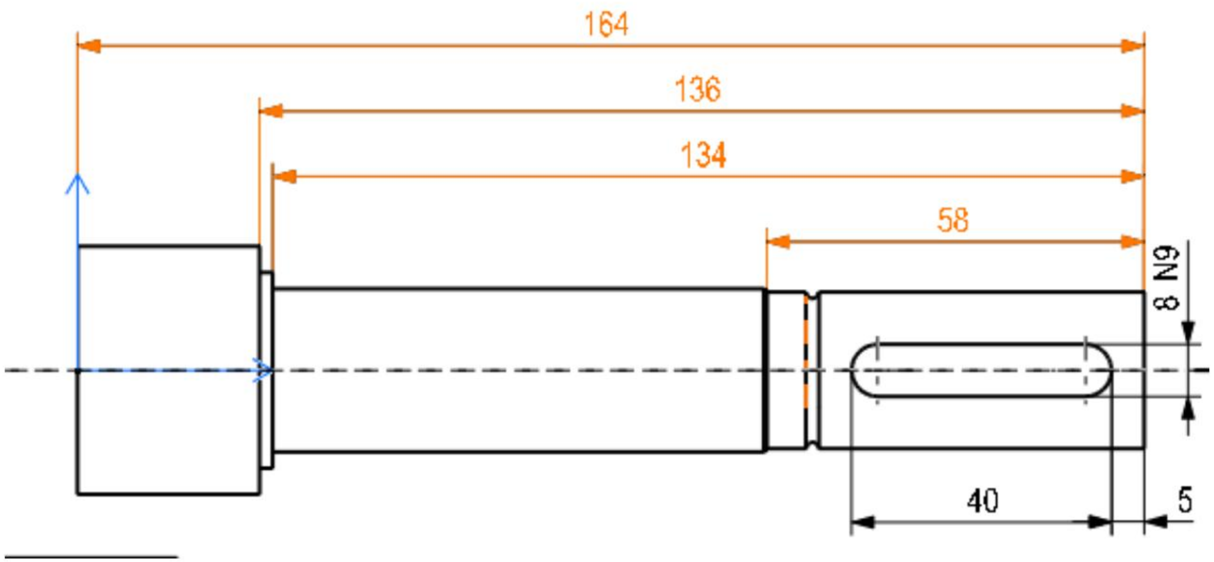
Execution of horizontal dimensions 58, 134, 136 and 164

STACKED QUOTATIONS (behind **quotations)**

the shaft end line (which will serve as a reference) >

the first edge to be dimensioned (58mm) > the second edge to be dimensioned (134mm) > the third edge to be dimensioned (136mm) > the last edge to be dimensioned (164mm)

Set dimensions > next to the view to confirm the position of the dimension.



Note : Subsequently, it is advisable not to use the **Stacked dimensions option.** This option does not allow you to modify or delete a single dimension, you must delete all the stacked dimensions and redo them.

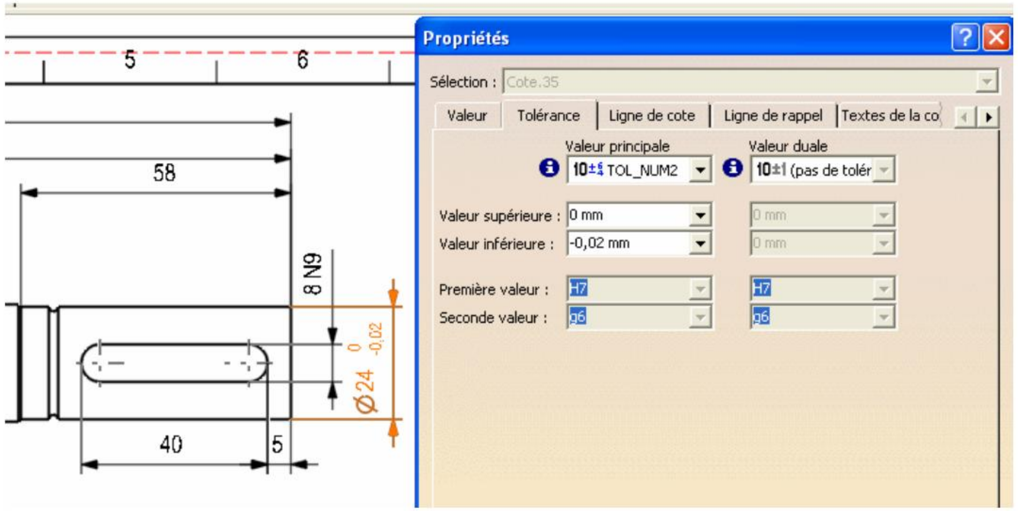
Execution of vertical diameter dimensions Ø24 0/-0.02, Ø25, Ø30, Ø38 0/-0.02

  **DIAMETER QUOTATIONS** (behind **quotations)**

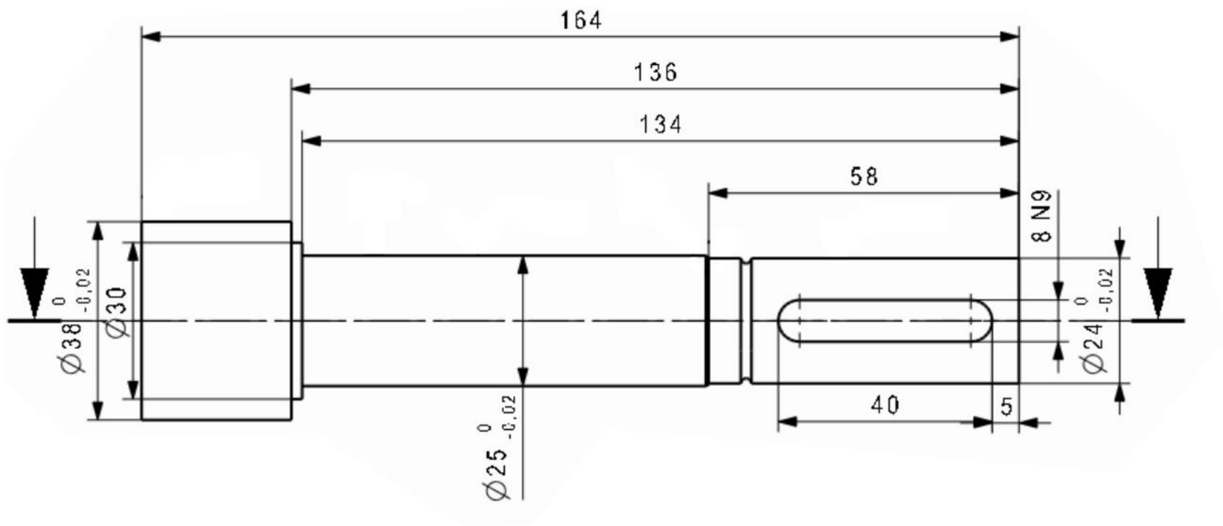
 the Ø24 diameter edges > set the dimension >

 on dimension > **Properties**

 **Tolerance** tab > **Principal Value:** select 10-4 +6 TOL_NUM2 > **Upper value :** enter **0** > **Lower value :** enter **- 0.02** > OK

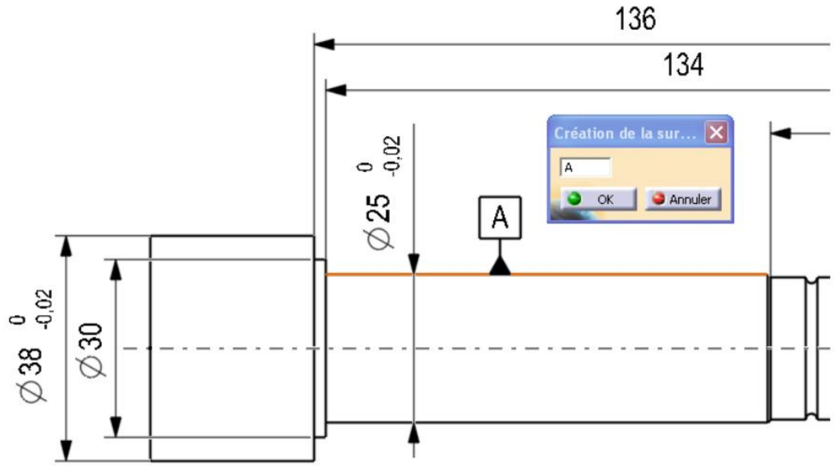


In the same way, carry out the other dimensions of diameter Ø25 **0/-0.02;** Ø30; Ø38 **0/-0.02.**



REFERENCE

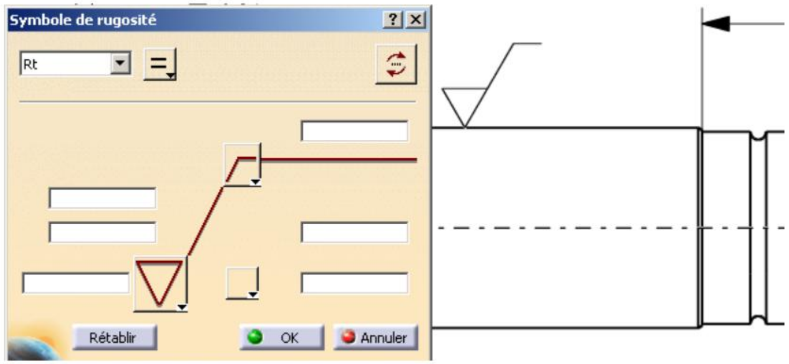
the upper line of diameter Ø25 > set the reference > enter **A** > okay



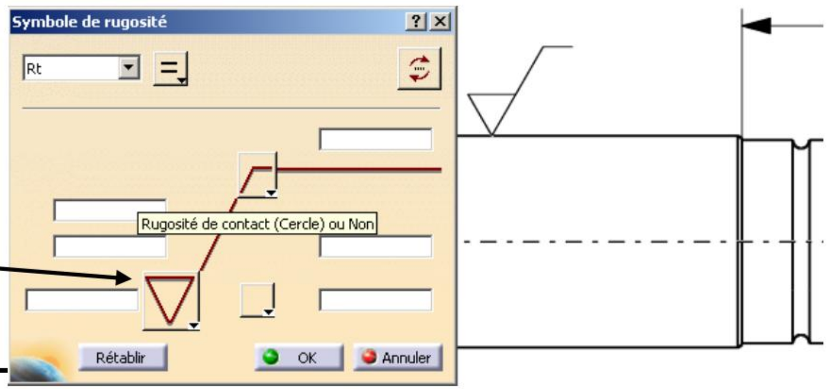
Execution of roughness symbols.

ROUGHNESS SYMBOL

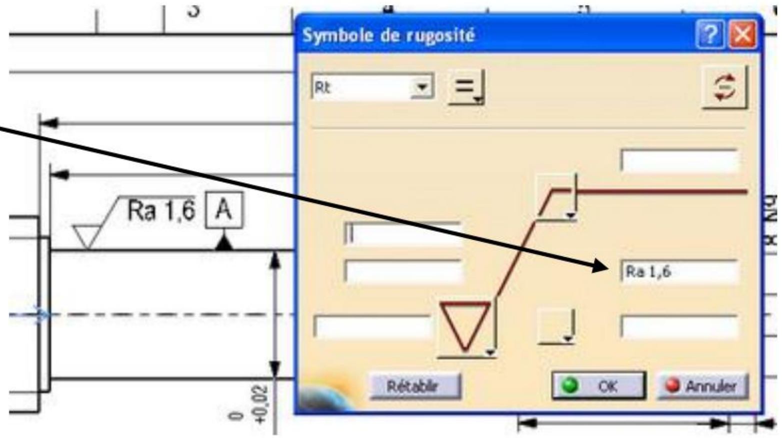
the upper line of the diameter Ø25 (positioning of the anchor)



Contact Roughness (Circle) or No > symbol without circle

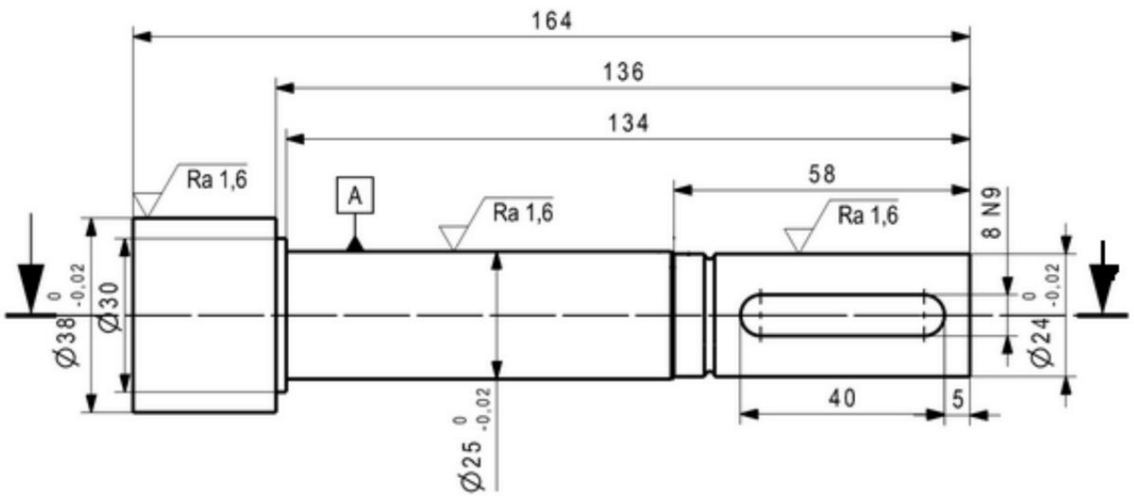


Number or text > enter **Ra 1.6**



 okay

Execute the other symbols in the same way.



CUP QUOTATION

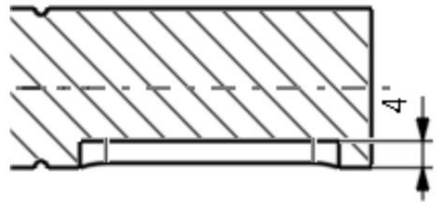
Activate **section view** (2x on the Cup frame: the frame turns red)

Keyway depth

QUOTATIONS

the bottom of the keyway > the outer edge of the shaft > Set the dimension

next to the view to confirm the position of the dimension.



Dimensions 51, R1, Ø22 of the throat.

QUOTATIONS

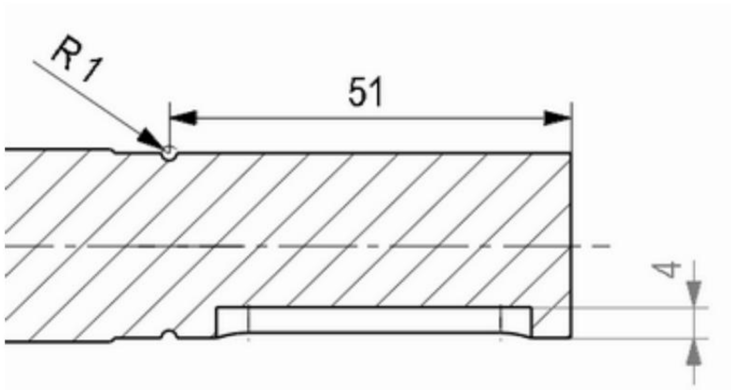
the shaft end line > the back of the groove > Set the dimension.

next to the view to confirm the position of the dimension.

RADIUS QUOTATIONS

the back of the groove > Set the dimension

next to the view to confirm the position of the dimension.

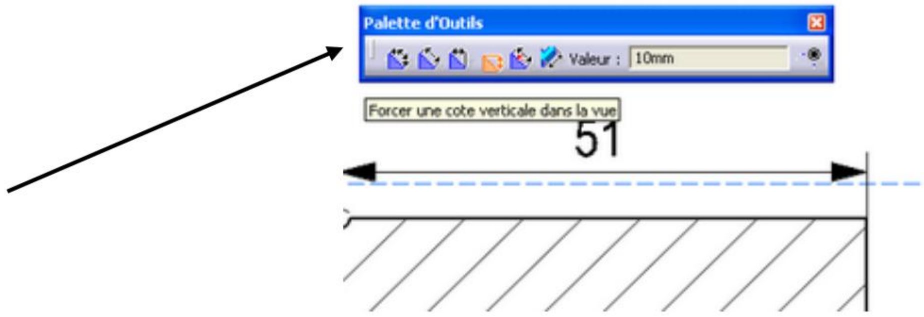


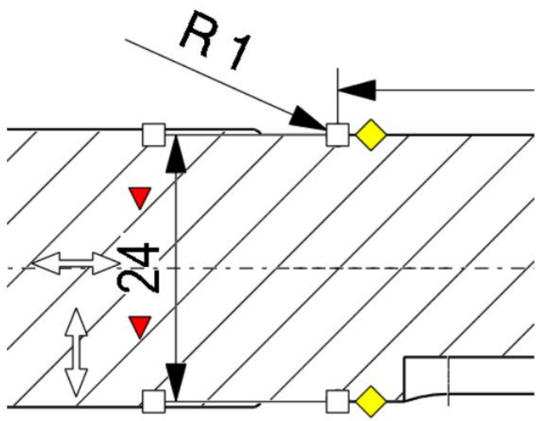
QUOTATIONS

in **Tool Palette** > **Force Vertical Dimension** in View.

back of the throat up.

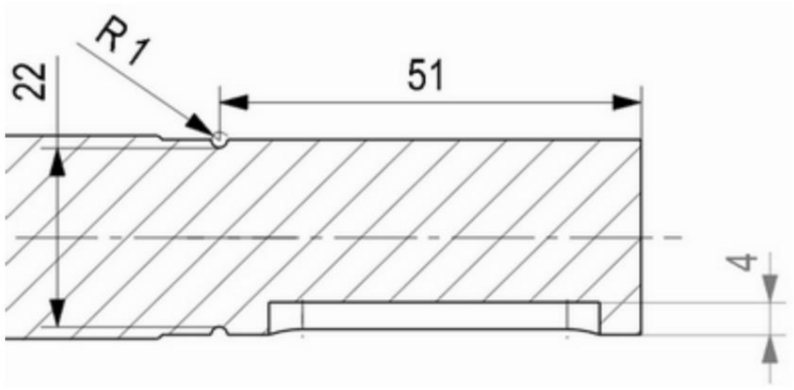
back of the throat down.



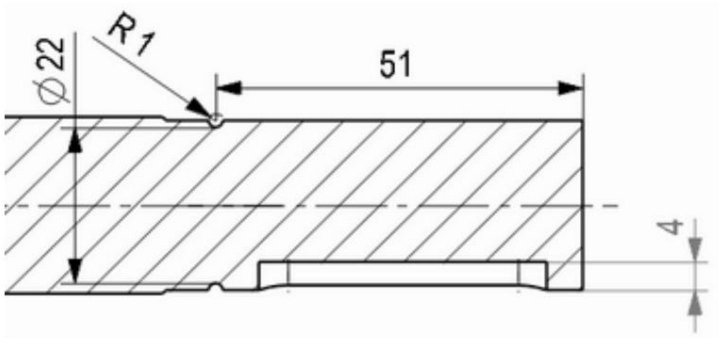


Note : As explained previously: move the yellow diamonds to position 4, in order to obtain the desired dimension **22**

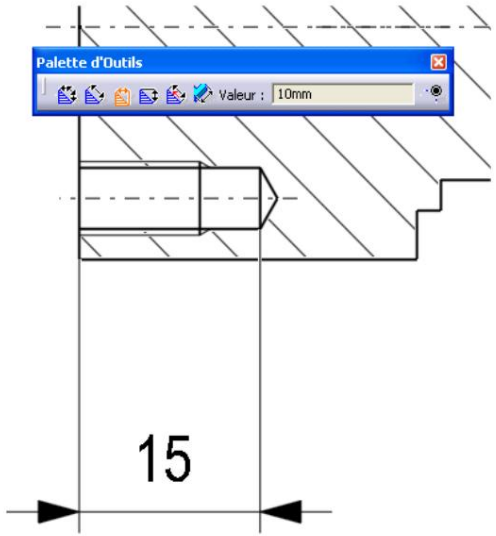
Ask the dimension



dimension 22 > **properties** > tab **Texts of the dimension** > **Prefix :** choose the symbol ÿ



next to the view to confirm the position of the dimension.



Dimensions of the depths of the holes and threads M6

QUOTATIONS

the line of the shaft end and the edge to be dimensioned.

in **Tool Palette** > **Force a horizontal dimension** in the view

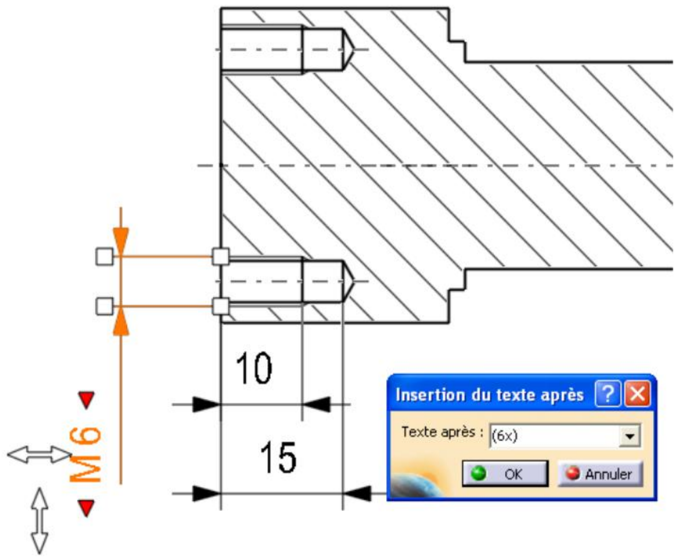
> Set the dimension > next to the view to confirm the position of the dimension.

TAPPING DIMENSIONS (behind **DIMENSIONS)**

a line of the M6 thread.

Dimensions M6 and 10 appear,

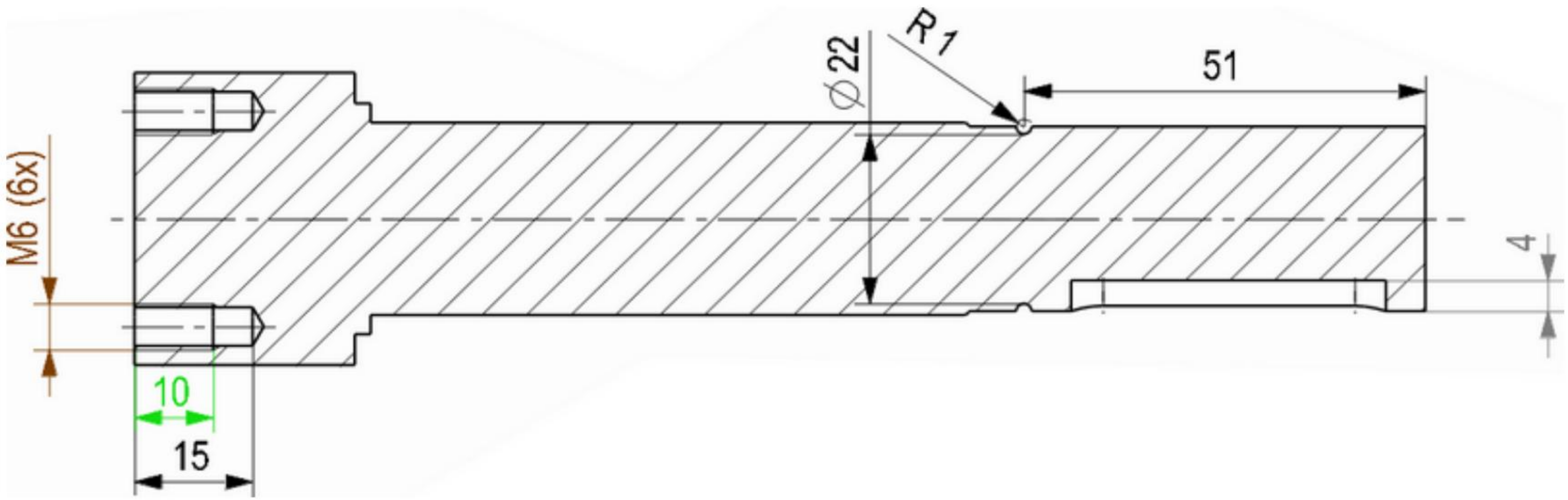
Position the dimensions.



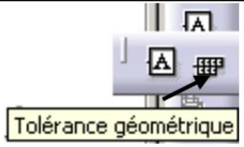
dimension M6.

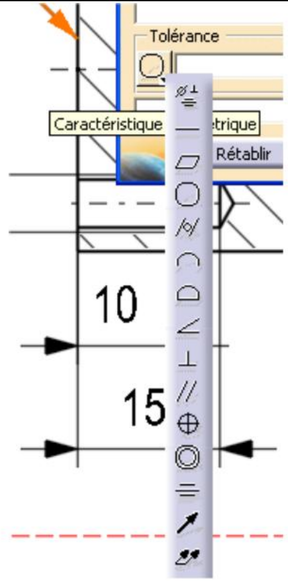
the red triangle after 6 > **Text after** > enter **(6x)** > OK

next to the view to confirm the position of the dimension.



GEOMETRIC TOLERANCES :

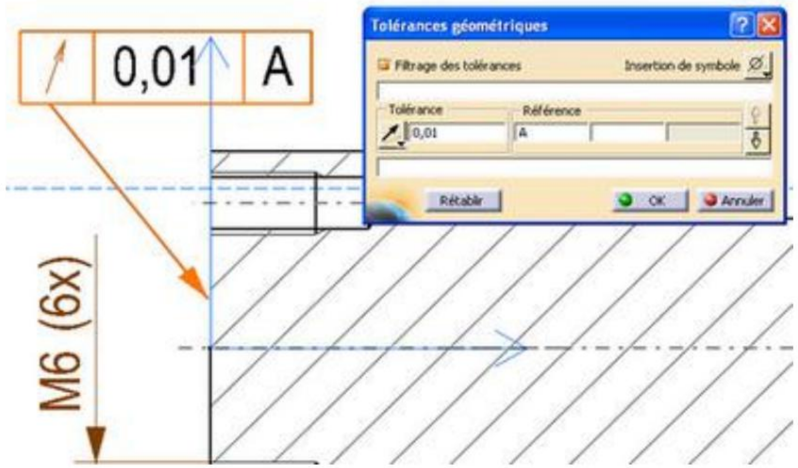




the edge of the shaft end (positioning of the anchor)

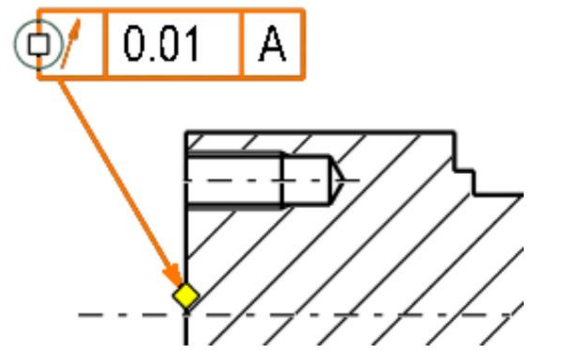
Place the start of the geometric tolerance symbol.

In **Tolerance,** symbol (single beat)



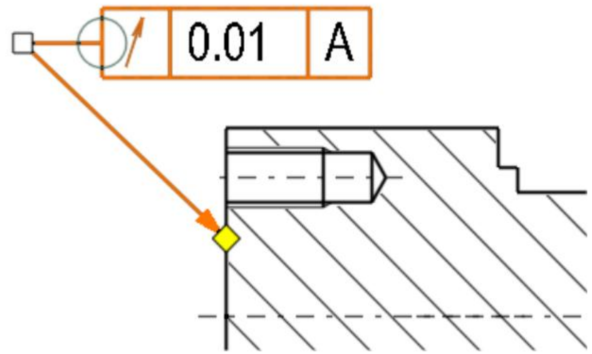
> **Tolerance :** enter **0.01**
> **Reference :** enter A
>OK

Adjust position



Modification of the anchor

In order to modify the type of anchoring of the tolerance, it is necessary to use the yellow diamond , the white circle and/or square .

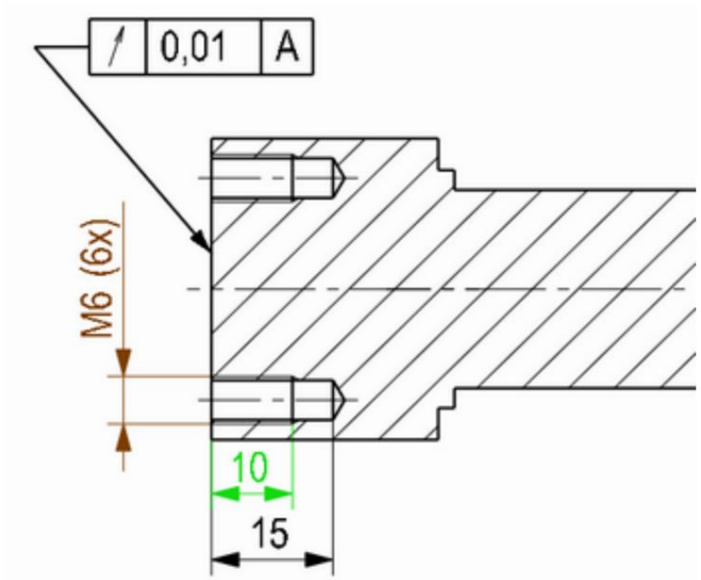


(maintained) the yellow diamond: it is then possible to move the anchor along the surface.

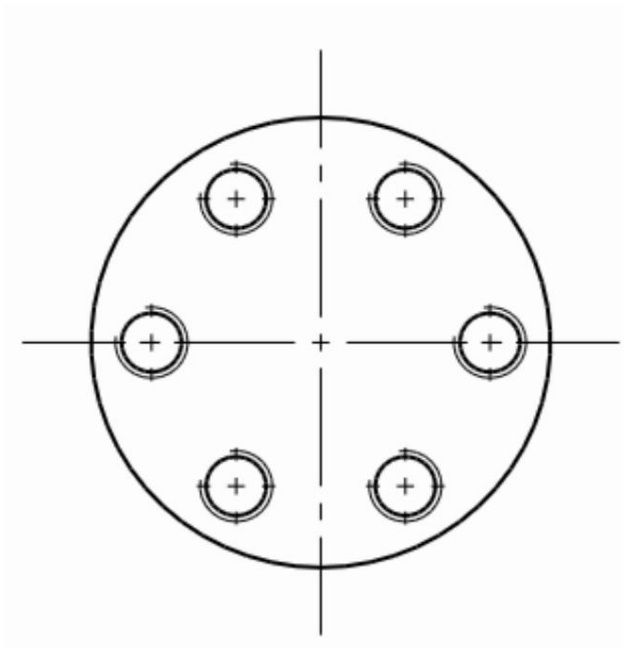
(maintained) the white square: it is then possible to move the square horizontally.

(held) the circle: it is then possible to move the anchor on the box

next to the view to confirm the position.



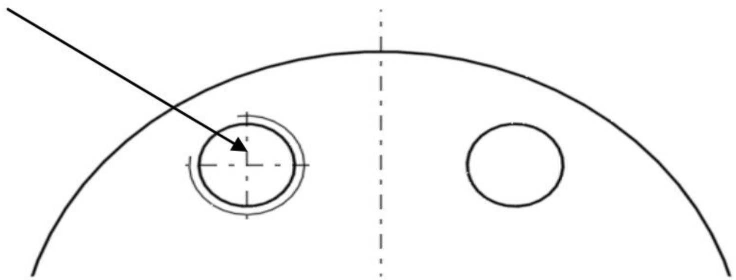
LEFT VIEW DIMENSION



Enable **left view :** 2x  view frame (the frame turns red)

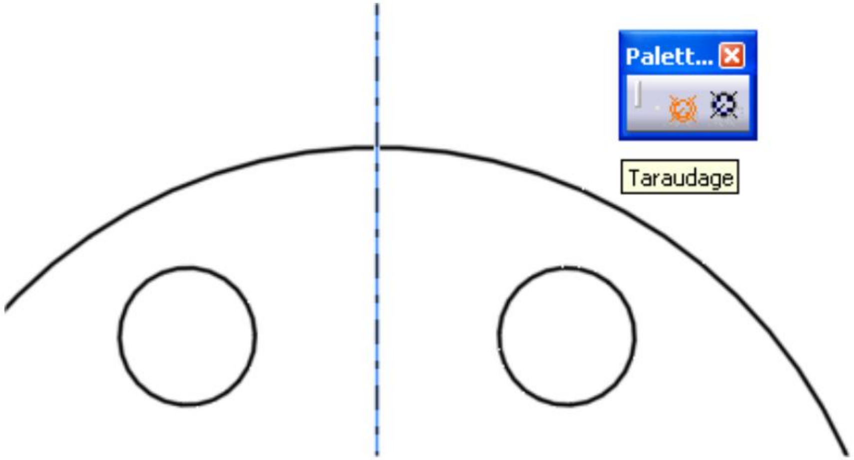
Modification of the center lines of the threads.

Erase all centerlines:  the centerlines to be deleted > press the **Delete** key on the keyboard.

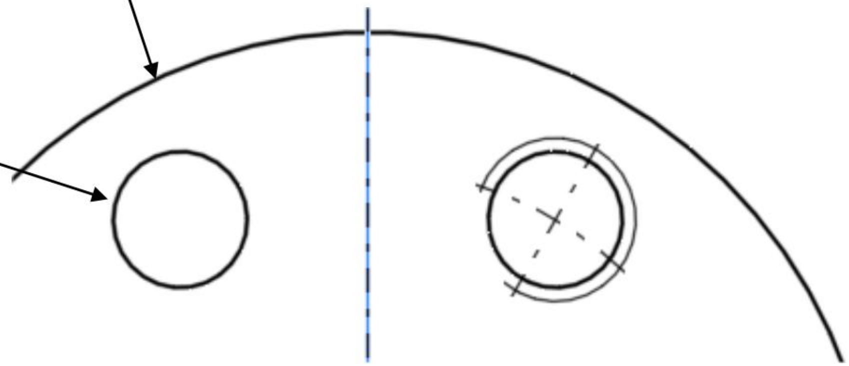


THREAD WITH REFERENCE (behind)

in **Tool Palette** > **Tapping**



tapped hole circle > reference circle (circle outside the part)

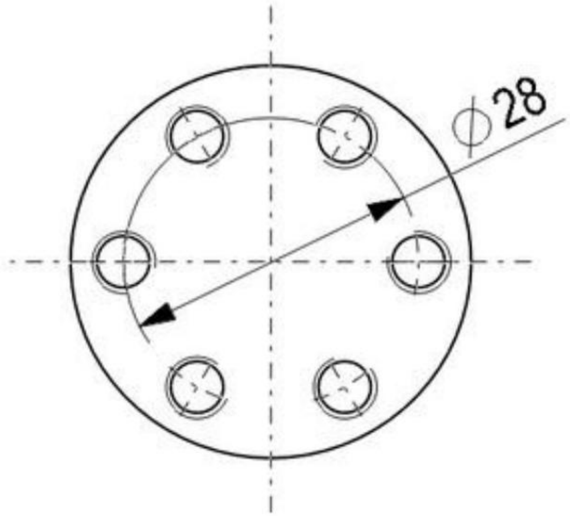


Do the same for all the other holes.

DIAMETER QUOTATIONS

circumference of the centers of the tapped holes > Set the dimension

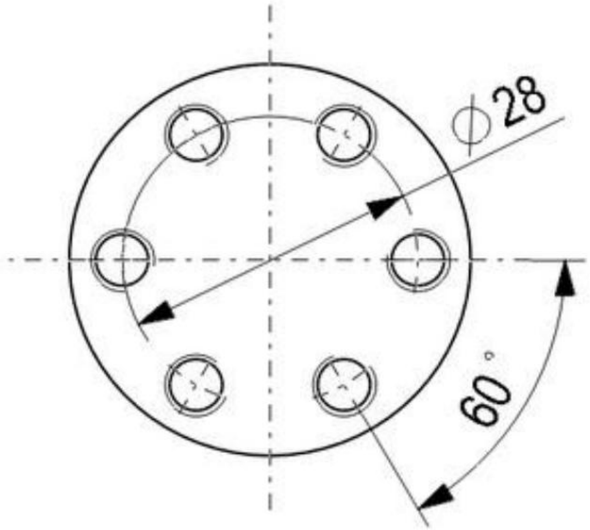
next to the view to confirm the position of the dimension.



ANGLE DIVISIONS

horizontal axis > axis of the tapped hole > set the dimension

next to the view to confirm the position of the dimension.

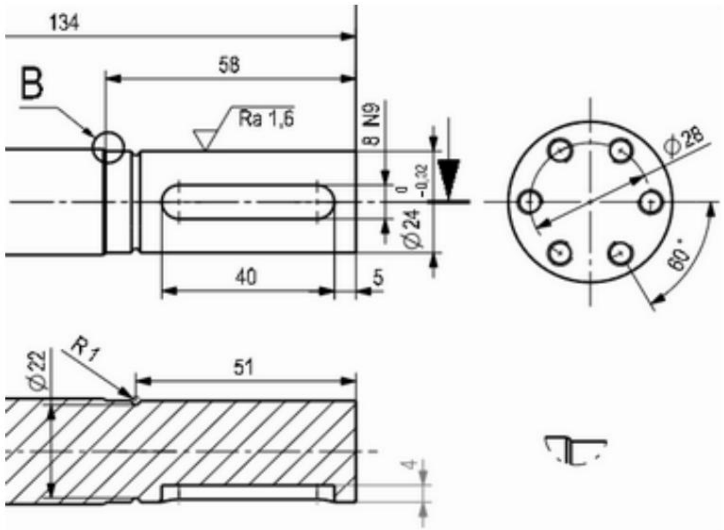


Activate Front **View** (2x  on the frame of the view, which turns red)

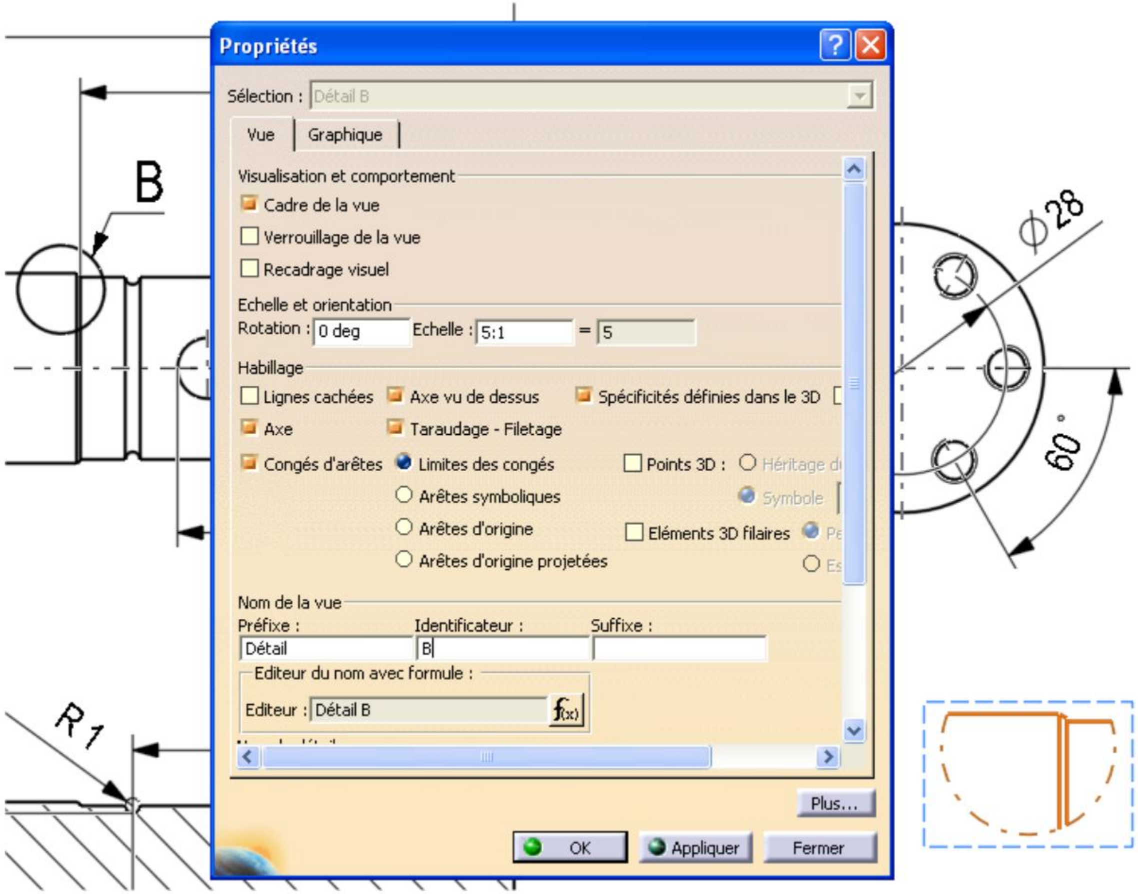
  **DETAIL VIEW**

 to set the center of the detail circle >  to set the size of the circle > set the detail

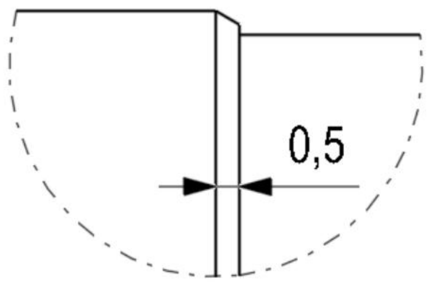
Reposition if necessary the name of the detail (B)



 on detail frame > **Properties** > **Scale ;** enter **5:1 >** OK



Chamfer dimensions in the detail view



Activate **Detail View** (2x on the frame of the view, which turns red)

QUOTATIONS

Dimension the width of the chamfer.

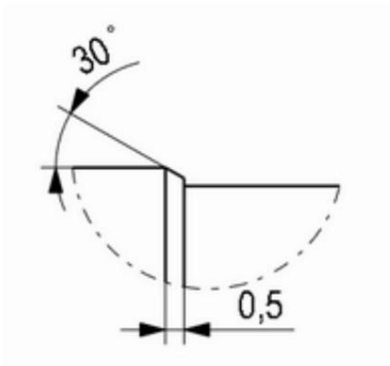
ANGLE DIVISIONS

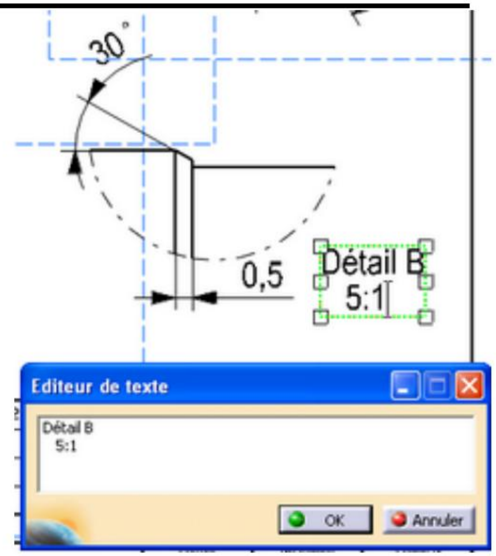
chamfer edge > tree ridge

Angular sector > Sector 3

Ask the dimension

next to the view to confirm the position of the dimension.



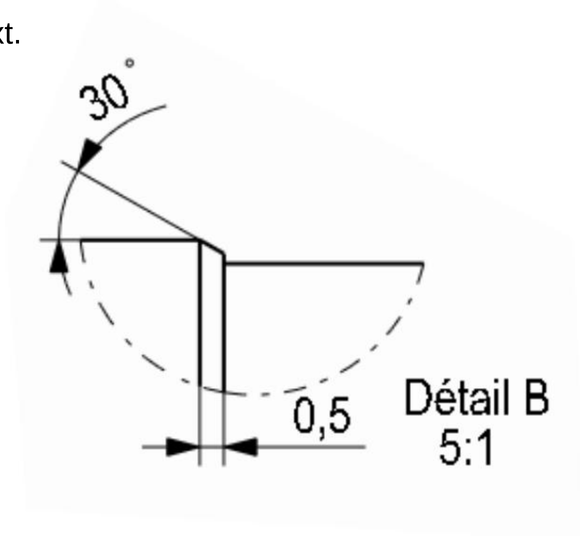


TEXT

text position > Text **Editor** > Enter **DETAIL B, 5:1 >** OK

Adjust text position

next to the view to confirm the position of the text.



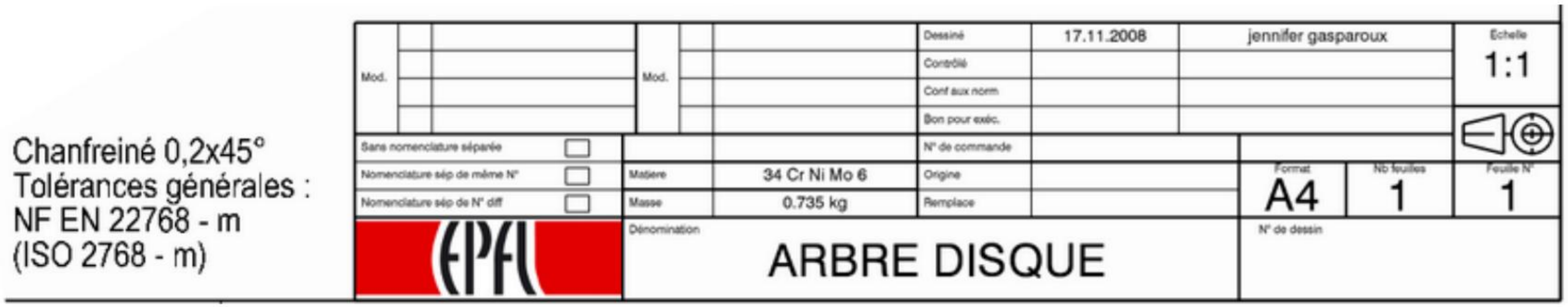
TEXT

text position > Text **editor** > Enter text below **>** OK . (to go to the line press **Shift** + **Enter)**

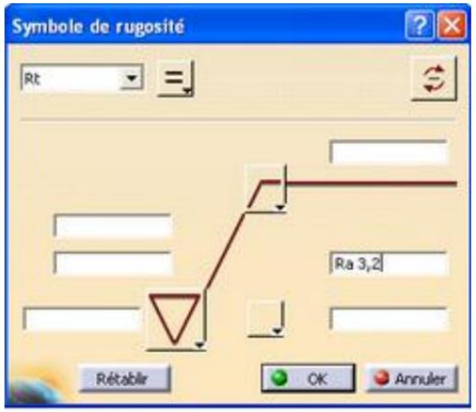
Adjust text position

next to the view to confirm the position of the text.

Text to enter →



Run General Roughness Symbols

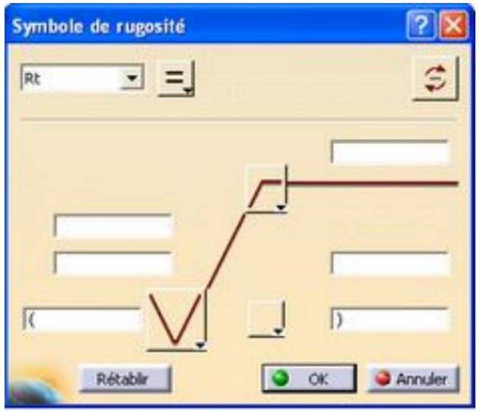


ROUGHNESS SYMBOL

above the cartridge

Number or text: enter Ra 3.2

Okay



ROUGHNESS SYMBOL

above the cartridge (to the right of the previous symbol)

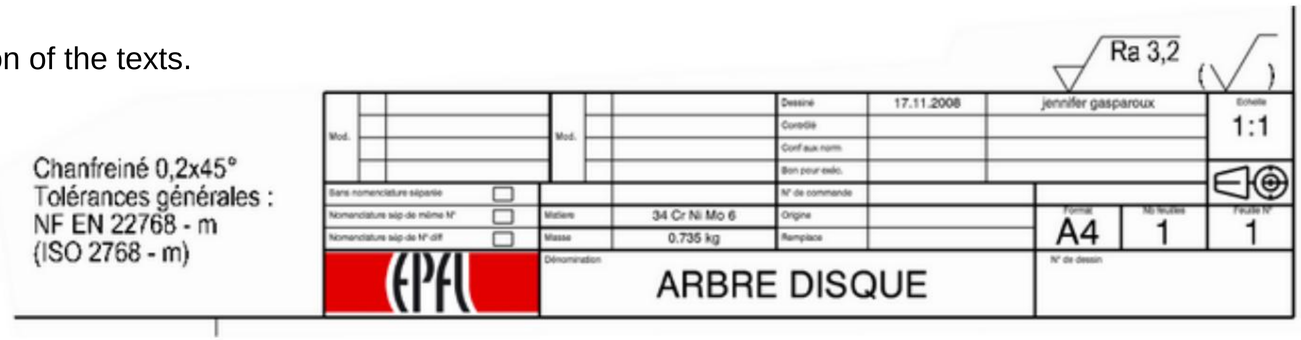
Number or text: enter parentheses

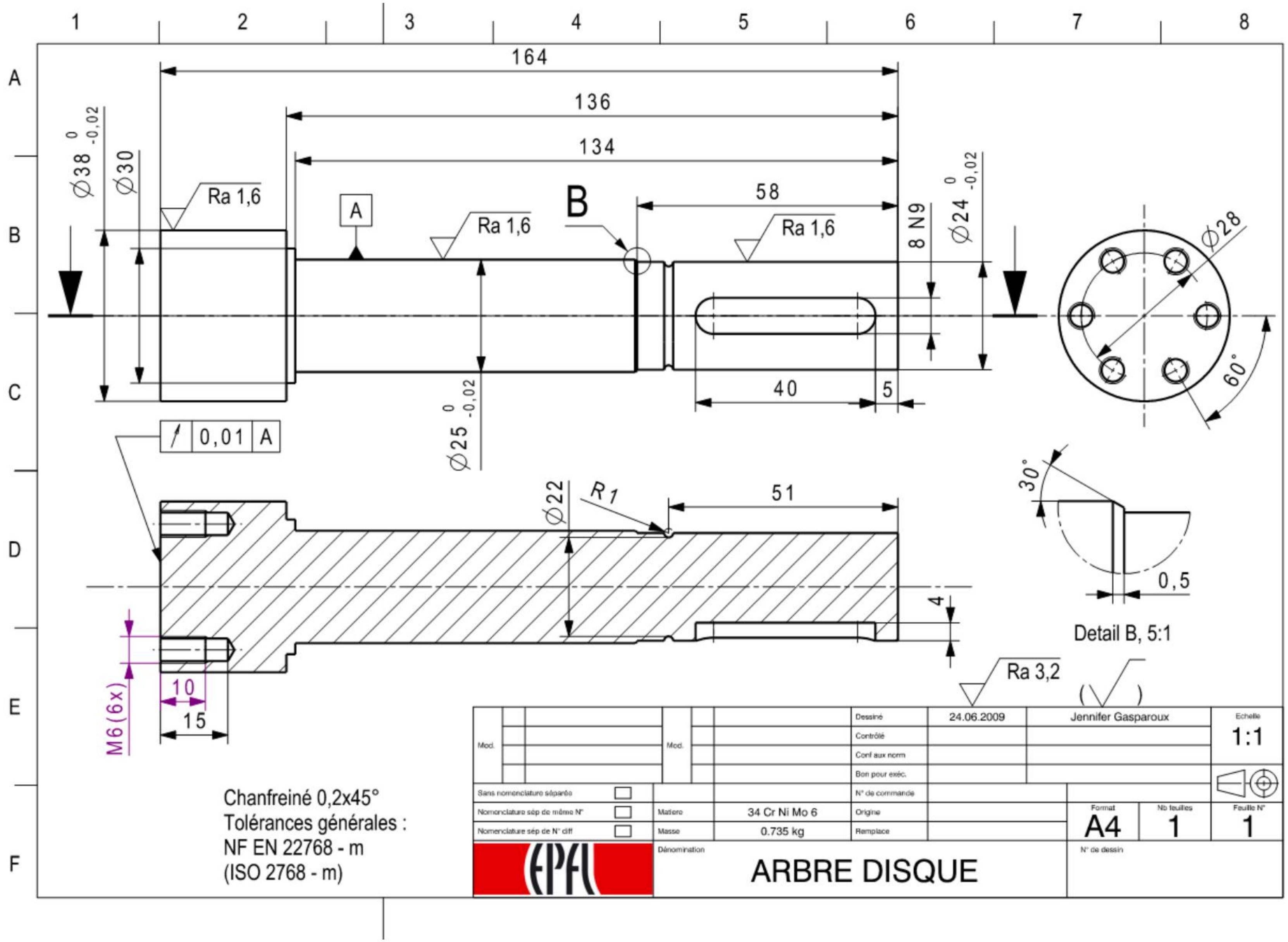
Okay

Adjust positions

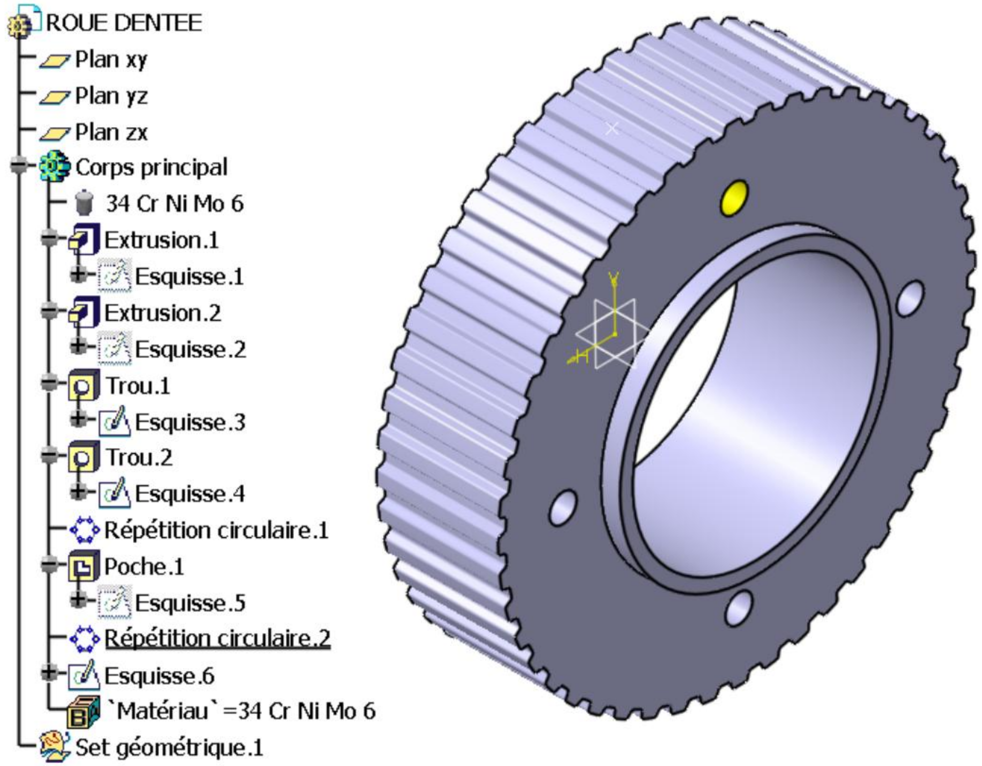
next to the view to confirm the position of the texts.

File > Save




164
136
134
58
40
5
51
B
A
Ra 1,6
Ra 3,2
Ø38 0 -0,02
Ø30
Ø25 0 -0,02
Ø24 0 -0,02
Ø22
Ø28
8 N9
R1
60°
30°
0,5
0,01 A
4
10
15
M6 (6x)
Detail B, 5:1
Chanfreiné 0,2x45°
Tolérances générales :
NF EN 22768 - m
(ISO 2768 - m)
Dessiné
24.06.2009
Jennifer Gasparoux
Echelle
1:1
Contrôlé
Conf aux norm
Bon pour exéc.
N° de commande
Mod.
Sans nomenclature séparée
Nomenclature sép de même N°
Nomenclature sép de N° diff
Matière
34 Cr Ni Mo 6
Masse
0.735 kg
Origine
Remplace
Format
A4
Nb feuilles
1
Feuille N°
1
Dénomination
ARBRE DISQUE
N° de dessin

6. BASIC EXERCISE 3: 3D GEAR WHEEL



FINAL RESULT

Construction process : > *Creation of spans & holes*
> *Creation of the teeth*

***Select =** click once with the left mouse button*

Start menu > **Part Design** > enter the name of the part: **GEAR WHEEL** > OK
File menu > **Save** > L: Catia > file name: **GEAR WHEEL** > Save

SPAN Ø 75.55 mm

the **ZX** plane in the tree structure > **SKETCH**

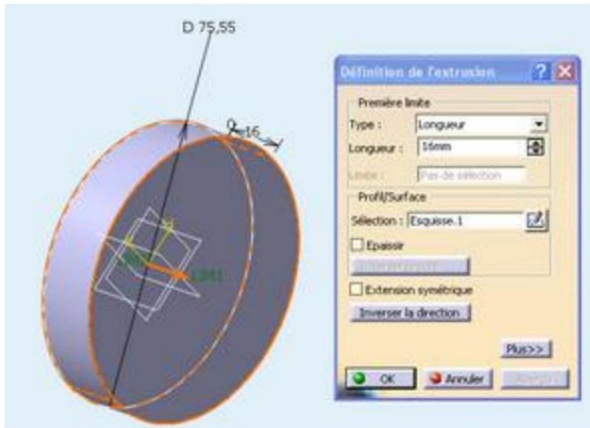
CIRCLE > place the center on the origin = approach the origin with the cursor until the symbol appears (coincidence between two points),

to set the center > circle anywhere in the window to put the of any diameter.

CONSTRAINTS > set dimension > **2x** on the diameter value > enter **75.55 >** OK

LEAVING THE WORKSHOP

EXTRUSION > Type: Length > enter **16** > OK



SPAN Ø 43 mm

the face of the Ø 75.55 span as **the sketch plane**

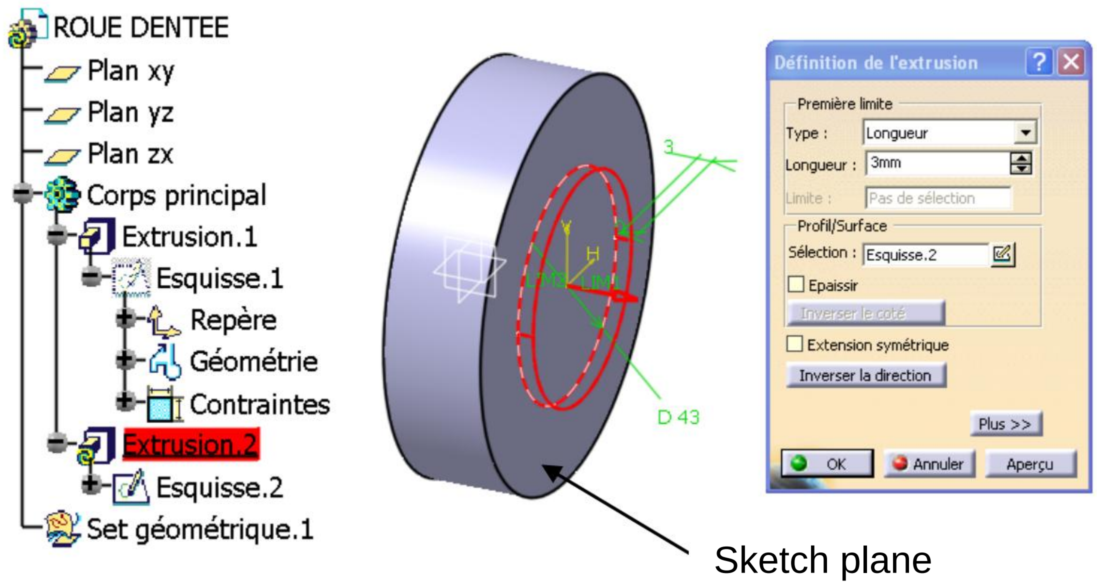
SKETCH

CIRCLE > the center on the origin > anywhere in the window to place the circle of any diameter.

CONSTRAINTS > set the dimension > **2x** on the diameter value > enter **43 >** OK

LEAVING THE WORKSHOP

EXTRUSION > Type: Length > enter **3** > OK

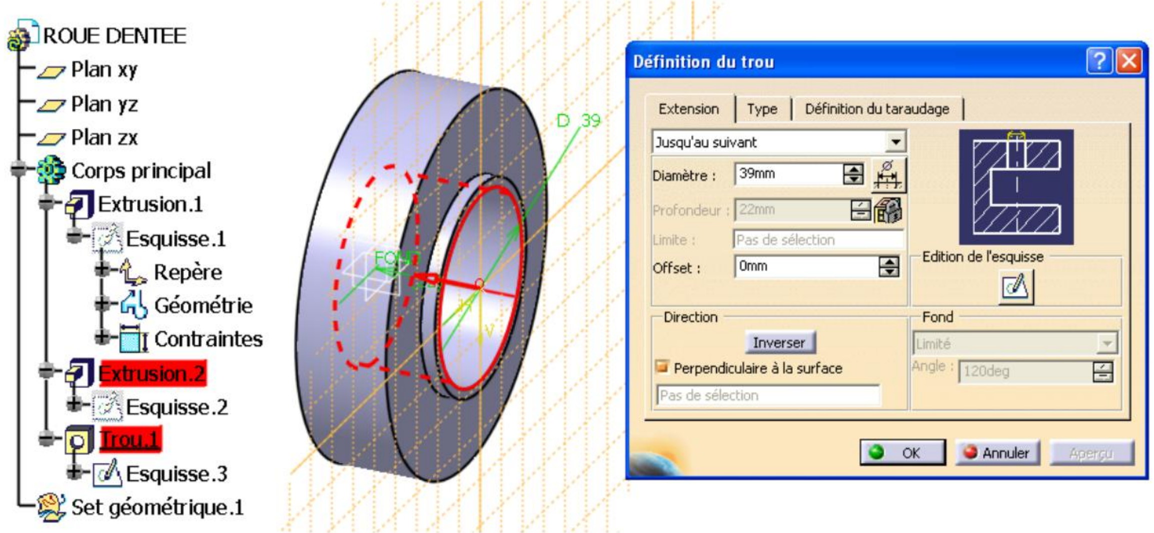


HOLE Ø 39 mm

HOLE

face of the Ø43 staff (the hole appears in red)

Extension tab : Up to next > Diameter : **39** mm > OK



Note : By default, Catia places the hole in the center of the chosen surface; if this is not the desired location, you must dimension the center of the hole by going to **Edit the sketch**

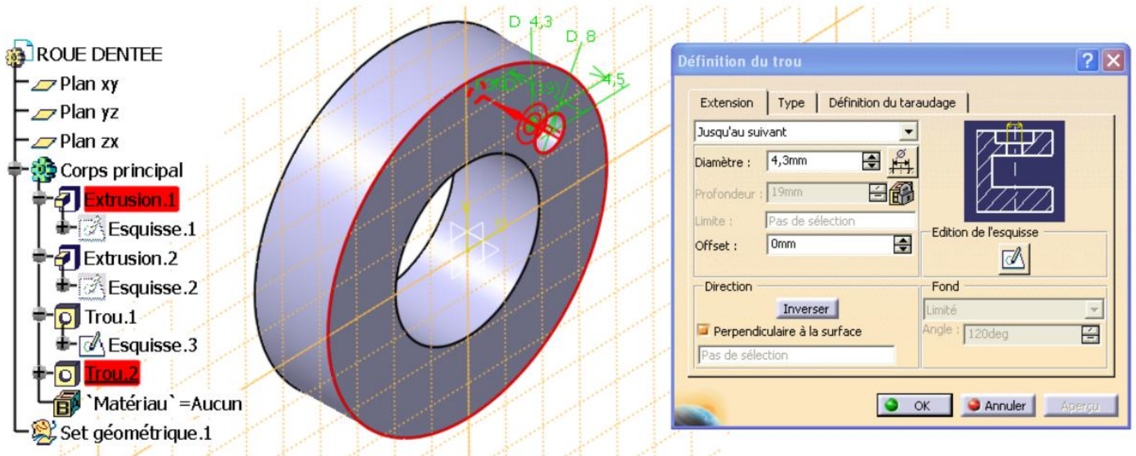
4 HOLES Ø 4.3 mm, BLADES Ø 8 mm prof. 4.5mm

HOLE

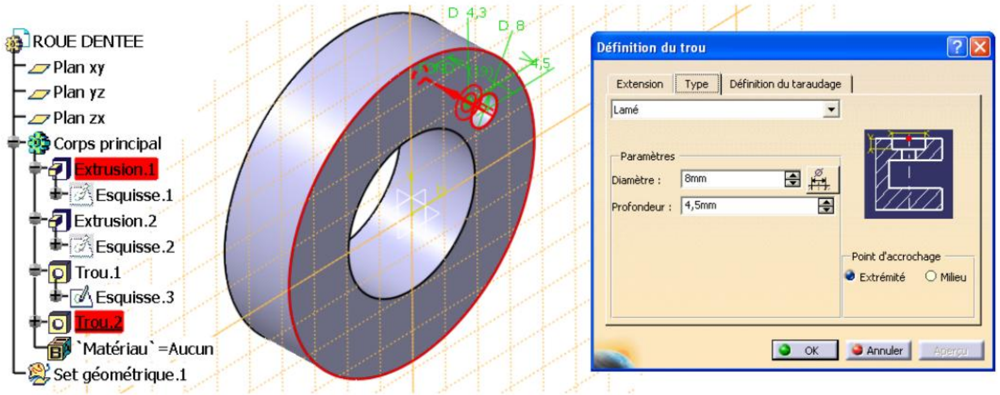
the face of the Ø 75.55 span (opposite the Ø43 span)

Several topological error messages > OK (the hole is empty: no material to remove)

Extension tab : Up to next > Diameter : **4.3** mm



Type tab : Blade > Diameter: **8** mm > Depth : **4.5** mm



Extension tab > **Editing the sketch**

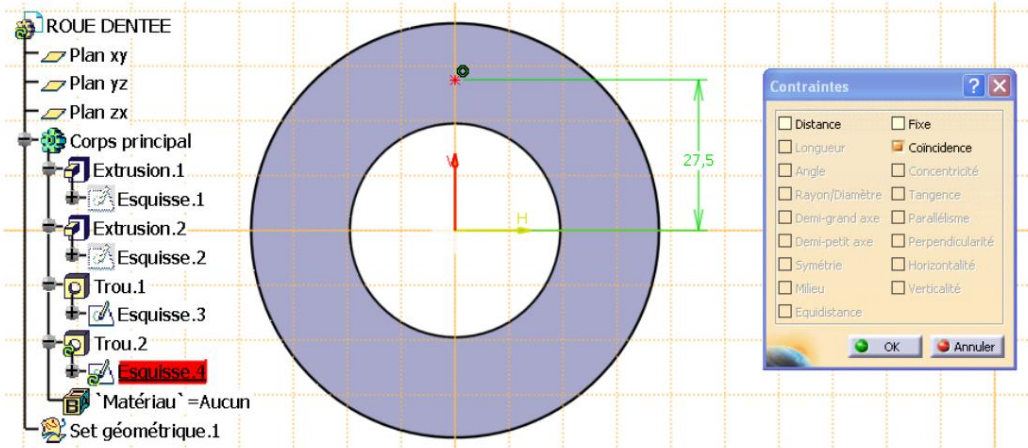
(and hold down) on the center of the hole and move it to facilitate dimensioning

CONSTRAINTS > the horizontal axis **H** and on the point > set the dimension > **2x** on the dimension value > enter **27.5 >** OK

on the vertical axis **V** > hold down the **Ctrl** key and **Ctrl** key on the point (two selected elements turn red) > drop the

SELECTED CONSTRAINTS IN A DIALOG BOX

Coincidence > OK



LEAVING THE WORKSHOP

okay

CIRCULAR REPEAT

> 2x Escape/Esc to no longer have a selection (no more items in red) (or next to the object)

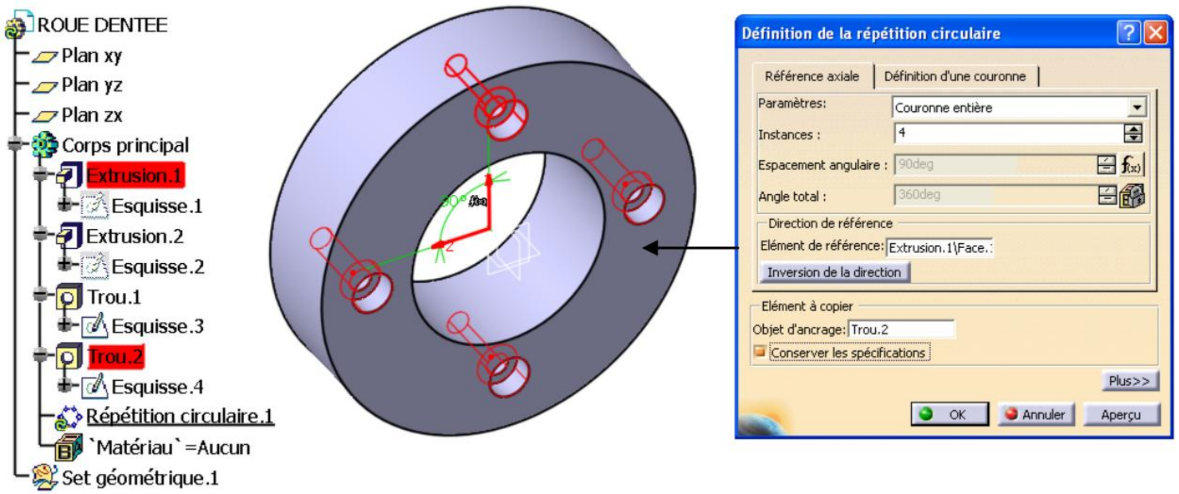
Hole.2 in tree view (hole turns red)

CIRCULAR REPEAT (behind **RECTANGULAR REPEAT)**

Axial reference tab
> **Parameters:** whole crown
> **Instances:** 4

> **Reference element :** face of the Ø 75.55 span (opposite to the Ø43 span)

> Enable **Keep Specifications**
>OK



CREATION OF THE T5 TYPE TEETH WITH 48 TEETH

(Toothed belt Z48T5)

Calculations:

Pitch: pb = 5 mm
Number of teeth: z = 48
Tooth height: ht = 1.2 mm
Distance between 2 teeth (top of the tooth): s = 2.65 mm
Angle between 2 teeth: 2ÿ = 40° Radii int. and ext. of the teeth: rb = rt = 0.4 mm Distance from the top of the tooth to the pitch diameter of the belt: u = 0.42

Diameters:

Pitch diameter of the belt: d = pb * z/ÿ = 5 * 48 / 3.14 ÿ 76.394
Outside diameter of the crown: d0 = d - 2u = 76.394 – 2* 0.42 ÿ 75.554
Inside diameter of the crown: di = d0 - 2 ht = 75.554 – 2*1.2 ÿ 73.154

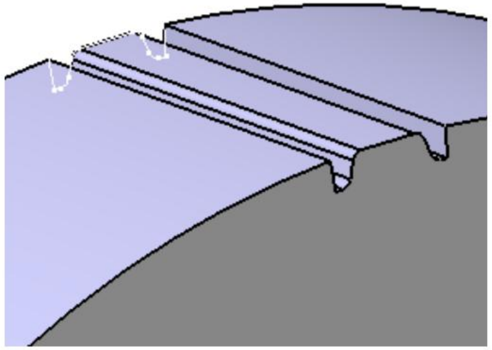
Angles:

Pitch angle: (360° / (ÿ * d)) * 5 = 7.5° or 360° / 48 = 7.5° (= 2*3.75°)
Angle between 2 teeth (top of the tooth): (360° / (ÿ * d)) * 2.65 ÿ 3.975° (ÿ 2*1.9993)

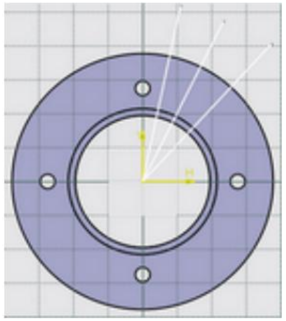
Preliminary remarks :

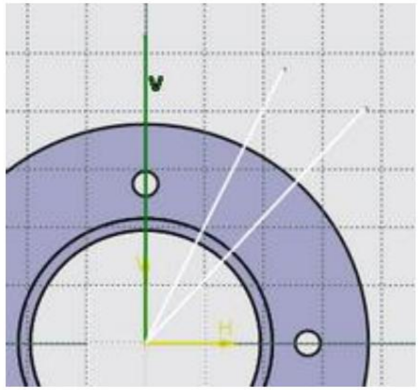
- The bottom of the tooth is an arc of a circle of Ø 73.15mm
- The top of the tooth is an arc of a circle of Ø 75.55mm
- The lines that are not used for the pocket must be construction lines (lines dotted)
- The curve that delimits the pocket does not necessarily have to be closed, but its ends must be outside the part.

Teeth:



Creation of the sketch : the **ZX** plane in the tree structure > **SKETCH**



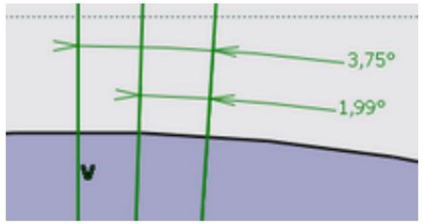


BUILDING FEATURES:

RIGHT > the origin = approach the origin with the cursor until the symbol appears (coincidence between two points)

> outside the room (top right) > **Construct 3 straight lines** in this way

leftmost right > **CONSTRAINTS chosen in a dialog box** > Verticality > OK



CONSTRAINTS > rightmost right and middle right > set dimension

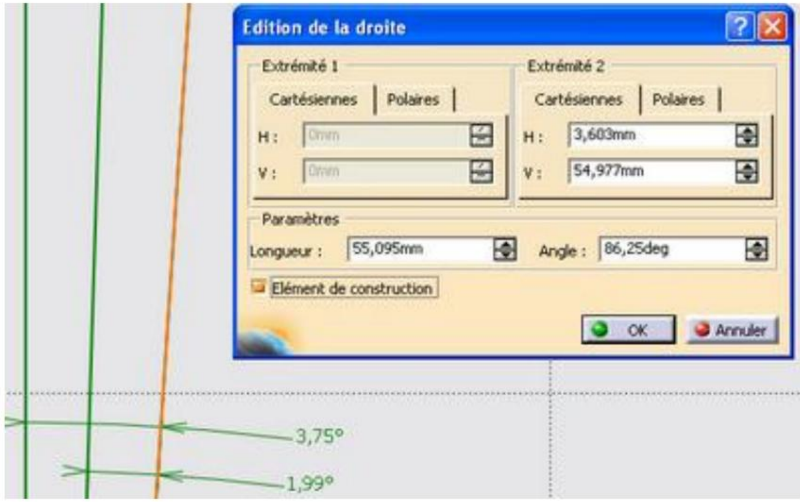
> **2x** angle value > enter **1.99°** > OK

CONSTRAINTS > rightmost right and vertical line > set the dimension

> **2x** angle value > enter **3.75° >** OK

2x a line > activate **Construction element** > OK

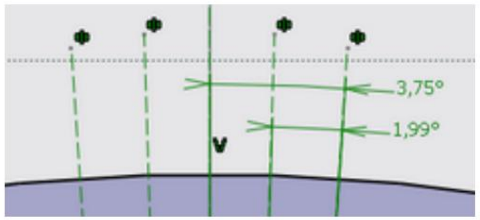
> Do the same with the other 2 straight lines (the lines become dotted)

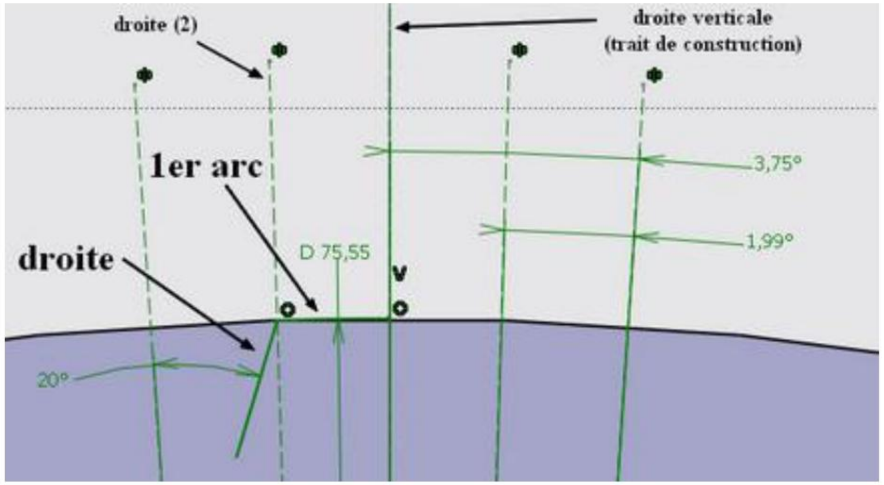


>Hold down the Ctrl key and select the two non-vertical lines

MIRROR > axis of symmetry (= the vertical line)

Note : During the symmetry operation, the previously established constraints will be taken into account.





CONSTRUCTION OF A HALF TOOTH (without roundings):

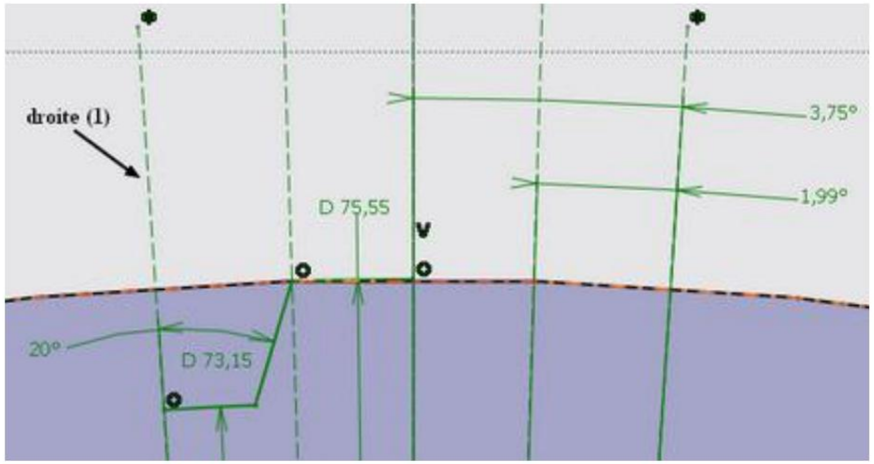
Tip : Don't hesitate to zoom in / zoom out to set the points.

ARC OF CIRCLE > on the origin > right of construction **(2)** and vertical line (cf. image on the right) (the symbol must appear to indicate the coincidence)

CONSTRAINTS > set dimension > > Radius object > Definition > **Dimension :** choose diameter > **Diameter:** enter **75.55** > OK

RIGHT > left end of the arc > bottom left as shown in the figure above.

CONSTRAINTS > on the previous right and on the leftmost construction line > set the dimension > **2x** on the angle value > enter **20° >** OK



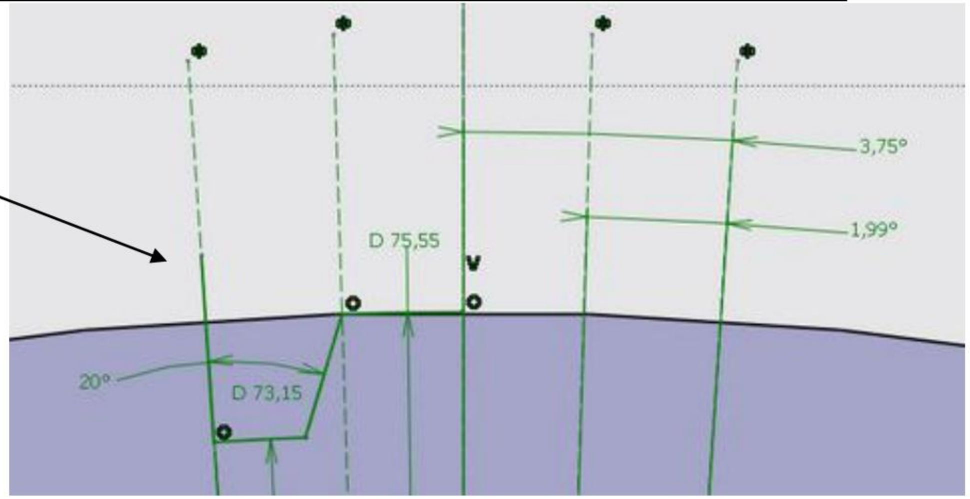
ARC OF CIRCLE > on the origin > left end of the last straight line built and construction line **(1)**

CONSTRAINTS > set the dimension > put a diameter of **73.15** as before **>** OK

RIGHT > end of the last arc built (Ø73.15)

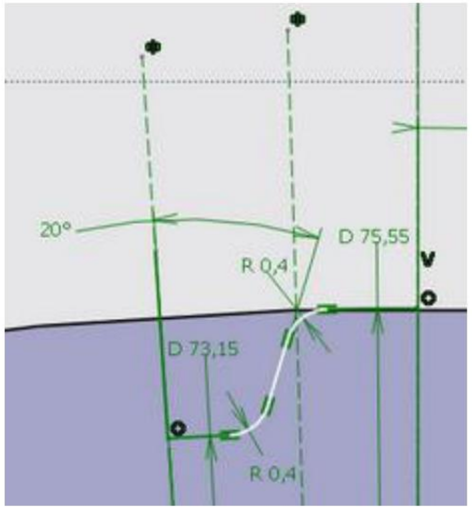
and on the construction line (1), outside the room

next to the object to remove all selections

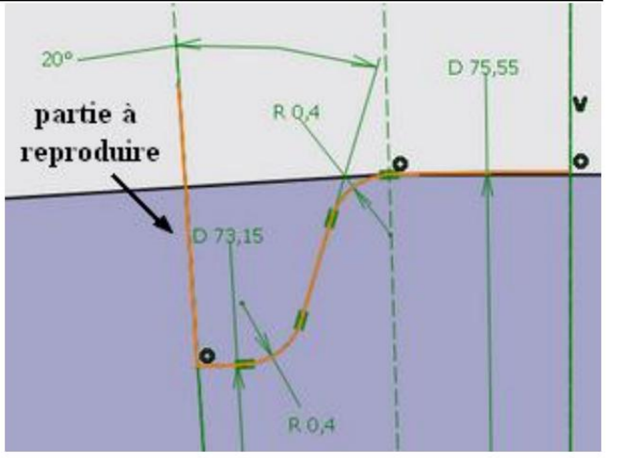


CONSTRUCTION OF ROUNDS:

Create the two roundings of radius 0.4 (see figure) as follows:



ROUND > the two straight lines between which we want the rounding > place the rounding > 2x on the dimension > enter a radius of 0.4 > OK



CONSTRUCTION OF THE OTHER HALF TOOTH:

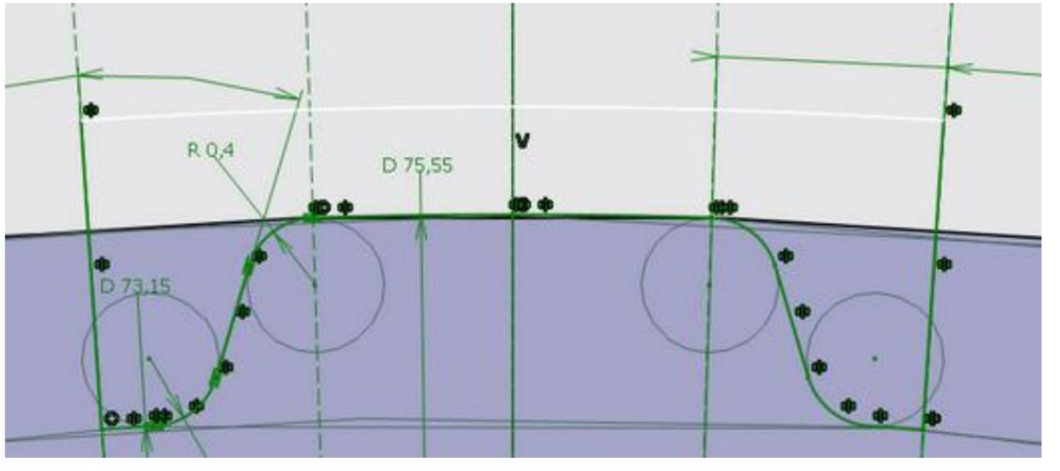
Proceed by symmetry to build the other side:

> Select the part to reproduce while holding down the **Ctrl** key (6 elements)

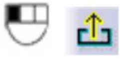
 **MIRROR** >  the vertical construction line (the middle line)

Then close the sketch:

 **ARC OF CIRCLE** >  on the origin >  on the extreme lines to close the contour (see image on the right)



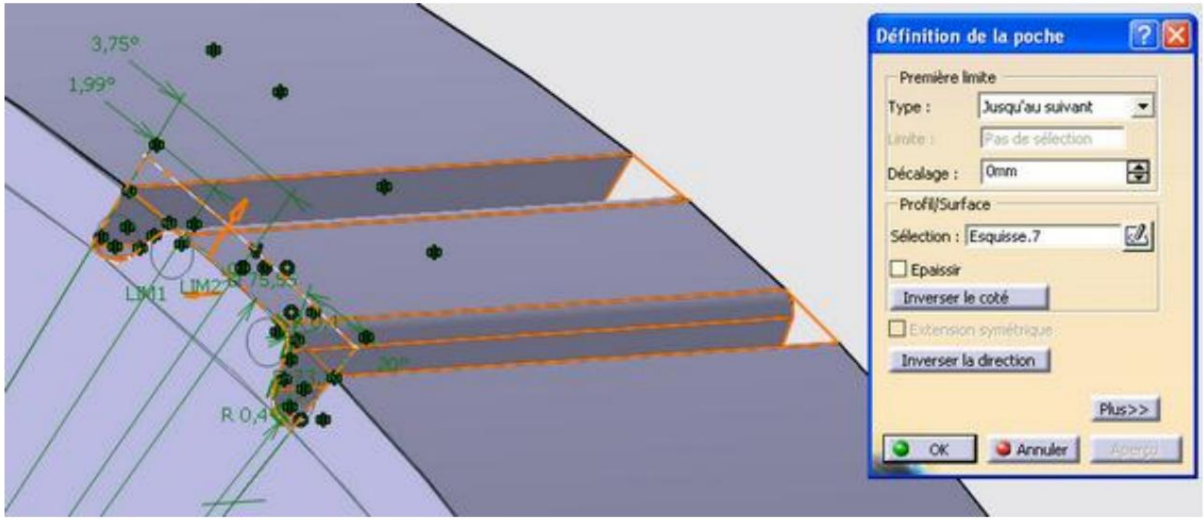
Note: The last arc made is white, which means the sketch is not fully constrained. It is therefore necessary to apply any diameter to this arc, so that it is greater than the diameter of the wheel.

 **LEAVING THE WORKSHOP**

POACHED

> Type : Up to next (reverse direction and side if necessary)

Preview, if the result is satisfactory (figure beside) > OK



CIRCULAR REPEAT :

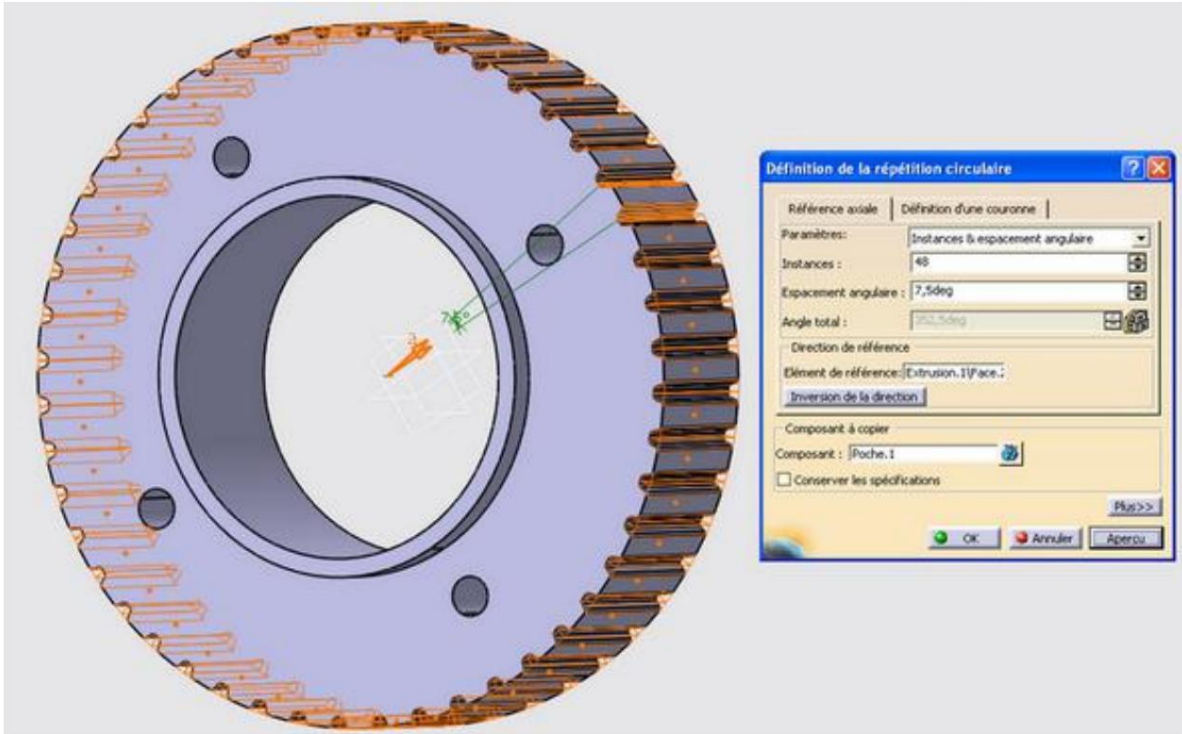
Pocket.1 in tree view (pocket turns red)

CIRCULAR REPEAT

Axial reference tab
> **Parameters:** Whole Crown
> **Instances :** enter **48**

> **Reference direction: Ø 75.55** > Disable: **Keep specifications** the face of the staff

>OK



MATERIAL APPLICATION

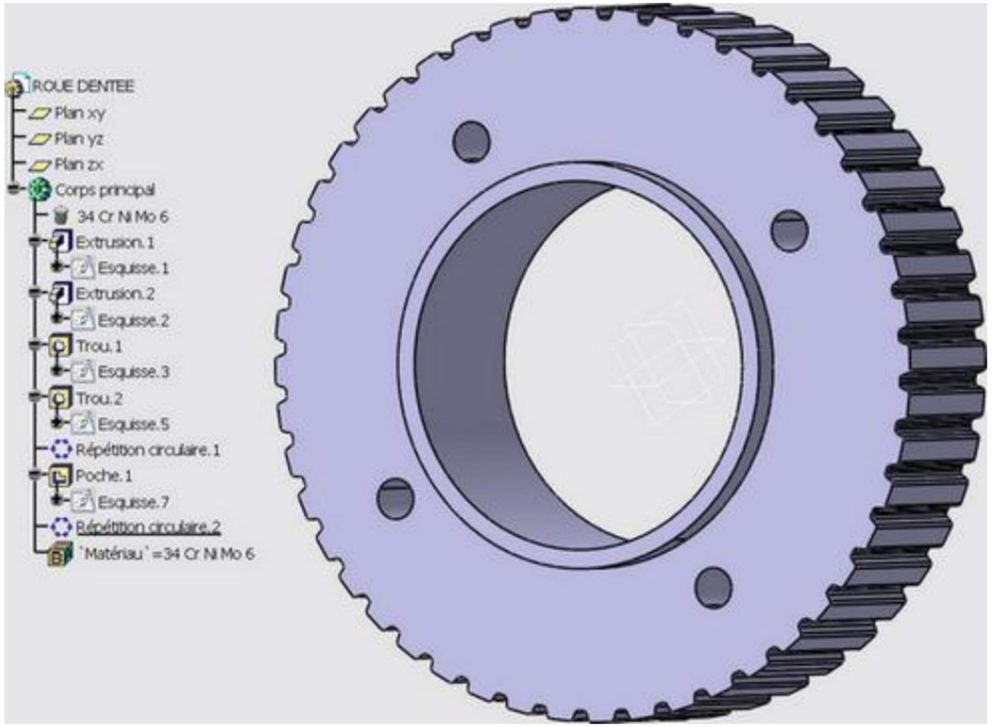
 **APPLY MATERIALS**

Ac improvement tab > 34Cr Ni Mo 6

in the **Main Body** tree
(The outline of the part turns red)

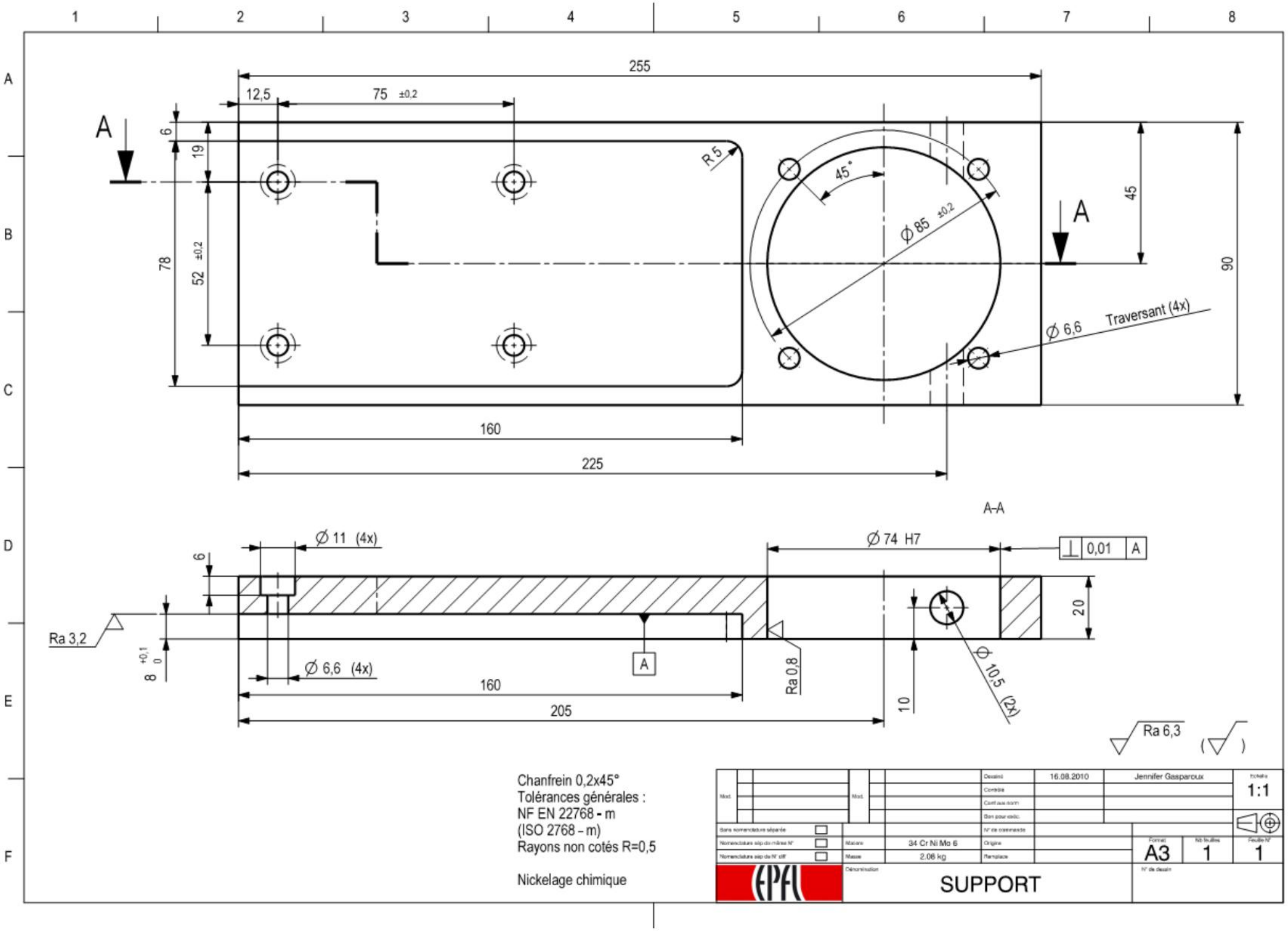
Apply Material
>OK

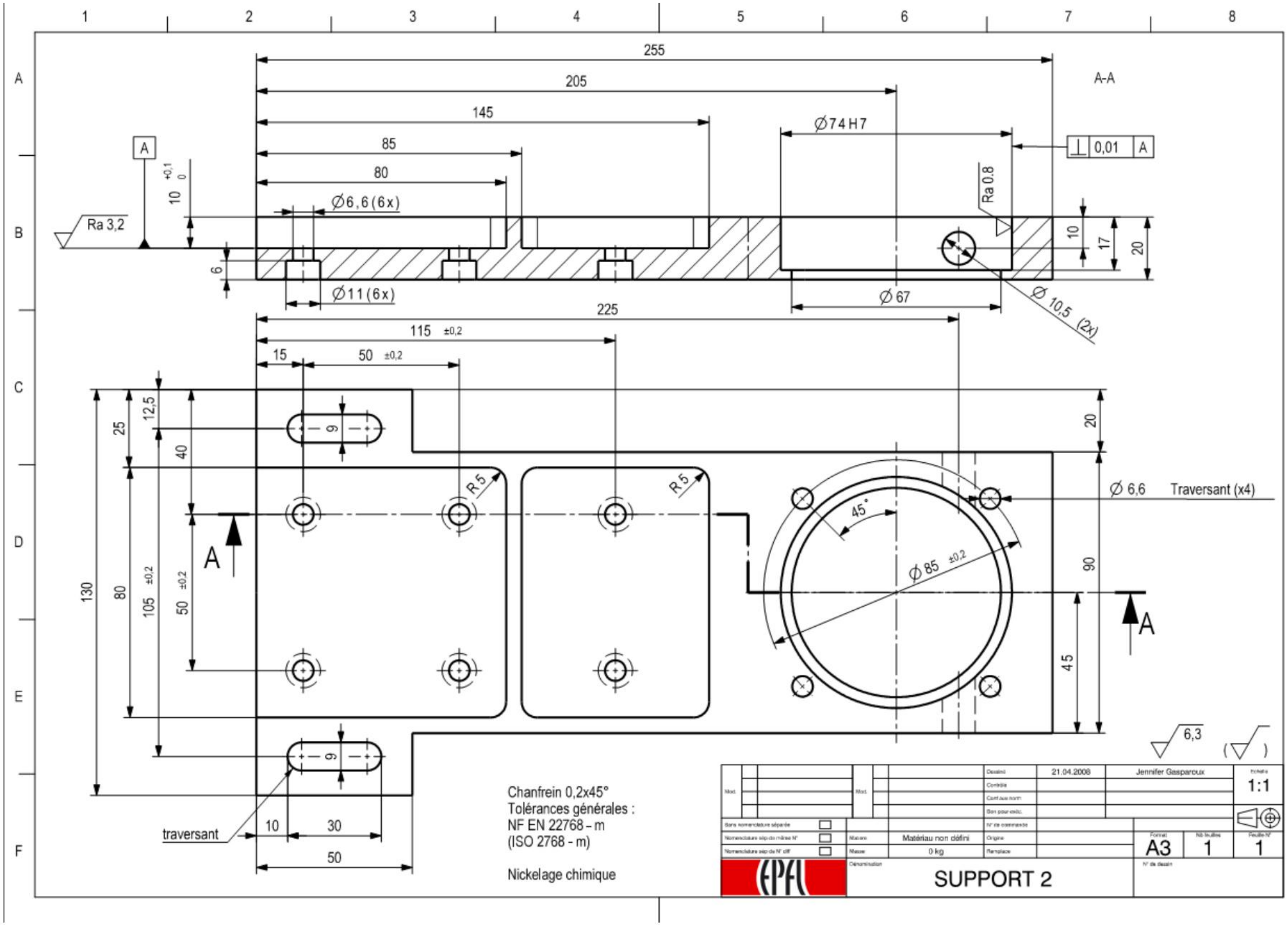
File > **Save**



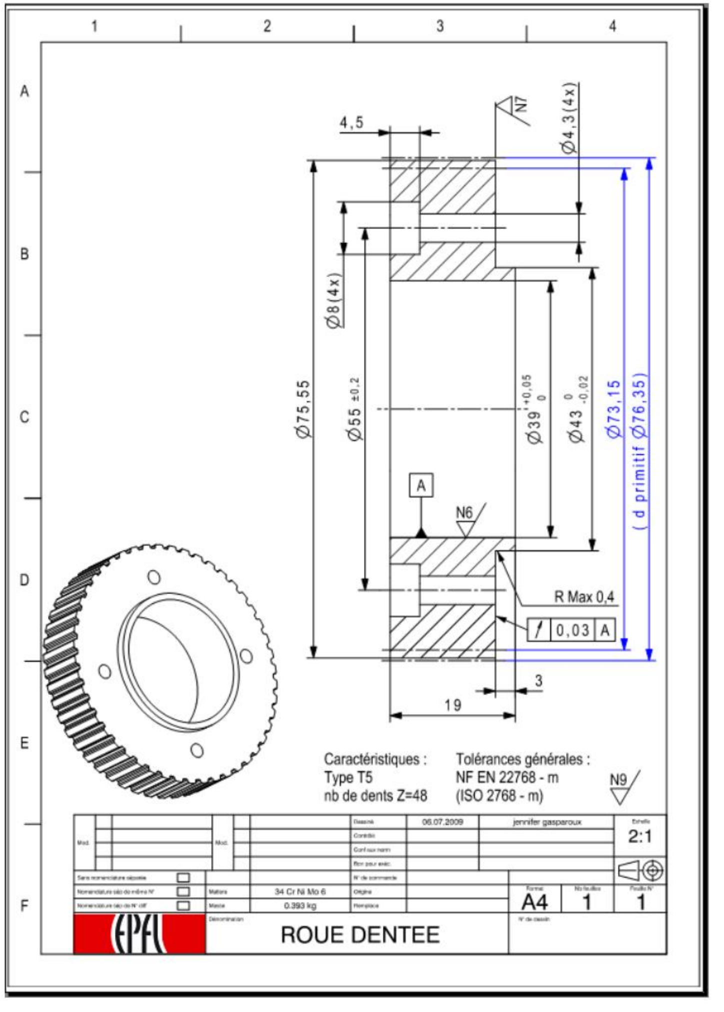
Machine Translated by Google

7. EXAMPLES OF TEST 1



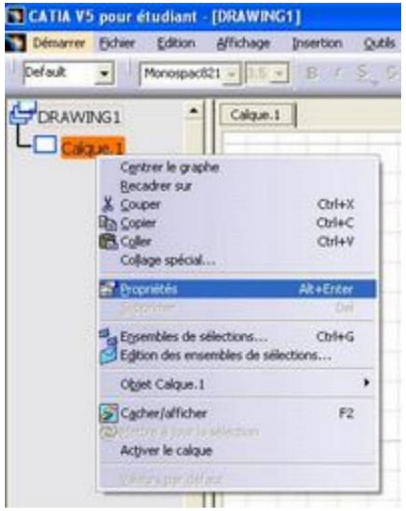

A-A
255
205
145
85
80
Ø74 H7
0,01 A
Ra 0.8
Ra 3,2
Ø6,6 (6x)
Ø11 (6x)
Ø 67
Ø 10,5 (2x)
225
115 ±0,2
15
50 ±0,2
12,5
25
40
130
80
105 ±0,2
50 ±0,2
R5
45°
Ø 85 ±0,2
Ø 6,6 Traversant (x4)
90
45
20
6,3
traversant
10
30
50
Chanfrein 0,2x45°
Tolérances générales :
NF EN 22768 – m
(ISO 2768 - m)
Nickelage chimique
21.04.2008
Jennifer Gasparoux
1:1
Matériau non défini
0 kg
A3
SUPPORT 2

8. BASIC EXERCISE 4: 2D GEAR WHEEL



FINAL RESULT

File menu > **New** > **Drawing > OK >** Standard: **DETAIL ISO** > Sheet style: **A4 ISO** > Orientation: **Portrait** > OK
File menu **> Save** > **L :** Catia > file name: **GEAR WHEEL** > Save



Layer (in tree) > Properties > Scale: **2:1** > OK

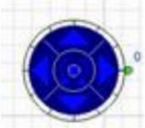
File menu > Open > GEAR WHEEL.CATPart > Open

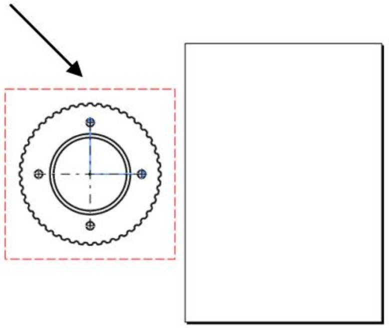
Window menu **> Horizontal Tile**

in the Drawing window (GEAR.CATDrawing)

 **FRONT VIEW**

ZX plane of the 3D window (GEAR WHEEL.CATPart): The selected view is displayed in 2D.

Using the tool , rotate the front view so as to obtain the same view as that of the figure opposite



(Hold) the frame and move the piece to the left of the A4 sheet

next to the view to confirm the position.

CARTRIDGE CREATION

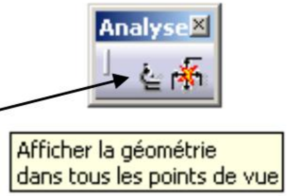
The **A4** cartridge appears on the screen.

The name of the **GEAR WHEEL** drawing appears in the title block. To modify the cartridge, proceed as explained on page 36.

BROKEN CUT > on the vertical axis of the part (above the piece)

> **2x** on the vertical axis of the room (below the room > plan (A4 sheet) in the center of to place the cut.

View > Toolbars > Analyze >
>Show geometry in all viewpoints

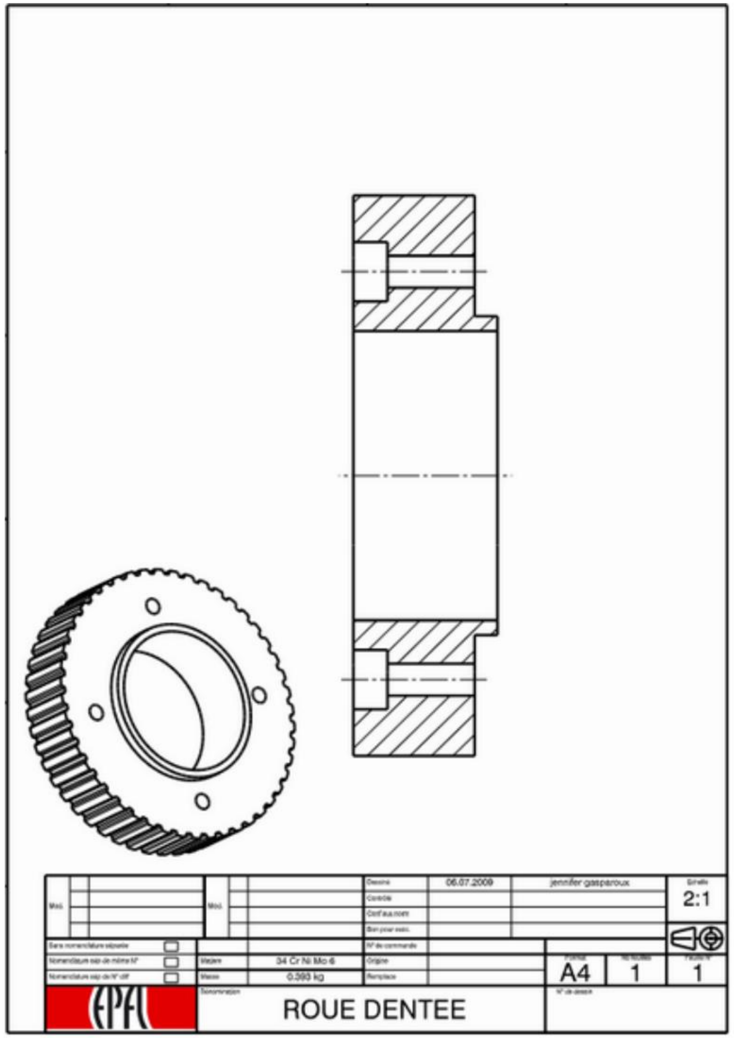


on the different views (front view and section)
This tool allows you to visualize the position of the 3D part for each view of the drawing.

ISOMETRIC VIEW (behind FRONT VIEW) > on "WHEEL DENTEE" (in the tree structure of the 3D window) > choose the view where you can see the most details (see figure)

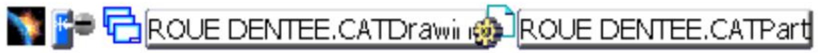
Position the view somewhere outside the A4 sheet > next to confirm

the position > **2x** on this view > on the frame of this view > Properties > Scale: 1:1 > OK
Reposition the view on the A4 sheet, at the bottom left of the section.

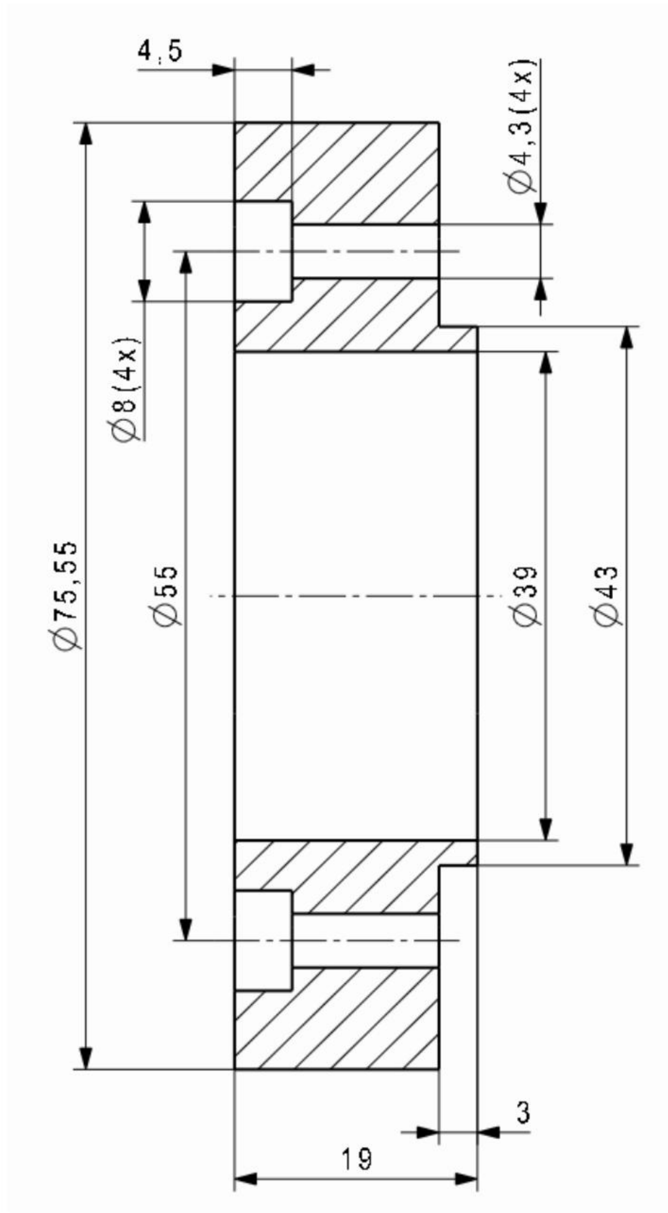


Checking links

File menu > **Office** > check that the two documents are side by side



close **office**



DIMENSION OF THE BROKEN CUT For more visibility now, you can shrink the 3D window and enlarge the Drawing window.

Activate view frame **(2x** on the frame, the latter turns red)

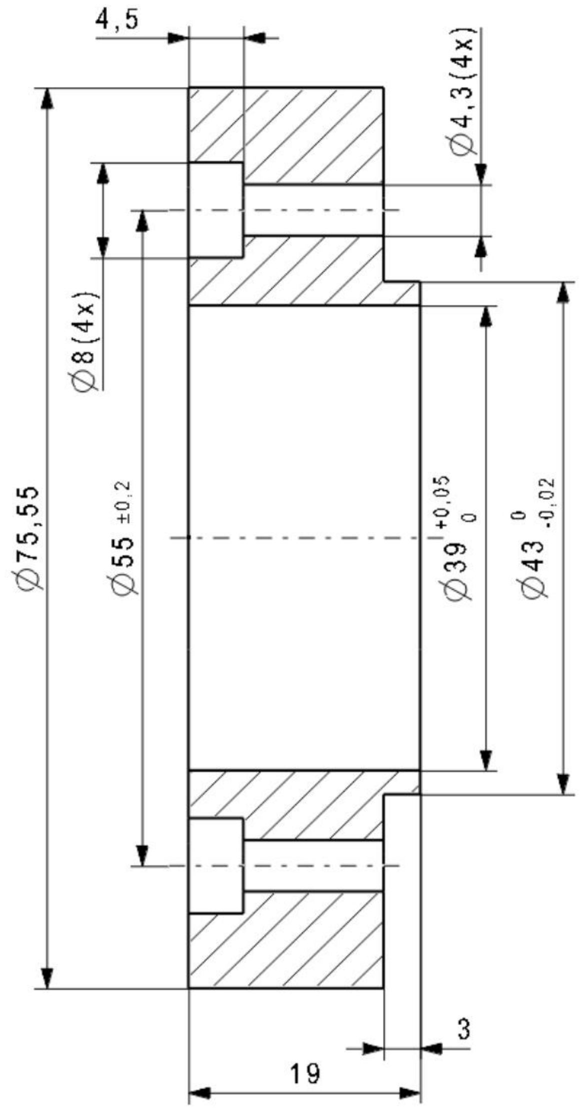
QUOTATIONS

Create dimensions **4.5; 19; 3**

DIAMETER QUOTATIONS

Create **Ø 8; Ø 4.3; Ø55; Ø39; Ø43; Ø 75.55** on the **Ø 8** > **Properties > Dimension texts** tab > to the right of Main value enter **(4x) >** OK

Do the same with **Ø 4.3**



on **Ø 55** > **Properties >** Tolerance tab **> Main Value:** select
10-4 +6 TOL_NUM2 >
Upper value : enter **+0.2** >
Lower value : enter **- 0.2** > OK

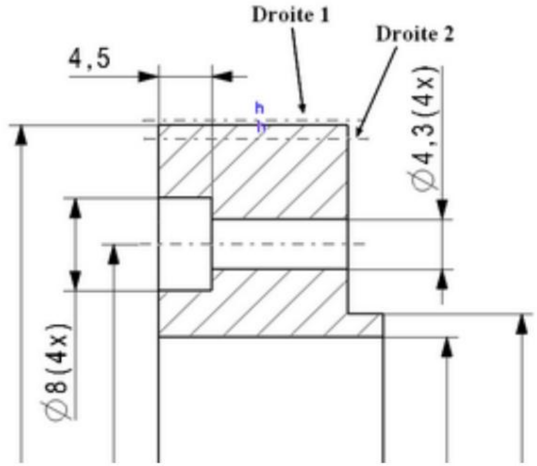
Do the same with **Ø 39; Ø 43** (see the image on the right for the corresponding tolerances)

LAW

Create a horizontal line above the wheel (the **line** must be longer than the thickness of the wheel)
If necessary, it is possible to deactivate the snapping of the points, which allows greater flexibility for setting the line; to this, click on the icon .

On the **right** > **Properties** > **Chart** tab
> **Line: 4 > Thickness: 2 >** OK

2x on the **right (Editing the right)**
> End 1 > V: 38.175 > End 2
> V: 38.175 > OK (leave
default values for H)

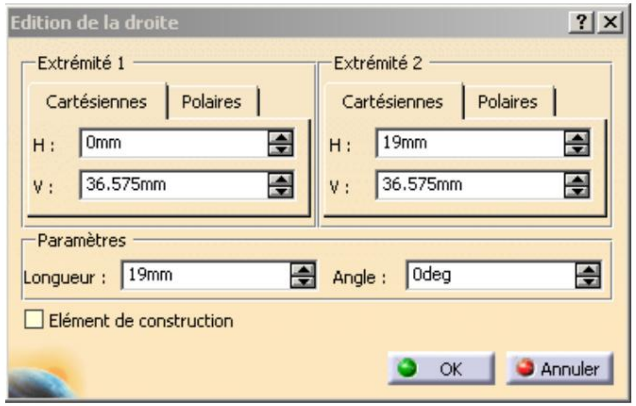


LAW

Create a horizontal line above the wheel (the **line** must be longer than the thickness of the wheel)

On the **right** > **Properties** > **Chart** tab
> **Line: 4 > Thickness: 2 >** OK

2x on the **right (Editing the right)**
> End 1 > V: 36.575 > End 2
> V: 36.575 > OK (leave
default values for H)



Select the two lines (right **1 >** hold the **Ctrl** key pressed > the **right 2)**

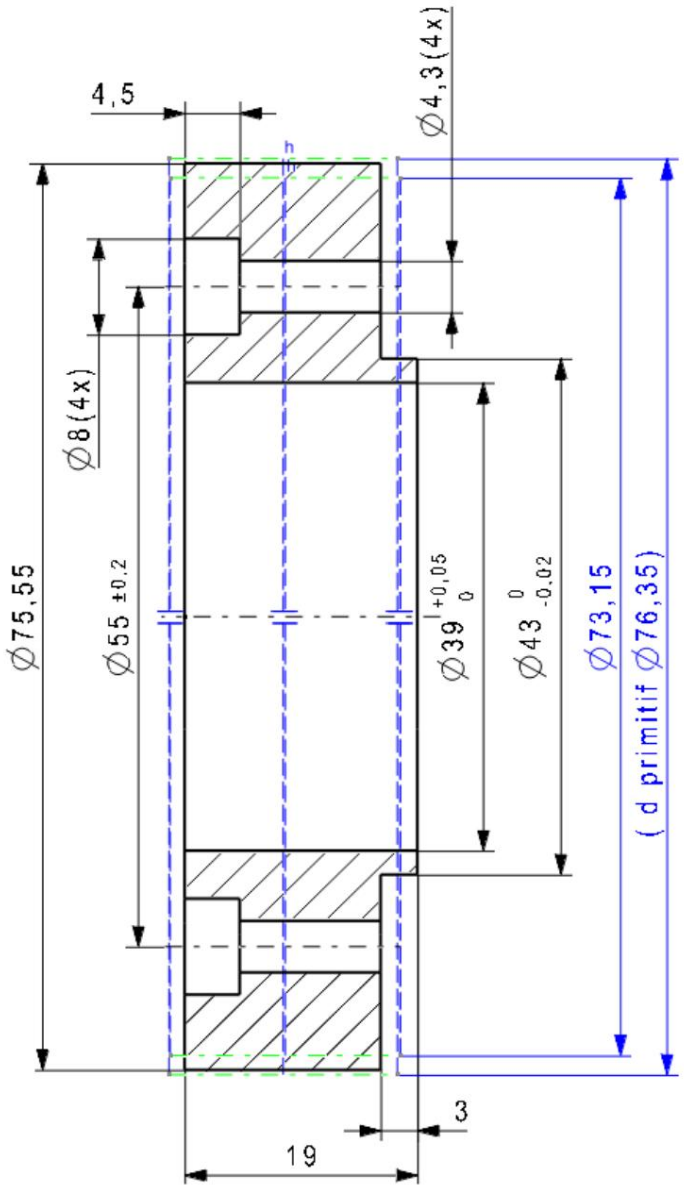
MIRROR > horizontal axis

DIAMETER QUOTATIONS > right **1** > its symmetry

DIAMETER QUOTATIONS > the **right 2** > its symmetry

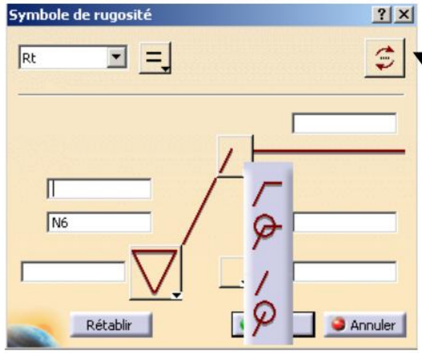
Ø76.35 **dimension > Properties > Dimension text** tab **>** to the left of **Main Value** enter **(dprimitive >** to the right of **Main Value** enter) **>** OK

To hide the **h** symbols above the created lines, on each of symbols and **space bar.**



ROUGHNESS SYMBOL

Inner surface (Ø39) of the gear wheel > enter **N6** above the symbol > Choose the **Roughness** symbol corresponding to the figure opposite > **Ok >** position the symbol



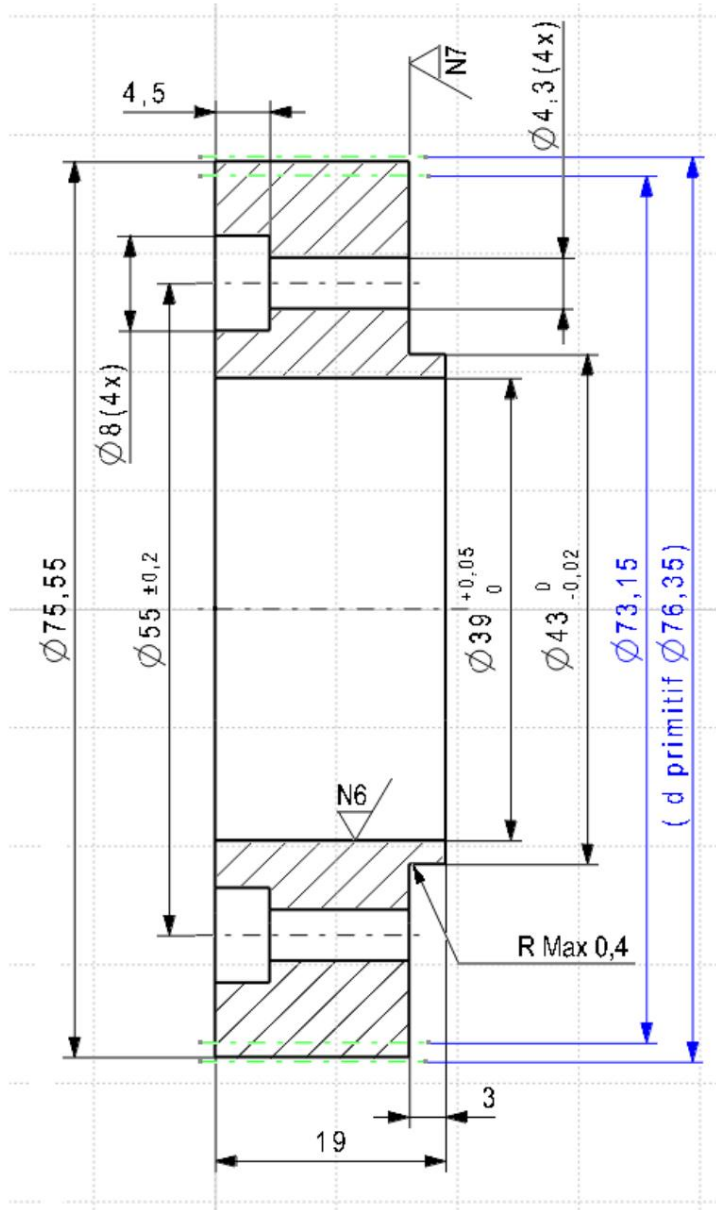
To change the orientation of the symbol.

In the same way, place the **Roughness** symbol N7 as shown in the figure opposite.

Note : for more details on the standardization of these symbols, see the **Fanchon** (pages 116-117)

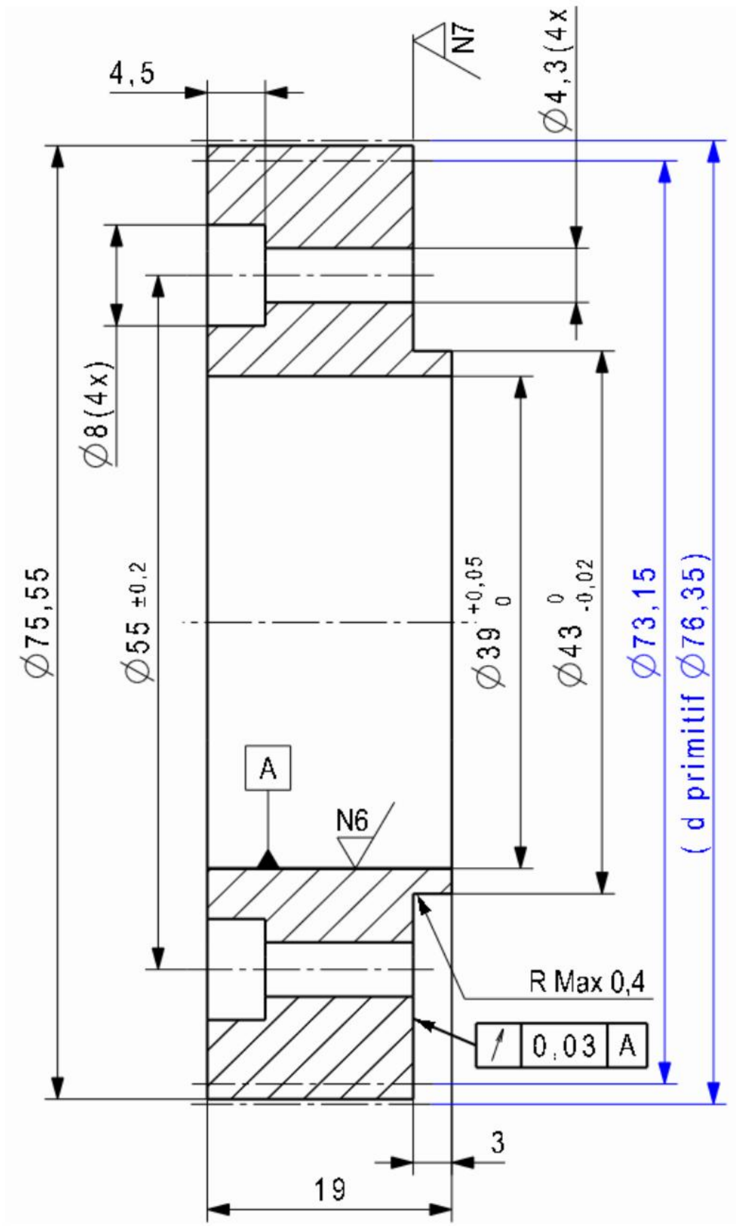
ATTACHED TEXT (behind TEXT)

the corner of the **Ø43 span >** enter **R Max 0.4 >** OK (position the anchor as needed)



REFERENCE

the inner surface (Ø39) of the toothed wheel > set the reference > enter **A** > OK



GEOMETRIC TOLERANCE (behind **REFERENCE)**

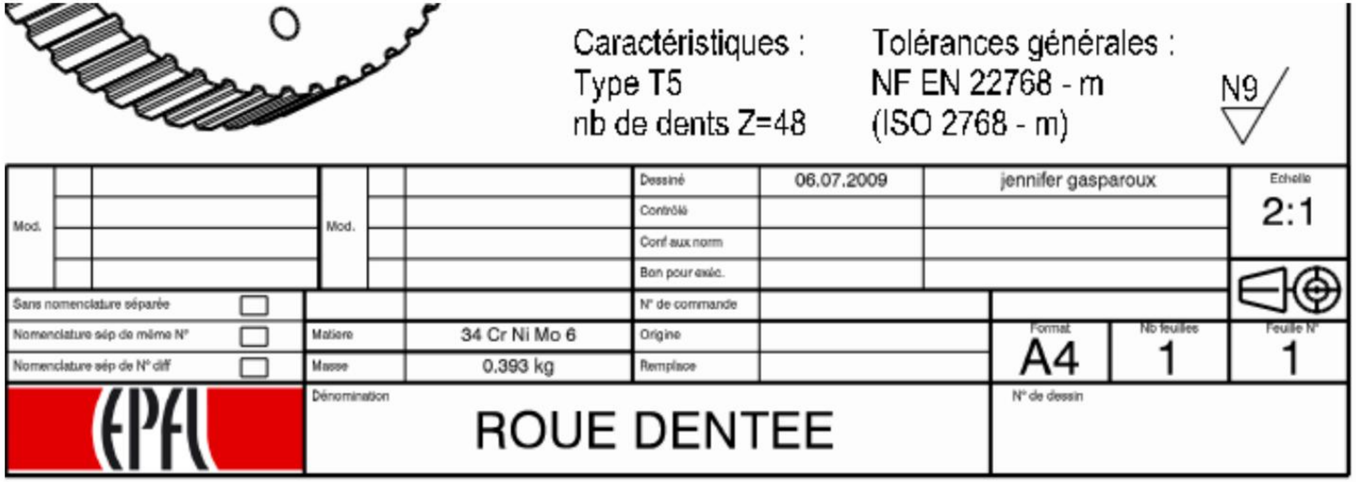
the straight surface of the toothed wheel (Ø75.55) > Select the symbol **single beat** **> Tolerance: 0.03 > Reference: A >** OK
(Position the anchor if necessary)

Note: for more information on geometric tolerances: **Fanchon** (chapter 10) the text as needed) above the title block > in **Text editor** write: **Characteristics:...(see** the image below) **> TEXT** > OK (position

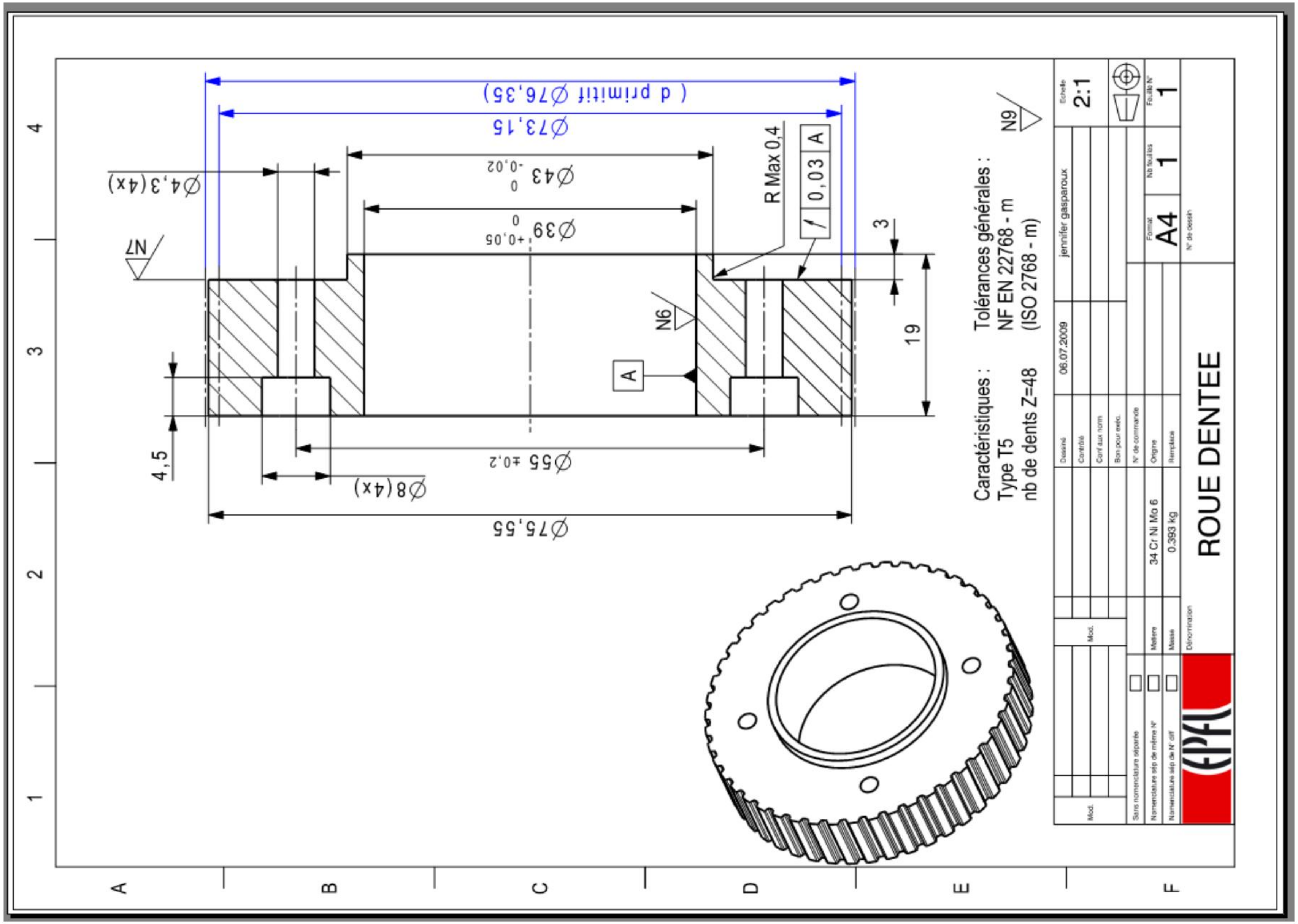
TEXT > above the cartridge > in **Text editor** write: **General tolerances:...(see** the image below below) **>** OK (position the text as needed)

ROUGHNESS SYMBOL

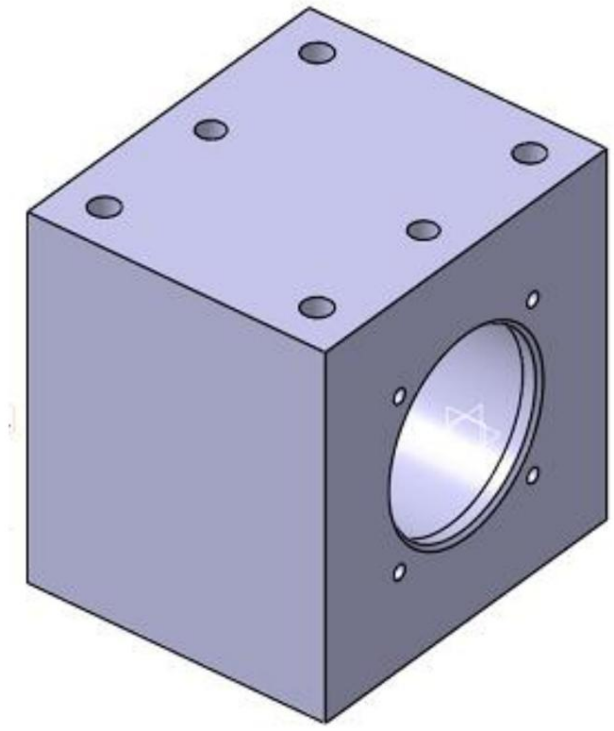
above the title block > enter **N9** above the symbol > Choose the **Roughness** symbol corresponding to the figure below below > OK (position the symbol if necessary)



File menu > Save


(d primitif ∅76,35)
∅73,15
∅43 0 -0,02
∅39 +0,05 0
∅4,3(4x)
R Max 0,4
0,03 A
N7
N6
N9
A
3
19
4,5
∅55 ±0,2
∅8(4x)
∅75,55
Caractéristiques :
Type T5
nb de dents Z=48
Tolérances générales :
NF EN 22768 - m
(ISO 2768 - m)
2:1
jennifer gasparoux
06.07.2009
A4
1
1
34 Cr Ni Mo 6
0,393 kg
ROUE DENTEE

9. BASIC EXERCISE 5: 3D ROLLING BLOCK



FINAL RESULT

Start Menu > **Part Design** > enter part name: **BEARING BLOCK** > OK
File menu **> Save > L: catia** > file name: **BLOCK BEARING** > Save

REALIZATION OF THE BLOCK 82 x 82 x 70

the **ZX** plane in the tree structure > **SKETCH**

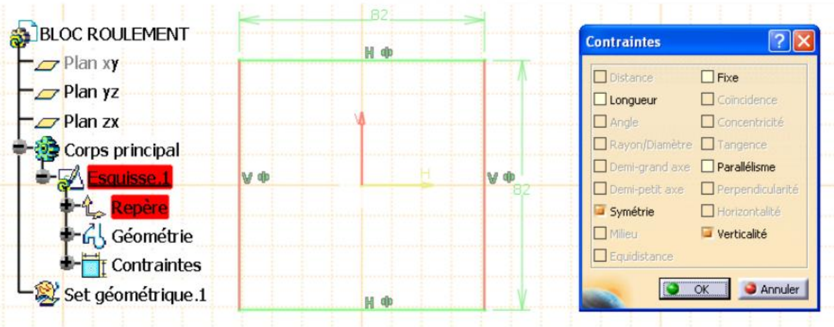
RECTANGULAR > for the 1st corner for the 2nd corner.

CONSTRAINTS > on each of the vertical sides of the rectangle > set the dimension > **2x** on the odds value, enter **82** **>** okay

CONSTRAINTS > on each of the horizontal sides of the rectangle > set the dimension > **2x** on the odds value, enter **82** **>** okay

on one of the vertical sides of the rectangle > keep the **Ctrl** key pressed: (the three selected elements turn red) > release the **Ctrl** key the other vertical side and the vertical axis **V**

Note: Select items in the order listed! Otherwise the block will not be centered on the origin.



SELECTED CONSTRAINTS IN A BOX OF DIALOGUE

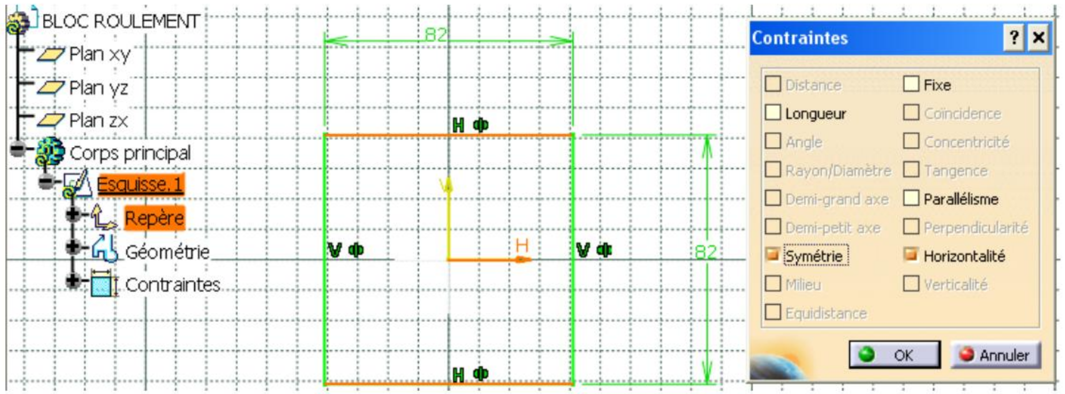
Symmetry and verticality > OK

on one of the horizontal sides of the rectangle >

keep the **Ctrl** key pressed: the other side

horizontal and the horizontal axis **H** (the three selected elements turn red) > release the **Ctrl** key

Note: Select items in the order listed! Otherwise the block will not be centered on the origin



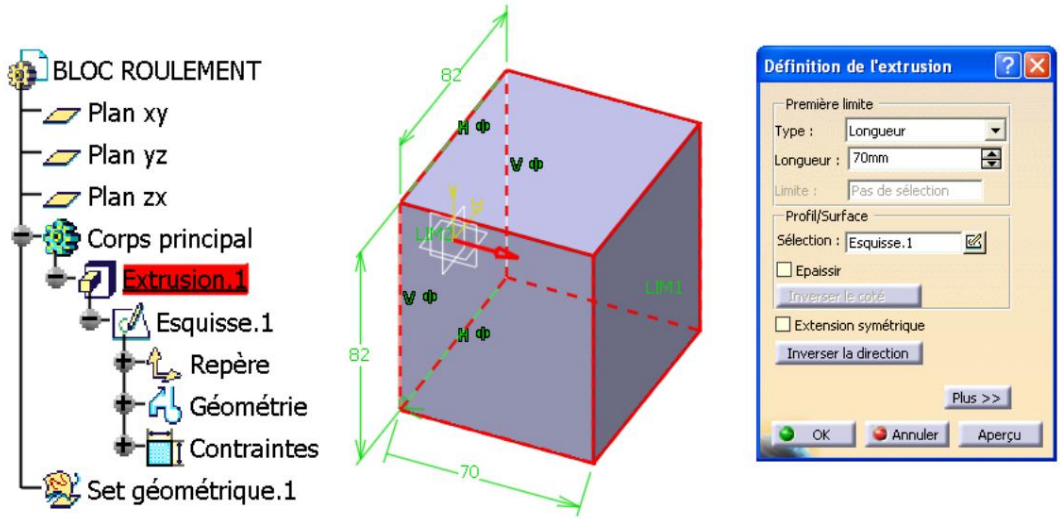
SELECTED CONSTRAINTS IN A DIALOG BOX

Symmetry and horizontality > OK

LEAVING THE WORKSHOP

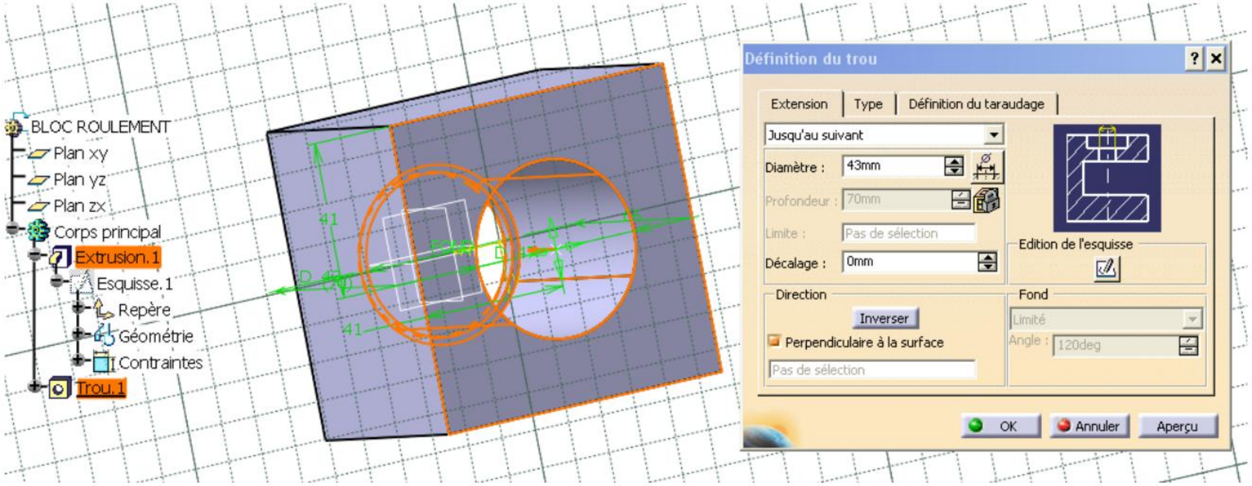
EXTRUSION > Type : Length > enter **70** (reverse direction if necessary) > selection: last sketch created

okay



THROUGH HOLE Ø 43 BLADE Ø 47 prof. 66mm

HOLE



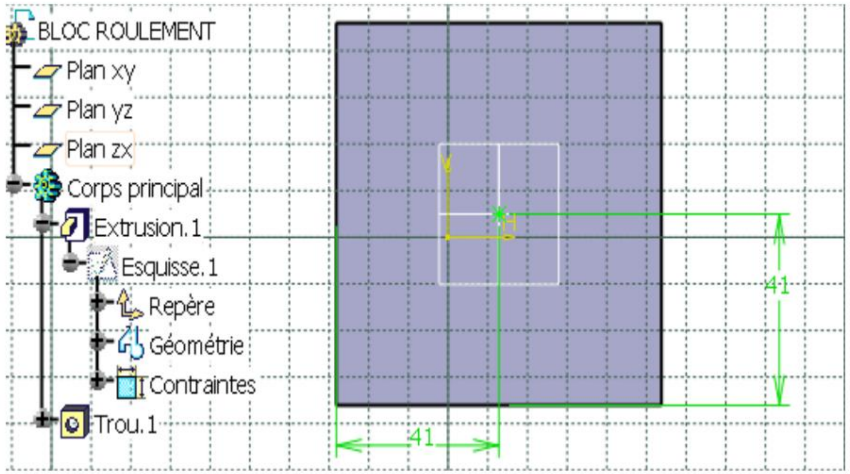
a square face of the block (the hole appears in red) and the **Hole Definition** window appears

Extension tab : Up to next >
Diameter : 43mm

CONSTRAINTS > hole center (white dot) > a vertical edge > set dimension > **2x** on the odds value, enter **41 >** okay

CONSTRAINTS > hole center > a stop horizontal > set dimension > **2x** on the odds value, enter **41** **>** okay

LEAVING THE WORKSHOP (we find the definition of the hole)



Type tab : Laminate > **Diameter : 47** mm > **Depth : 66** mm > OK > OK

4 BLANK HOLES TAPPED M4 x 10/15

HOLE

the face of the block opposite to the one chosen previously (the hole appears in red)

Extension tab : Blind > **Bottom:** V -shaped

Type tab : Simple

Thread **definition** tab

enable: **Tapped** >

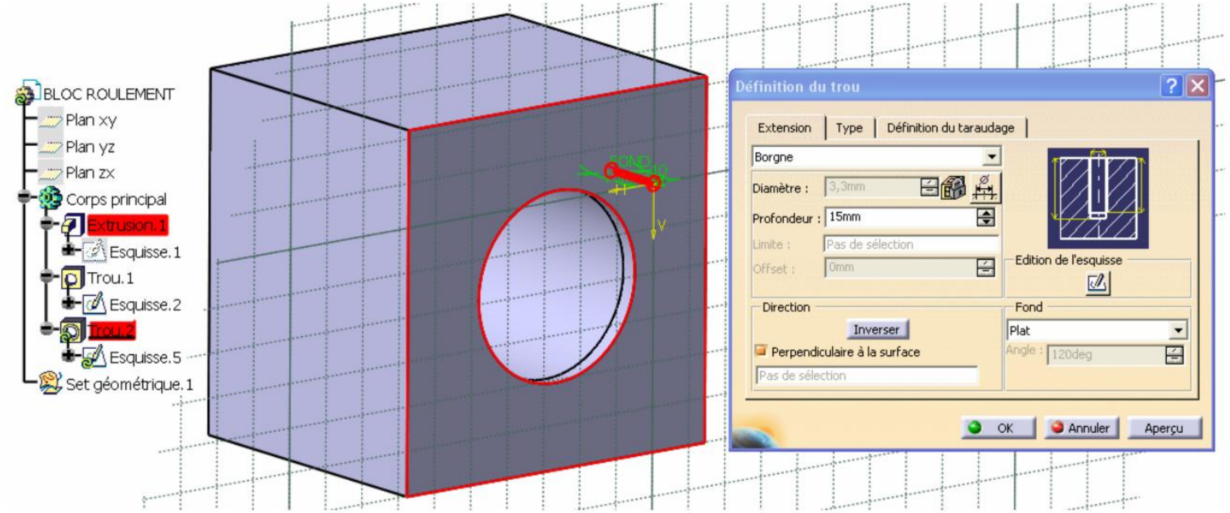
Type : Metric Coarse Pitch >

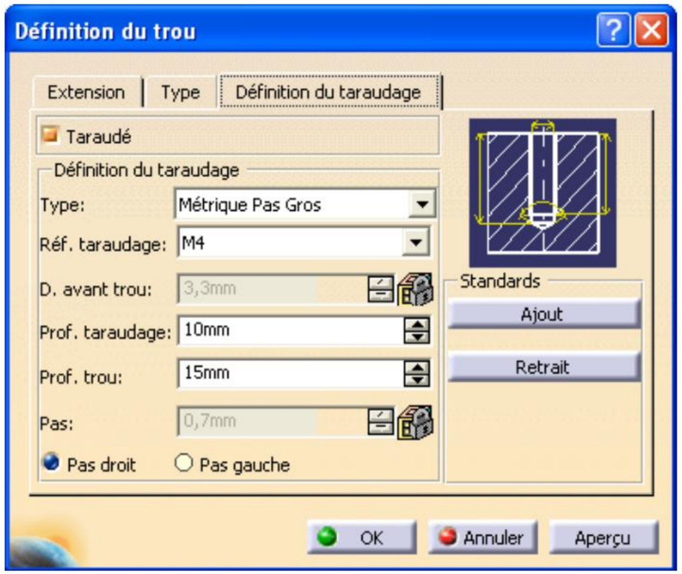
Ref. thread : M4 >

Teacher. thread : 10 mm >

Teacher. hole : 15mm >

activate: **No right** > OK

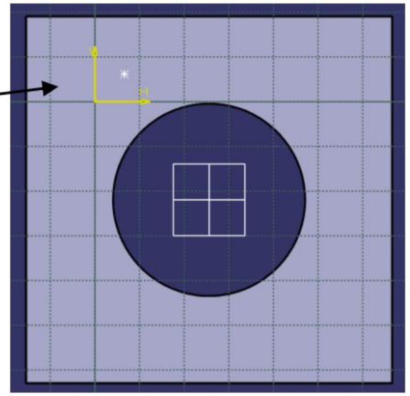




Extension tab > **Editing the sketch**

on the center point of the hole, keep it pressed and move it from the origin of the axes

beside the view

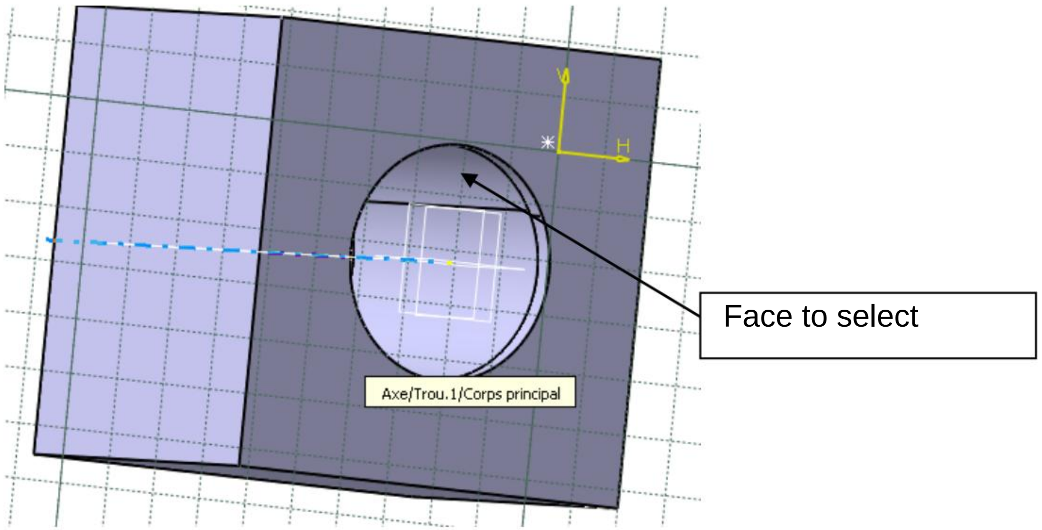


PROJECTION OF 3D ELEMENTS

Rotate (page 10) the view so that you can see the inside of the bore made previously

cylindrical face of the through hole Ø 43 > a white line appears, it represents the axis of the selected surface.

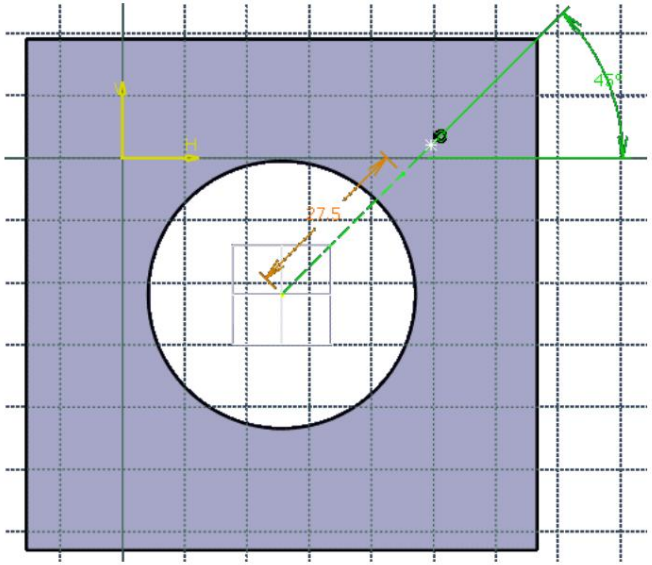
Note: The Project 3D Elements operation projects 3D elements into the sketch.
Here, this operation made it possible to project the axis of the cylinder on the plane of the sketch. A point is thus obtained which appears in red.



To return to the sketch plane : **NORMAL VIEW** located at the bottom of the screen.

RIGHT > draw a line between the red point (projected previously) and the center of the hole (white point)

beside the view



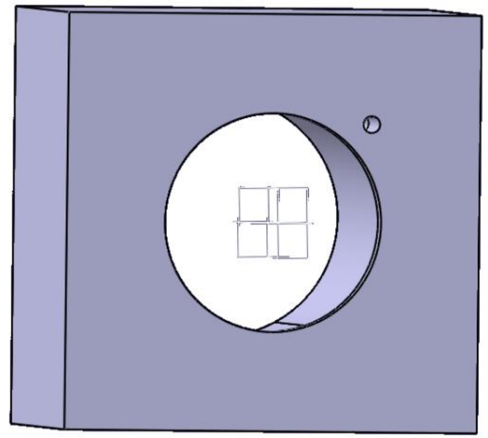
Note : the straight line automatically becomes **Construction element** (straight in dotted)

CONSTRAINTS > line previously created > horizontal axis >

set dimension > **2x** on the dimension value > enter **45° >** OK

CONSTRAINTS > straight line previously created > set the dimension >

2x on the dimension value > enter **27.5 >** OK



LEAVING THE WORKSHOP (we find ourselves in the definition of the hole)

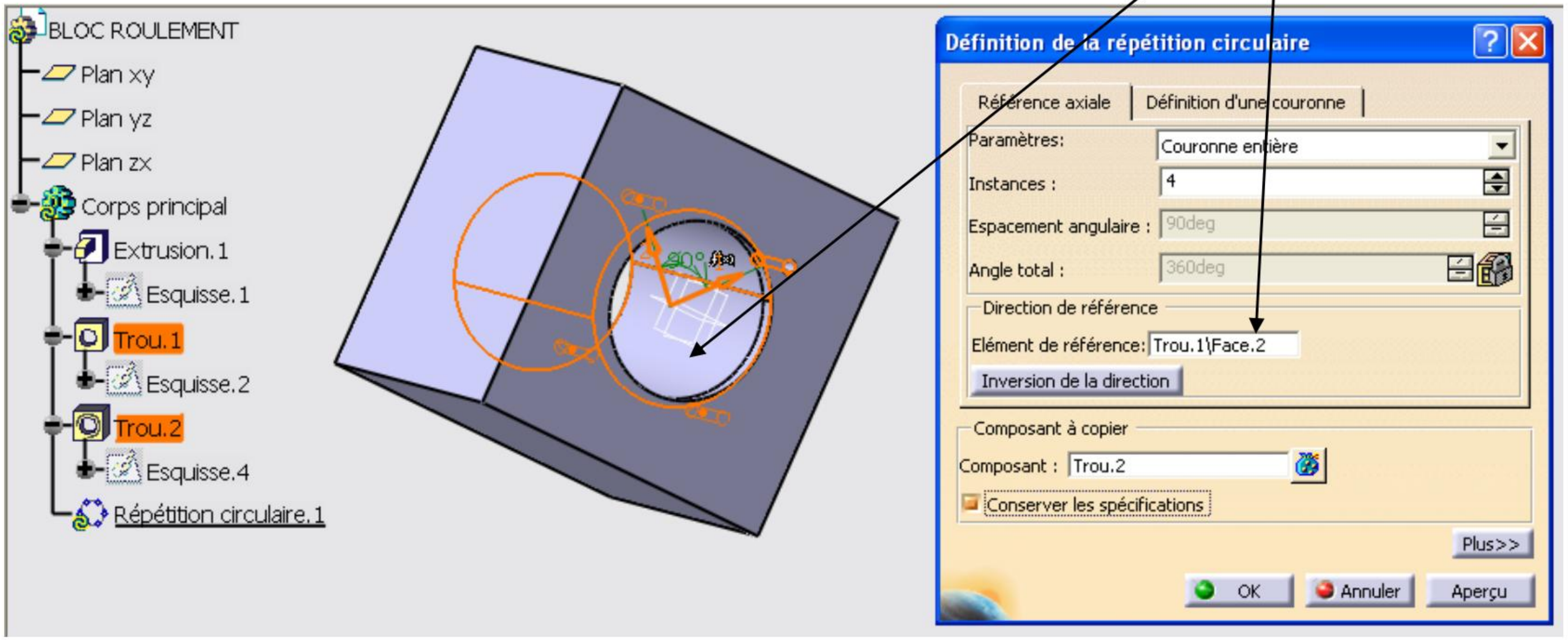
okay

Circular repeat

Hole.2 (= last hole created) in the tree structure (the hole turns red)

CIRCULAR REPEAT

Cylindrical face to select



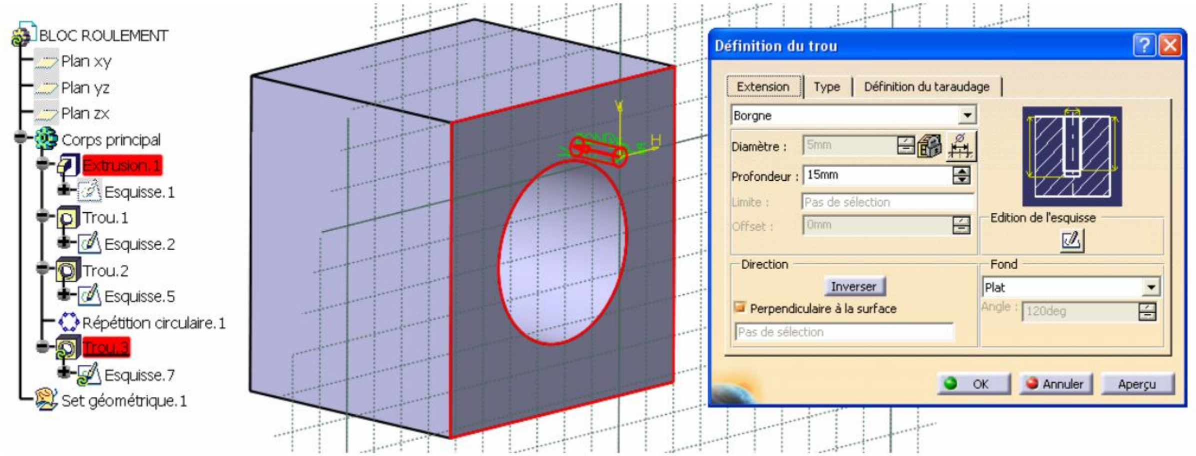
Axial reference tab > **Parameters:** Entire crown > **Instances :** enter 4 > **Reference element :** the cylindrical face (Ø43) of the block > activate: **Keep specifications**

okay

4 BLANK TAPPED HOLES M6 x 10/15

HOLE

the face of the block opposite the 4 previous holes (the hole appears in red)



Extension tab : Blind > **Bottom:** V -shaped

Type tab : Simple

Thread **definition** tab

enable: **Tapped** >

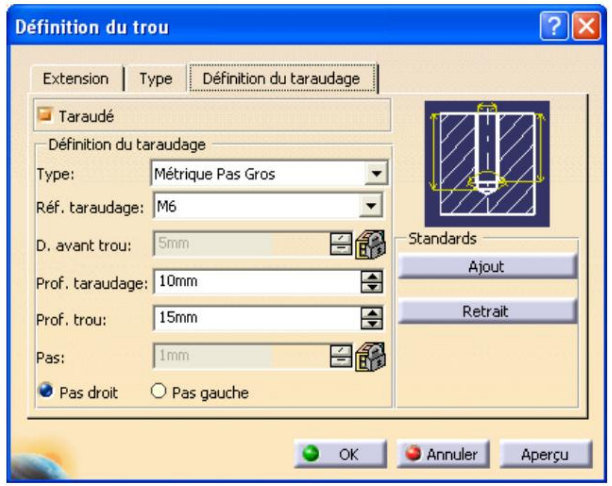
Type : Metric Coarse Pitch >

Ref. thread : M6 >

Teacher. thread : 10 mm>

Teacher. hole : 15mm >

enable: **Not right**



Extension tab > **Editing the sketch**

on the center point of the hole, keep it pressed and move it from the origin of the axes (without this operation, the following dimension is not valid, it is seen as overstressed)

PROJECTION OF 3D ELEMENTS

Rotate (page 10) the view so that you can see the inside of the bore made previously

cylindrical face of the through hole Ø 43 > a white line appears, it represents the axis of the selected surface.

To return to the sketch plane : **NORMAL VIEW** located at the bottom of the screen.

RIGHT > draw a line between the red point (projected previously) and the center of the hole (white point)

beside the view

CONSTRAINTS > previously created line horizontal axis > set dimension > **2x** on the dimension value > enter **45°** **>** okay

CONSTRAINTS > line previously created > set the dimension > **2x** on the dimension value > enter **30 >** OK

LEAVING THE WORKSHOP (we find ourselves in the definition of the hole)

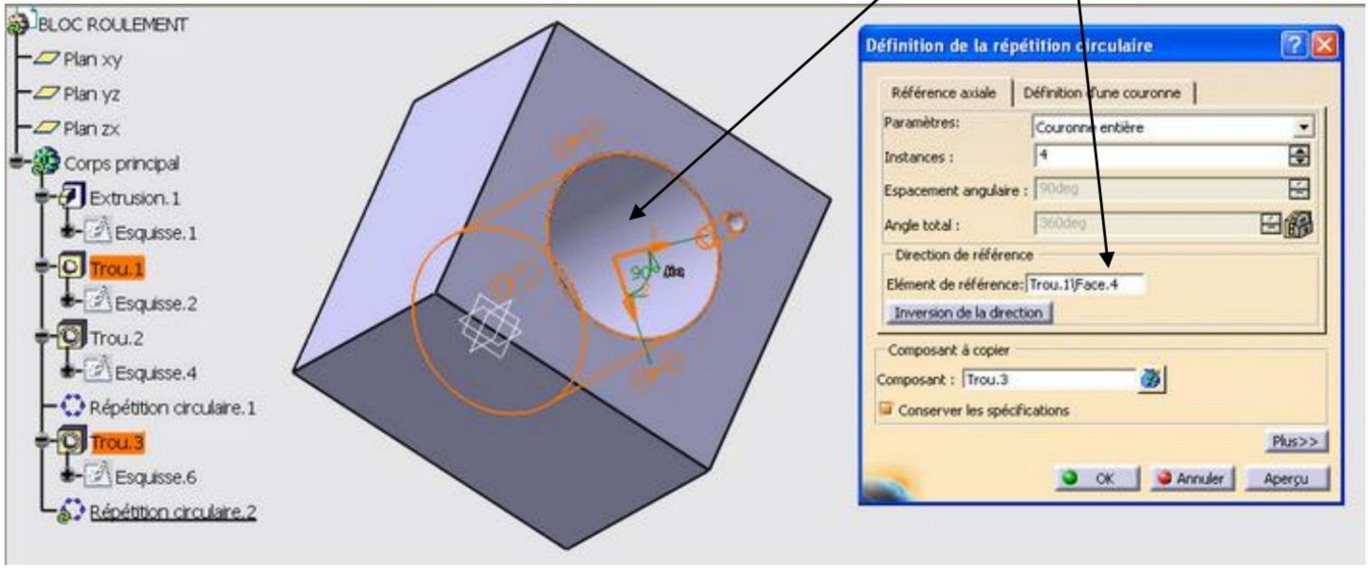
okay

Circular repeat

Hole.3 (= last hole created) in the tree structure (the hole turns red)

CIRCULAR REPEAT

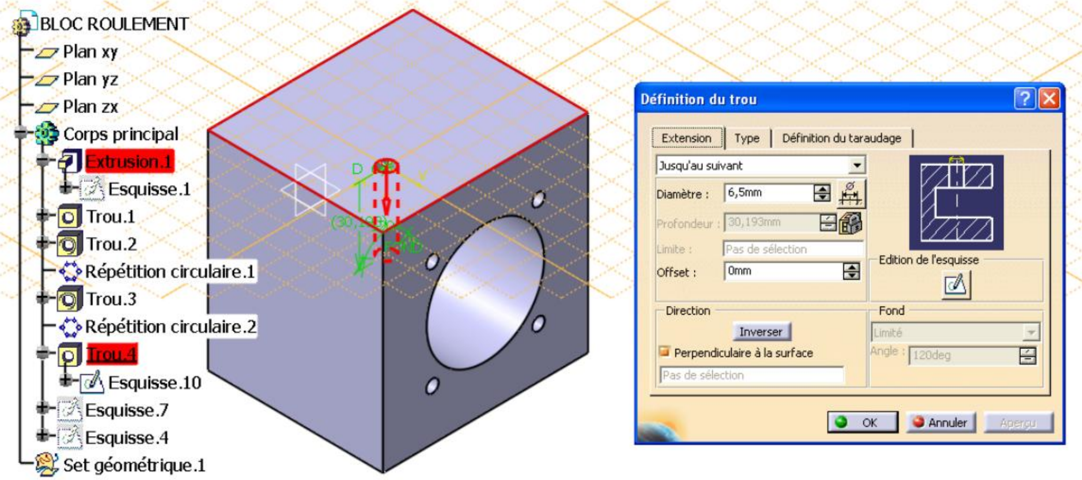
Cylindrical face to select



Axial reference tab > **Parameters:** Entire crown > **Instances :** enter **4** > **Reference element :** the cylindrical face (Ø47) of the block > activate: **Keep specifications**

okay

4 THROUGH HOLES Ø 6.5



HOLE

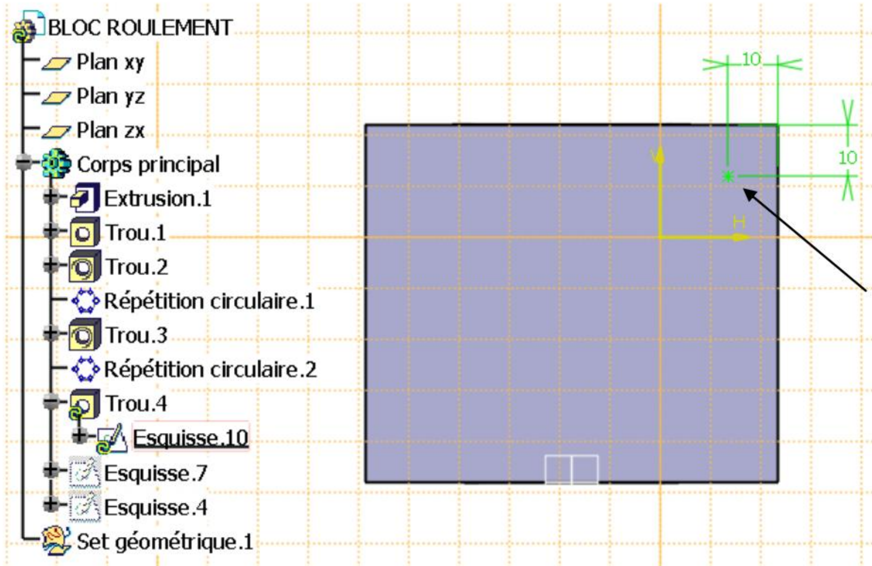
a block face ÿ to the previously chosen face.

Thread **definition** tab : Deselect **thread**

Type tab : Simple

Extension tab : Up to next > **Diameter : 6.5** mm

Editing the sketch



CONSTRAINTS > horizontal edge of the face about > set the dimension > **2x** on the dimension value > enter **10 >** OK

CONSTRAINTS > vertical edge of the face on point > set dimension > **2x** on the dimension value > enter **10 >** OK

LEAVING THE WORKSHOP

okay

Machine Translated by Google

Rectangular repeat

Hole.4 (=last hole created) in the tree structure (hole turns red)

RECTANGULAR REPEAT

tab: **First direction** >
Settings: Instances & Spacing >
Instances : enter **2** >

Spacing : enter **62** >

Reference direction : Plan on which was placed the hole > Reverse direction if necessary

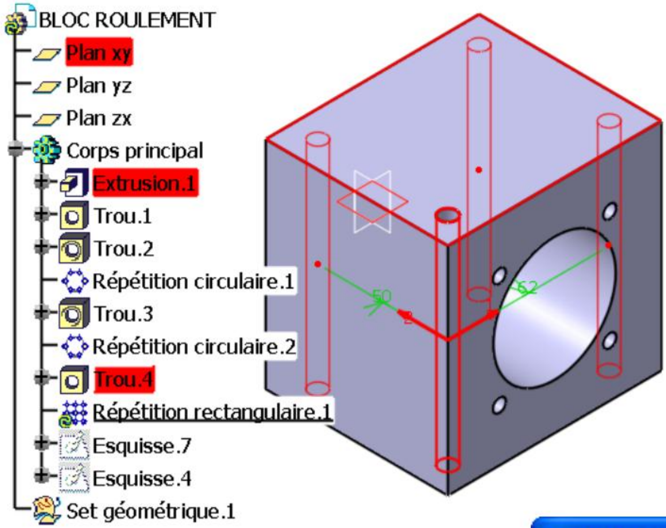
activate: **Keep specifications**

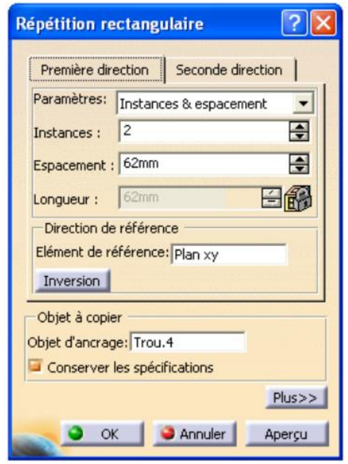
Second direction tab >

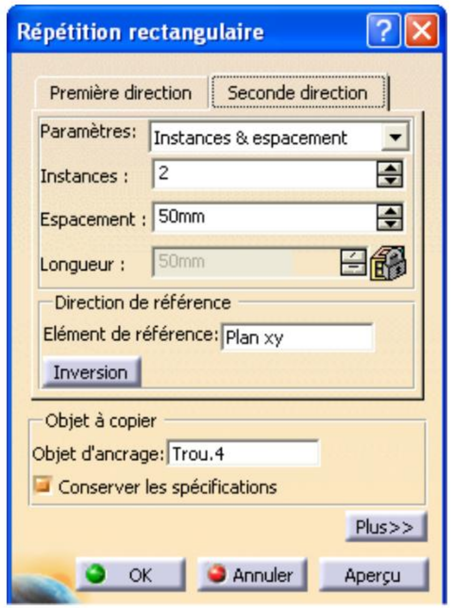
Settings: Instances & Spacing >
Instances : enter **2** >

Spacing : enter **50** > Reverse direction if necessary

okay

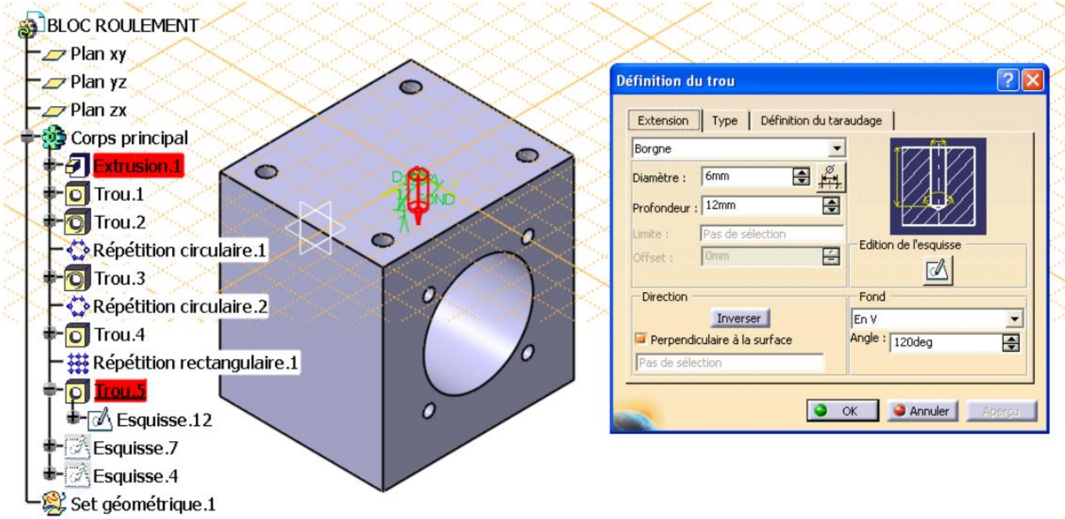






2 BLANK HOLES Ø 6

HOLE



the same face of the block as before (the hole appears in red)

Extension tab : Blind > **Diameter :** enter **6** >
Depth : enter **12** > **Bottom:** In V

Type tab : Simple

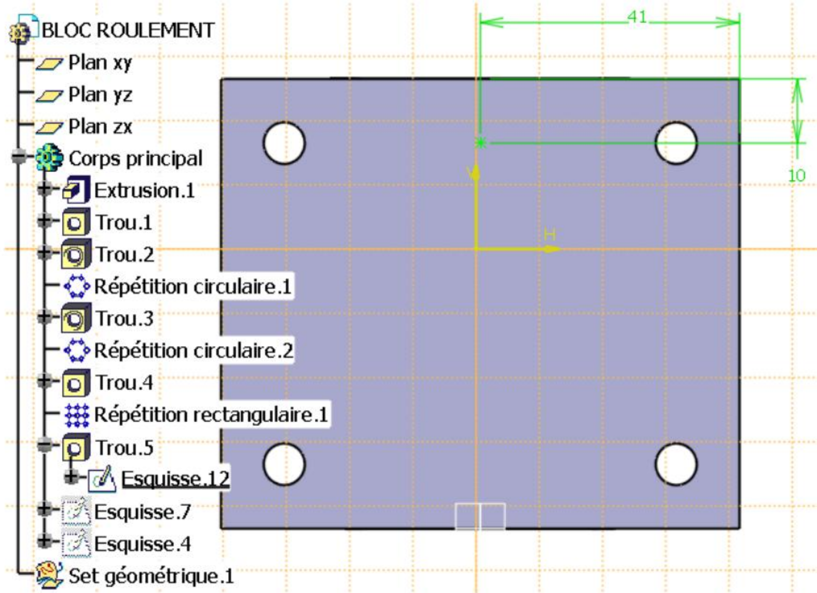
Extension tab > **Editing the sketch**

CONSTRAINTS > longest edge of the face on the point > set the dimension > **2x** > okay

on the odds value > enter **10**

CONSTRAINTS > shortest edge of the face on the point > set the dimension > **2x** > okay

on the value of the dimension > enter **41**



LEAVING THE WORKSHOP

okay

Rectangular repeat

Hole.5 in the tree

Second direction tab >

Settings: Instances & Spacing >
Instances : enter **2** >

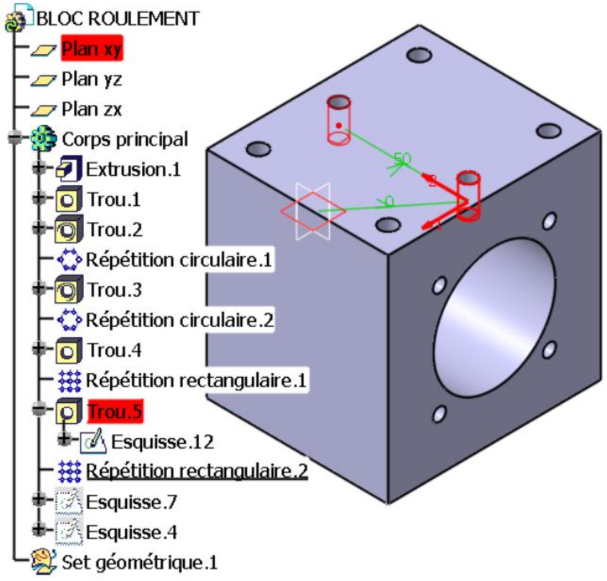
Spacing : enter **50** >

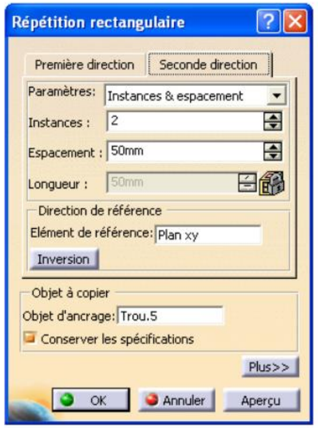
Reference direction : Plane on which the hole was placed > Reverse the direction if necessary activate: **Keep specifications**

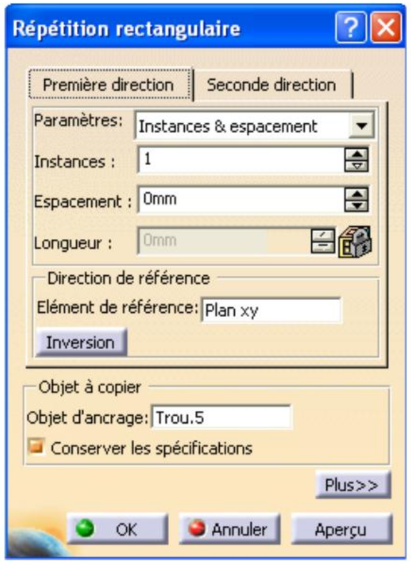
First direction tab >

Settings: Instances & Spacing >
Instances : enter **1** >

okay







BEVELS

  CHAMFER

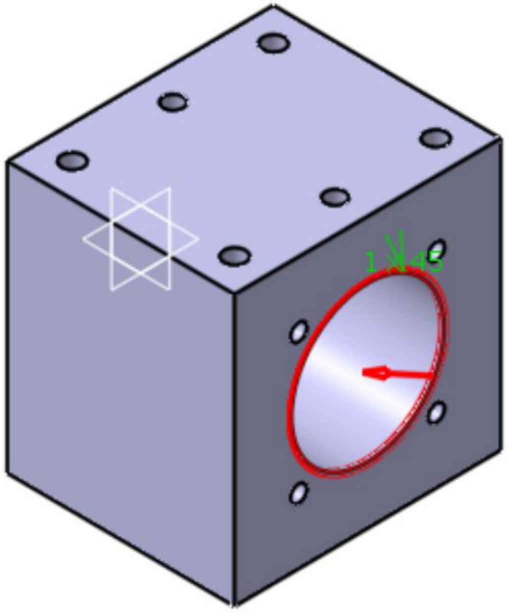
Mode: Length1/Angle >

Length 1: enter **1 >**

Angle : between **45** >

Objects to be chamfered: edge facing the outer face of the Ø 43 hole.

 **okay**



Repeat the operation for the edge facing the outer face of the Ø 47 hole.

APPLICATION OF MATERIALS

APPLICATION OF MATERIALS

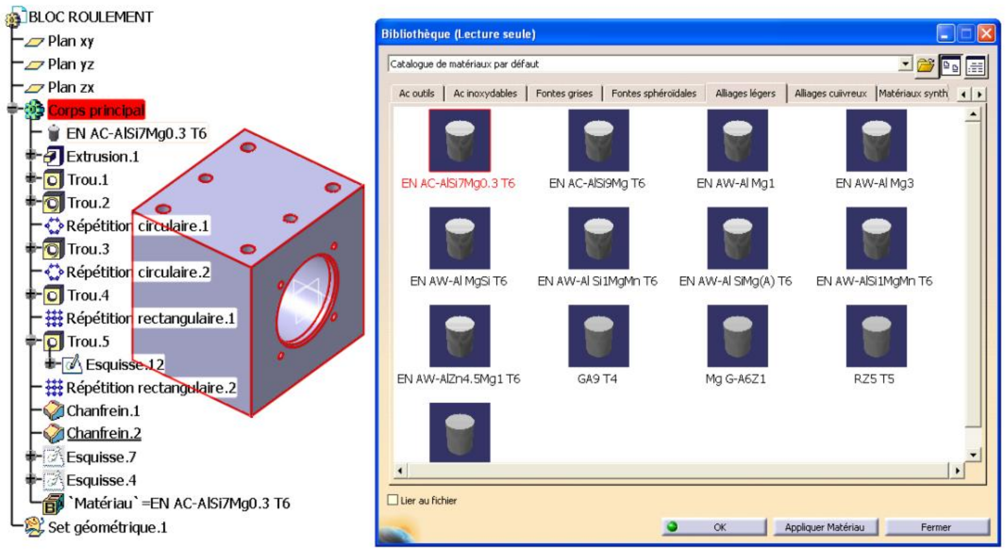
Light alloys tab > EN-AC-AlSi7Mg0.3T6

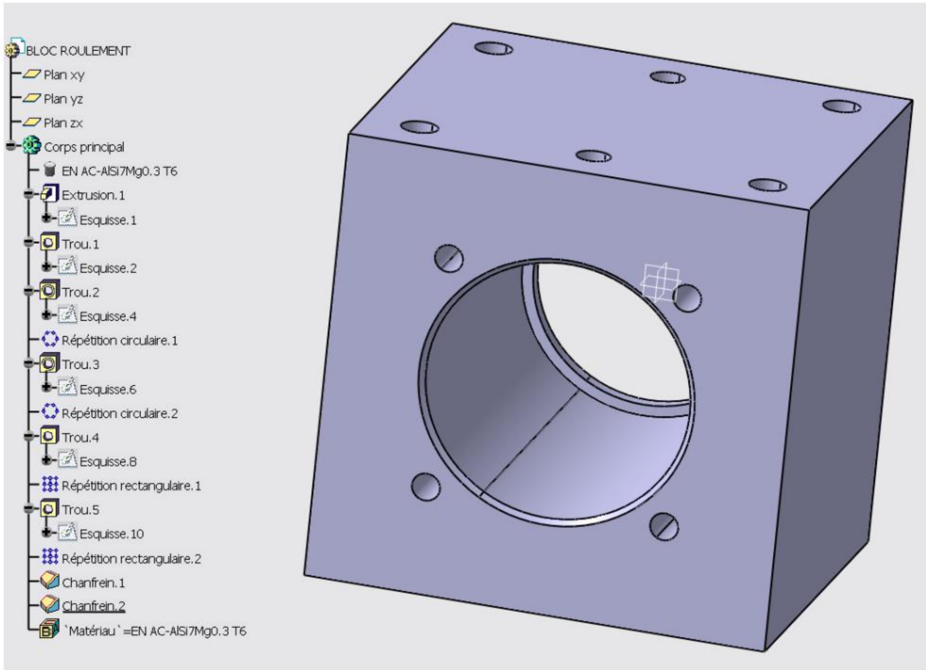
in the **Main Body** tree view (part outline turns red)

Apply Material

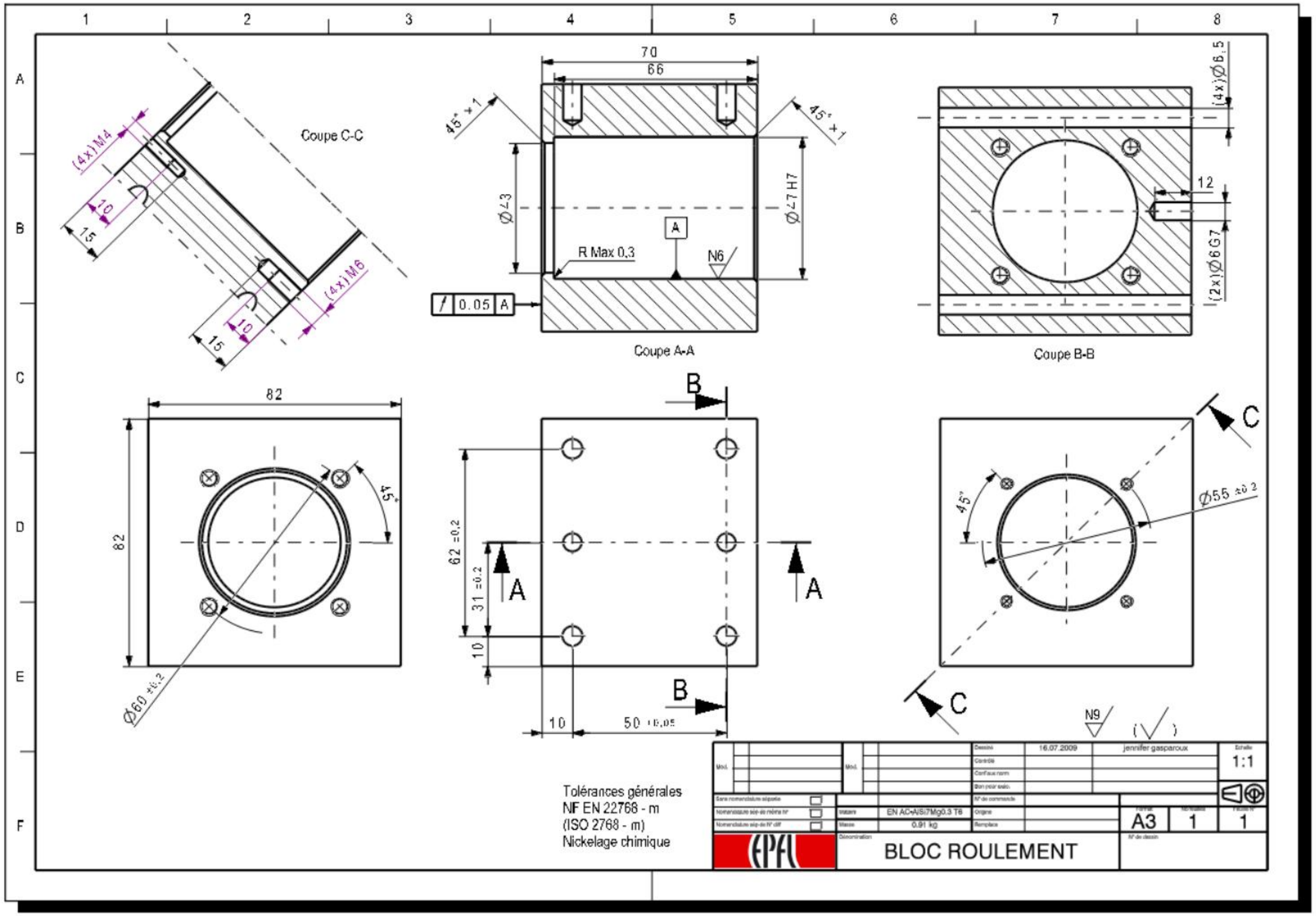
okay

File > Save





10. BASIC EXERCISE #6: 2D ROLLING BLOCK



FINAL RESULT

File menu > **New** > **Drawing > OK** > Standard: **DETAIL ISO** > Shape Style: **A3 ISO** > Orientation: **Landscape** > OK
File menu **> Save** > **L: catia** > file name: **BEARING BLOCK** > OK

File menu > Open >

BLOCK BEARING.CATPart > Open

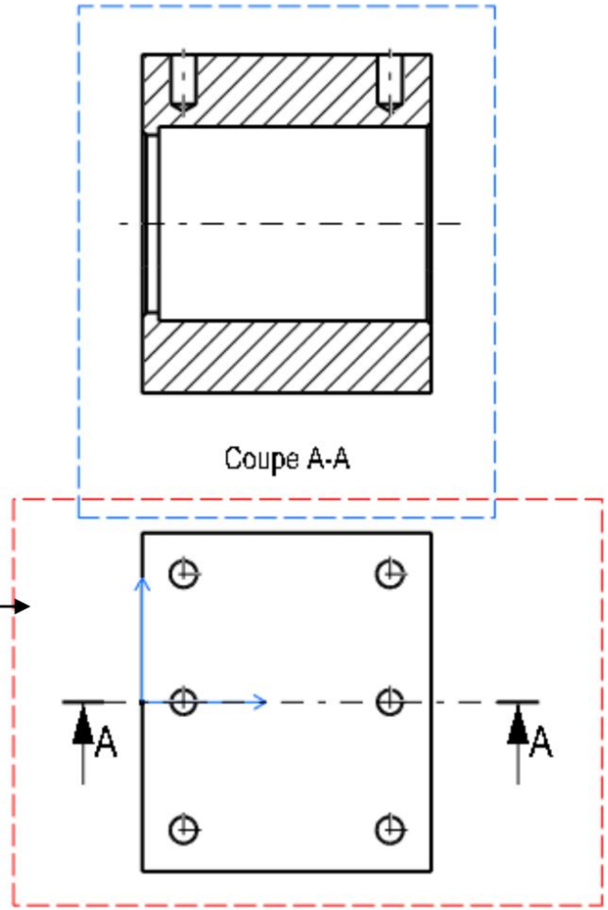
Window menu **> Horizontal Tile**

Creation of the different views

in the Drawing window and **FRONT VIEW**

the surface with the 6 holes. The selected view is displayed in 2D, using the tool

position the view as in the figure



(ATTENTION : Ø43 on the left and Ø47 on the right)

beside the view

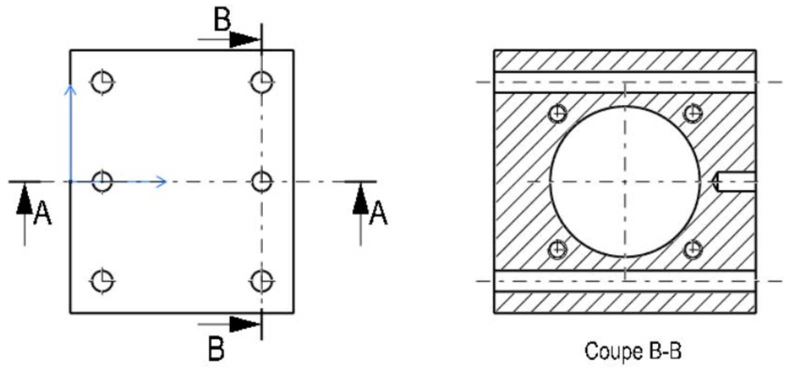
BROKEN CUT > create the **AA** cut

the frame of the broken section AA **> Object AA section** > **Add view name**

2x on writing **>** Clear scale, just leave AA cut > OK Reposition text if needed.

Note : the scale of a view should not be entered if it is the same as that noted in the title block.

BROKEN CUT > create the **BB** cut

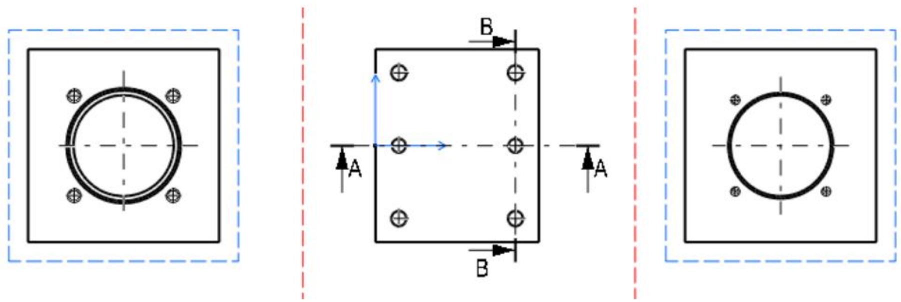


the frame of the broken section BB **> Positioning of views** >
Position Independently of Reference View > Move View Right of Section AA

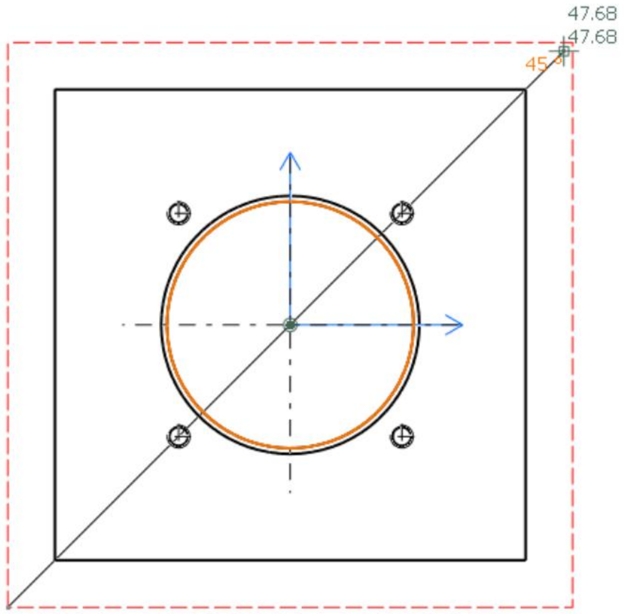
BB broken section frame **> BB section object** > **Add View name**

2x on writing **>** Erase scale, just leave cut BB > OK Reposition text if needed.

PROJECTED VIEW (behind FRONT VIEW) **>** create the right view



PROJECTED VIEW > create left view

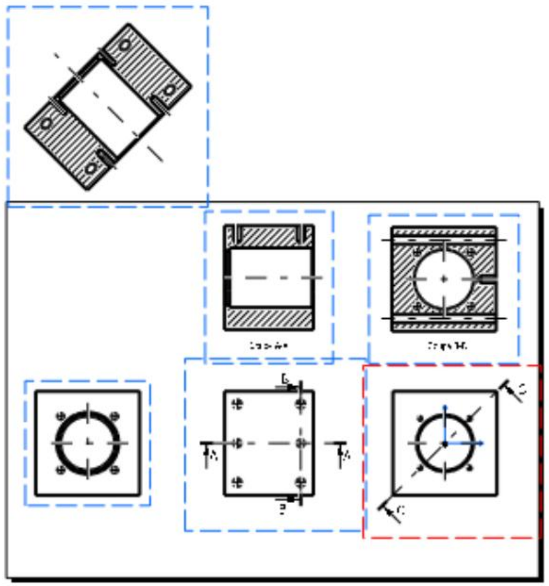


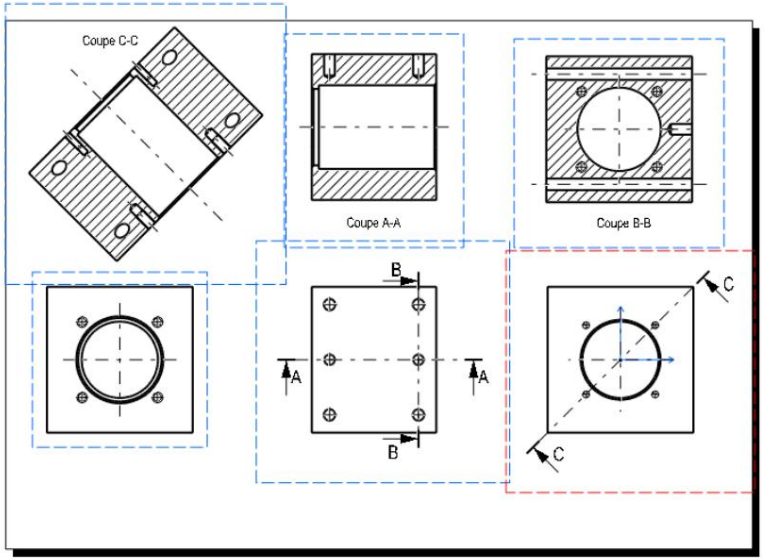
2x on the frame of the left view (thus the one on the right) > the frame turns red

BROKEN CUT > create the **CC cut > Procedure:** zoom in on one of the holes > the center > zoom out and **2x** outside the block, taking care to pass through the center of the part (indicated by the symbol) > place the section at the top left of the work plan.

CC broken section frame > Positioning views > Position independent of reference view > Move CC section in work plane.

the broken section frame CC **> Section Object CC** > **Add View Name >** Clear Scale, just leave CC Section > OK Reposition text if needed.

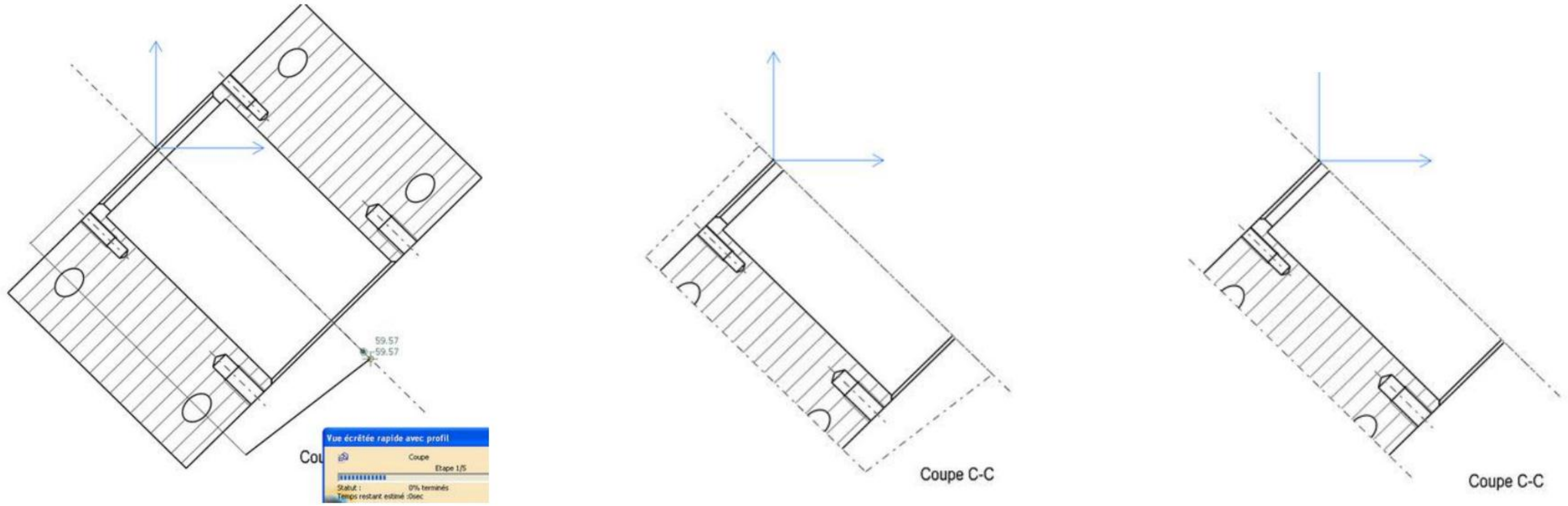




2x on CC cup frame > frame turns red

PROFILE OF THE QUICK CUT VIEW (behind **CUT VIEW) >** create a rectangle which passes through the axis of the block and surrounds it as shown in the figure below.

the 2 dotted lines perpendicular to the axis of the block > **F2** key on the keyboard (this key is used to hide the lines selected)

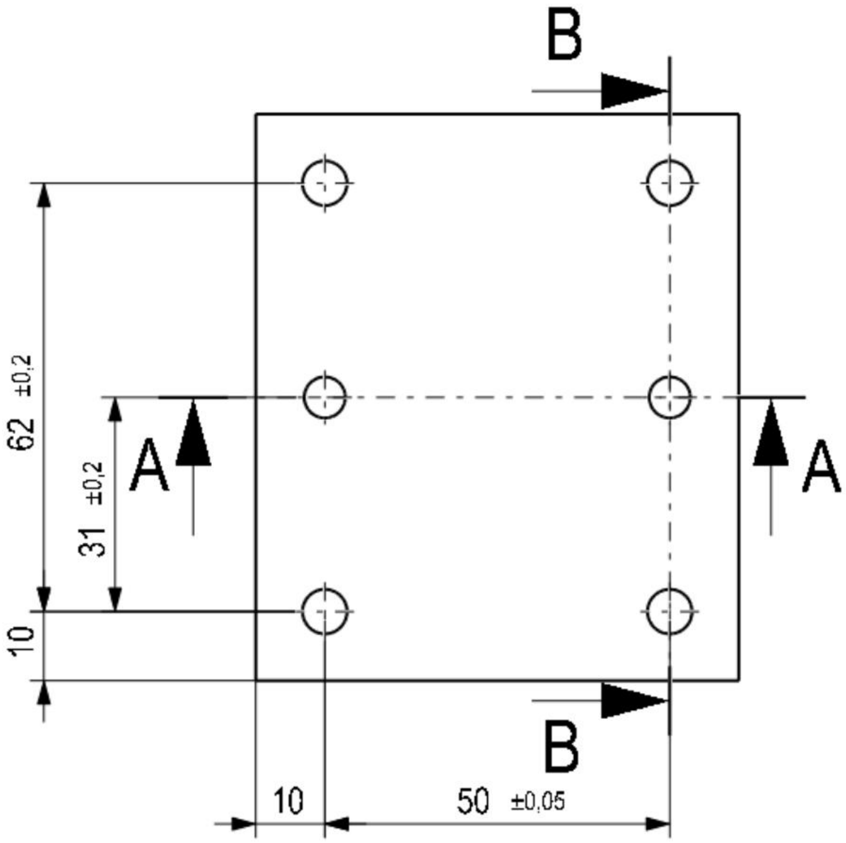


EPFL CARTRIDGE > the Epfl cartridge appears, modify the name if necessary.

Dimensions of the different views

Front view

2x on the front view frame > the frame turns red

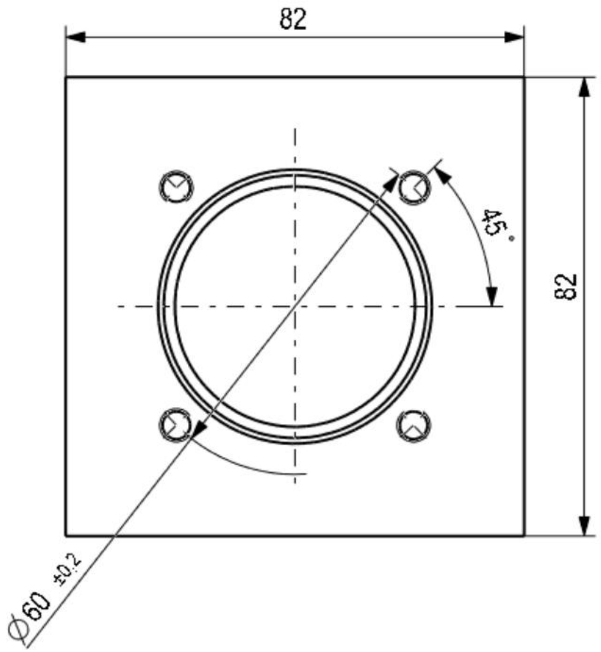


Using the **DIMENSION** tool, dimension the front view as shown in the figure.

To align dimensions (such as **10** and **50):**

on the dimension to be aligned > **Alignment** the rating of > reference > **Ok** (the 1st dimension aligns with the second)

On the left and right views, modify the center lines of the threads by following the same procedure as that followed for the DISC SHAFT part (pages 51, 52)



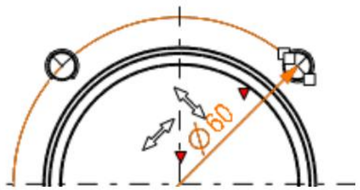
Right View

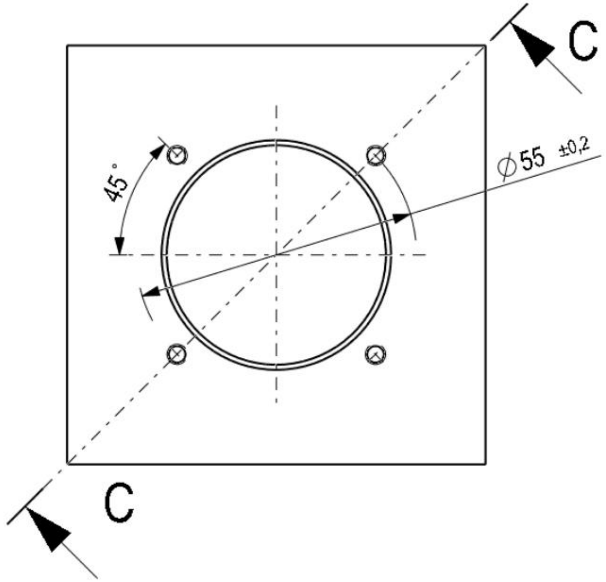
2x on the frame of the right view > the frame turns red

Using the dimension tools (DIAMETER  **DIAMETER**  **ANGLE DIVISIONS** and  **DIAMETER DIAMETER),** dimension the view on the right.

Note: To interrupt the dimension circle Ø60: before confirming the position of the dimension, when it is

still red in color, move to  white squares and obtain the desired interruption.





Left view

2x on the frame of the left view > the frame turns red

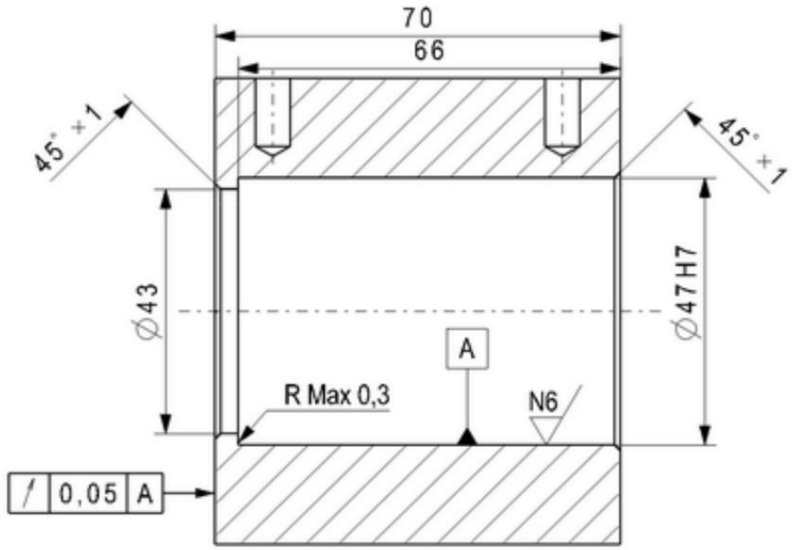
Using the dimension tools (DIAMETER  **DIAMETER**  **ANGLE DIVISIONS** and  **DIAMETER DIAMETER),** dimension the view on the right.

Section A-A

2x on the frame of the AA cup > the frame turns red

Dimension section AA using the following tools:

- QUOTATIONS
- DIAMETER QUOTATION
- ROUGHNESS SYMBOL
- REFERENCE
- GEOMETRIC TOLERANCES
- ATTACHED TEXT

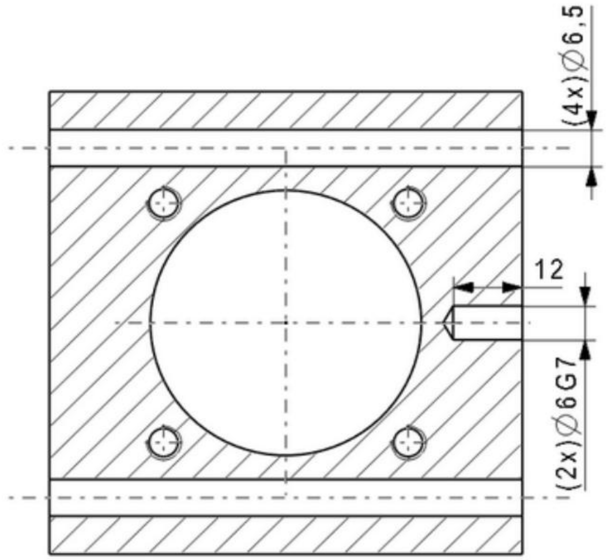


Coupe A-A

To insert the dimension of the 45°x1 chamfers

CHAMFER DIMENSIONS > Tool Palette

> select **angle x length** > the chamfer in question > Place the dimension



Coupe B-B

BB Cup

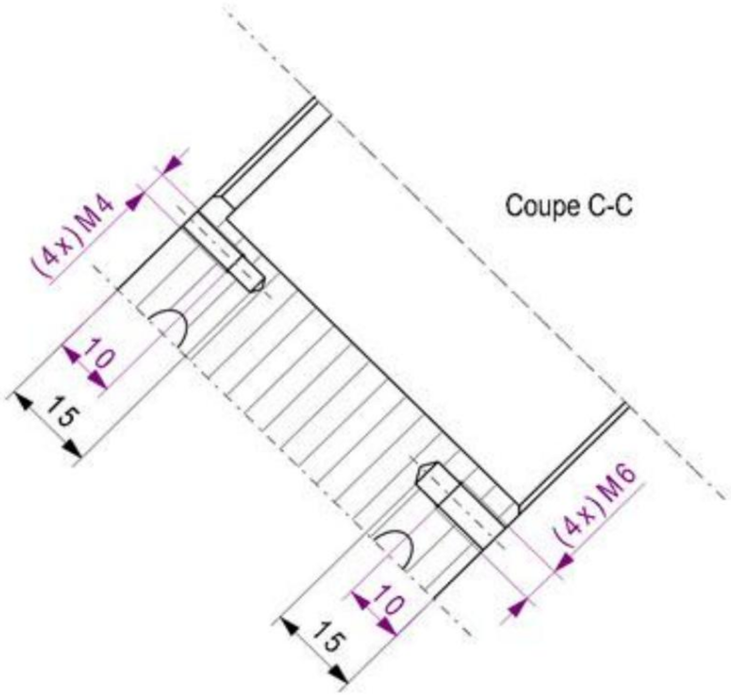
2x on BB cup frame > frame turns red

Using the DIAMETER DIAMETER and DIAMETER DIAMETER tools , dimension **the** BB cut as you learned in previous tutorials.

CC Cup

2x on CC cup frame > frame turns red

Using the **DIMENSION** tool, dimension the depth of the holes **(15).**



To insert the thread dimension

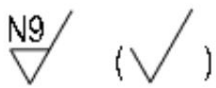
TAPPING DIMENSION > the tapping in question > Place the dimension (by default, the diameter and the depth appear with a different color)

To reverse the orientation of the hatch

the hatch > **Properties > Fill > Angle >** select **-45° > Ok**

Finally :

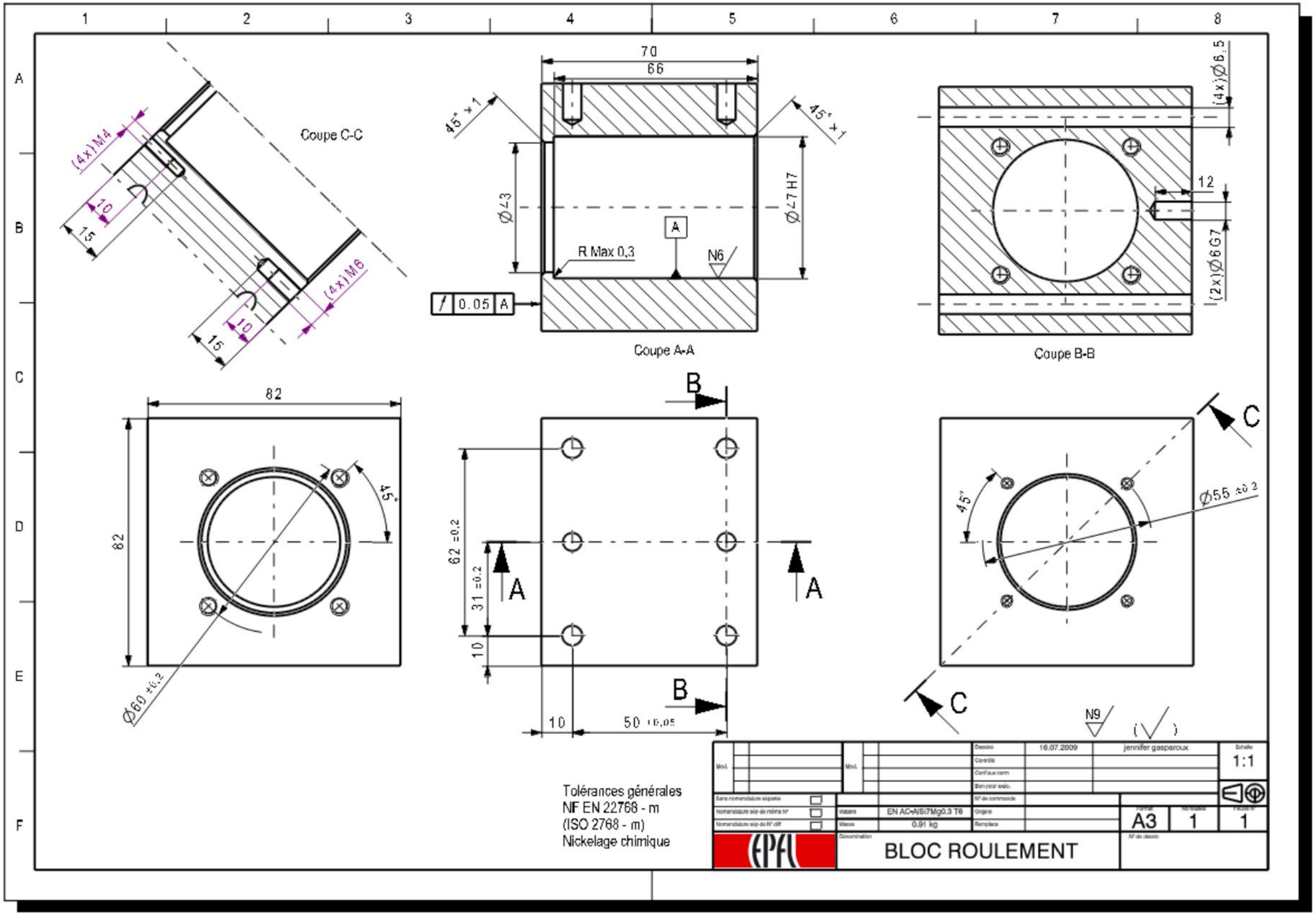
- Move views and dimensions so that there is no **overlap**
- Insert general surface finish > **ROUGHNESS SYMBOL**



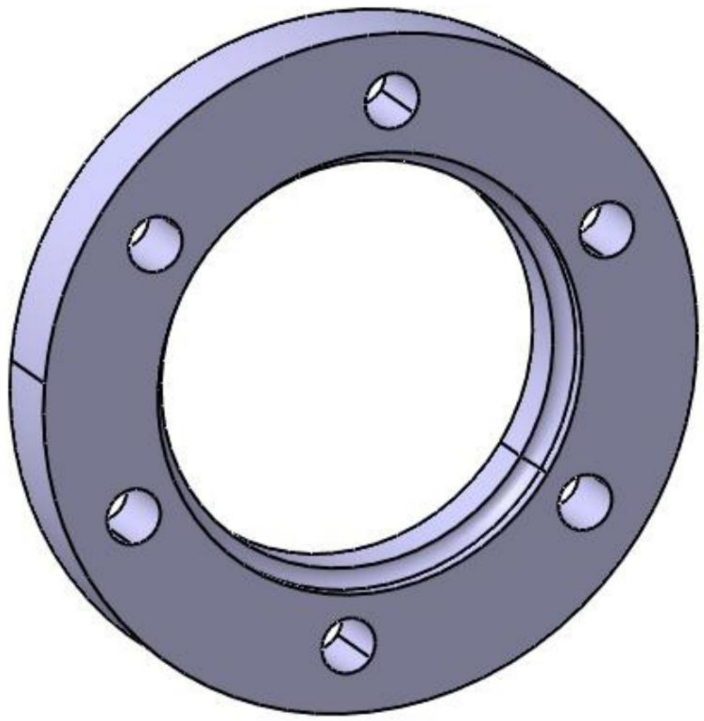
- Insert General Tolerances > **TEXT :**

Tolérances générales:
NF EN 22768 - m
(ISO 2768 - m)
Nickelage chimique

- **File** > **Save**


Coupe C-C
(4x) M4
(4x) M6
10
15
70
66
45° x 1
Ø43
Ø47 H7
R Max 0,3
A
N6
0.05 A
Coupe A-A
(4x) Ø6,5
12
(2x) Ø6 G7
Coupe B-B
82
45°
Ø60 ±0,2
B
62 ±0,2
31 ±0,2
10
50 ±0,05
A
C
Ø55 ±0,2
N9
Tolérances générales
NF EN 22768 - m
(ISO 2768 - m)
Nickelage chimique
16.07.2009
jennifer gasparoux
1:1
EN AC-AlSi7Mg0.3 T6
0.91 kg
A3
BLOC ROULEMENT

11. BASIC EXERCISE 7: SCREW RING



FINAL RESULT

Start Menu > **Part Design** > enter part name: **SCREW RING** > OK
File menu > Save > **L:catia** > file name: **VIS RING** > Save

CREATION OF THE SPAN Ø 40 mm

the **ZX** plane in the tree structure > **SKETCH**

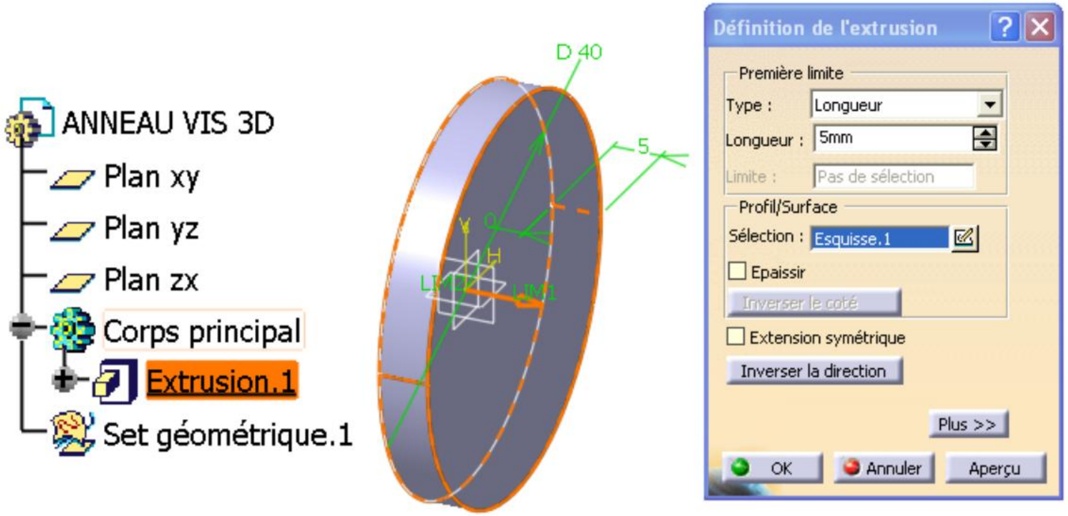
CIRCLE > the center on the origin > any diameter (circle turns red)

CONSTRAINTS > set dimension > **2x** on the diameter value > enter **40 >** OK

LEAVING THE WORKSHOP

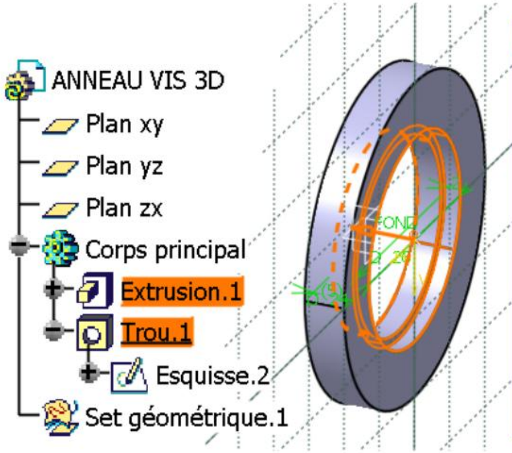
EXTRUSION > Type : Length > enter **5**

okay



CREATION OF THROUGH HOLE Ø 24, BLADE Ø 26 prof. 2mm

 HOLE

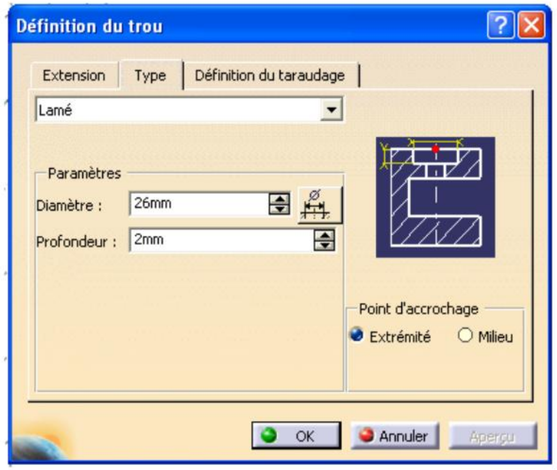


the face of the Ø40 staff (the hole appears in red)

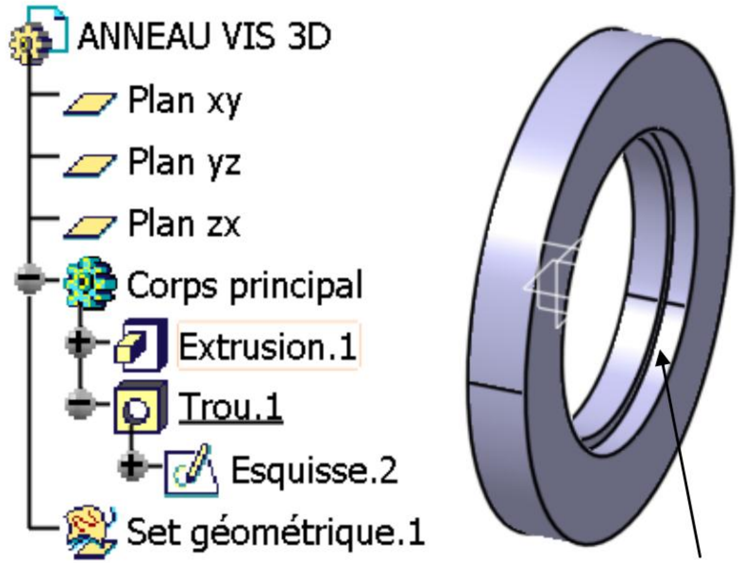
tab: **Extension:** Up to next > **Diameter : 24** mm

Type tab : Blade > **Diameter : 26** mm > **Depth : 2** mm

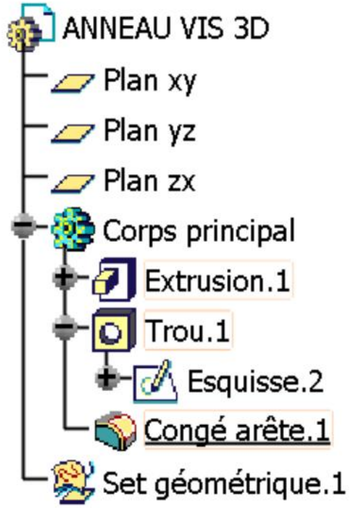
OK > OK

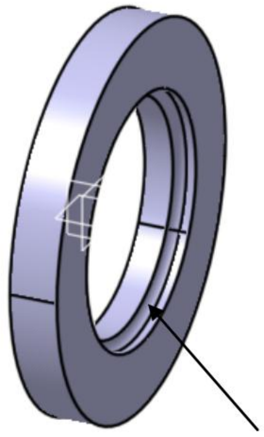


Note : if the face chosen to position a hole is circular, the hole defaults to the center. To change this location. It is then necessary to dimension the center point of the hole:



inner edge





Edge fillet = rounding

EDGE FILLET

on the inner edge (see previous drawing indication) > **Radius** > Enter **1** mm

okay

6 TAPPED HOLES M4

HOLE

the same face as for the previous hole (the hole appears in red)

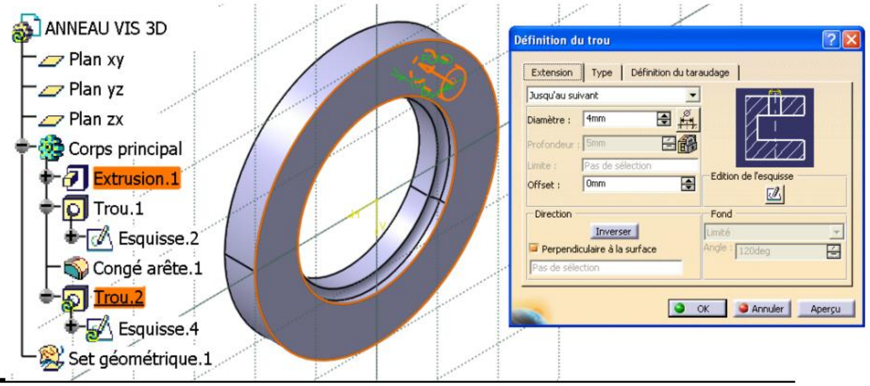
Topological operators error message > OK
(When a hole is requested, the default hole parameters correspond to those entered during the last hole, this error message appears when the dimensions of the hole "overflow" the part)

on the arrows and move the hole at 2 o'clock, so that it rests on the face of the staff.

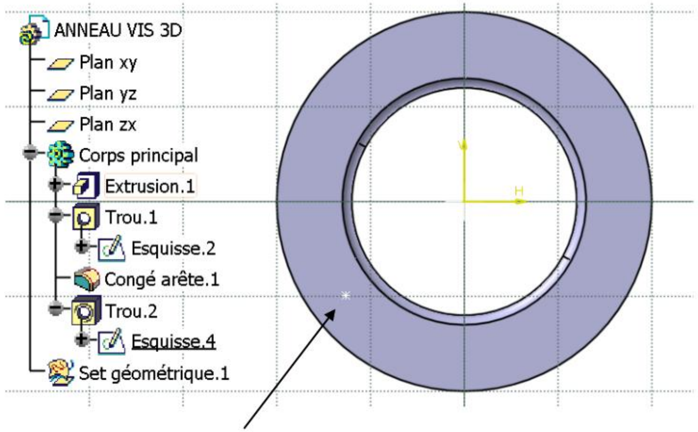
Extension tab : Up to next

> **Diameter :** 4mm

Type tab : Simple



tab: **Thread definition >** activate: **Tapped** > **Type :** Metric Coarse Pitch > **Ref. thread :** M4 > **Teacher. thread :** 5 mm > activate: **Not straight** >

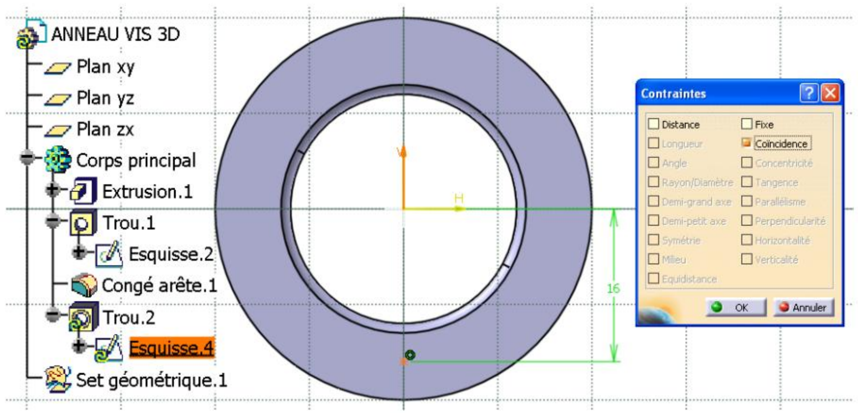


Extension tab > **Editing the sketch**

CONSTRAINTS > the horizontal axis H and on the point > set the dimension > **2x** on the dimension value > enter **16 >** OK

on the vertical axis > hold down the **Ctrl** key and > on point (two selected items turn red) release the **Ctrl** key

SELECTED CONSTRAINTS IN A DIALOG BOX



Coincidence > OK

LEAVING THE WORKSHOP

okay

CIRCULAR REPEAT

Hole.2 (= last hole created) in the tree structure (the hole turns red)

CIRCULAR REPEAT

tab: **Axial reference** >

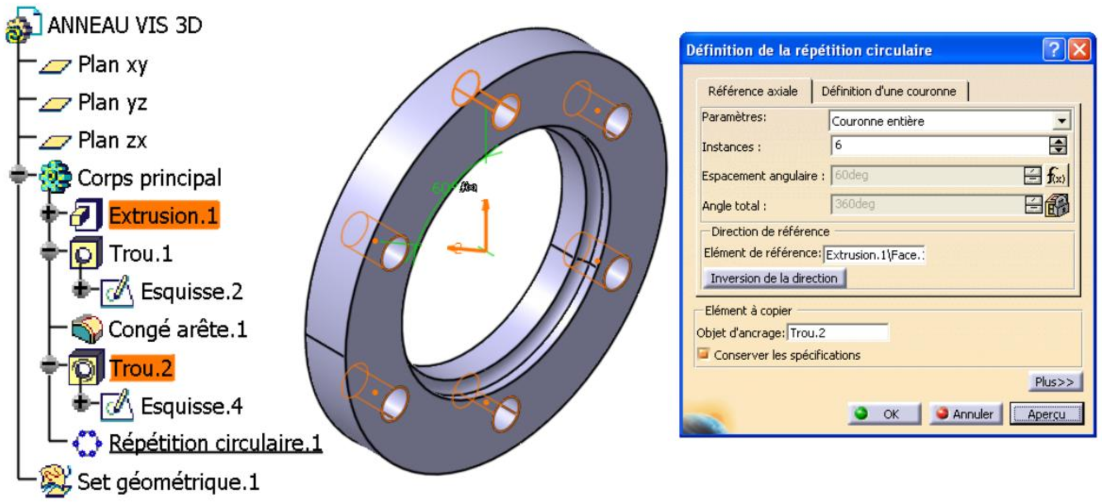
Settings: Full Crown >

Instances : enter **6** >

Reference direction : the face of the ring Ø40 >

enable: **Keep specifications**

okay



APPLICATION OF MATERIALS

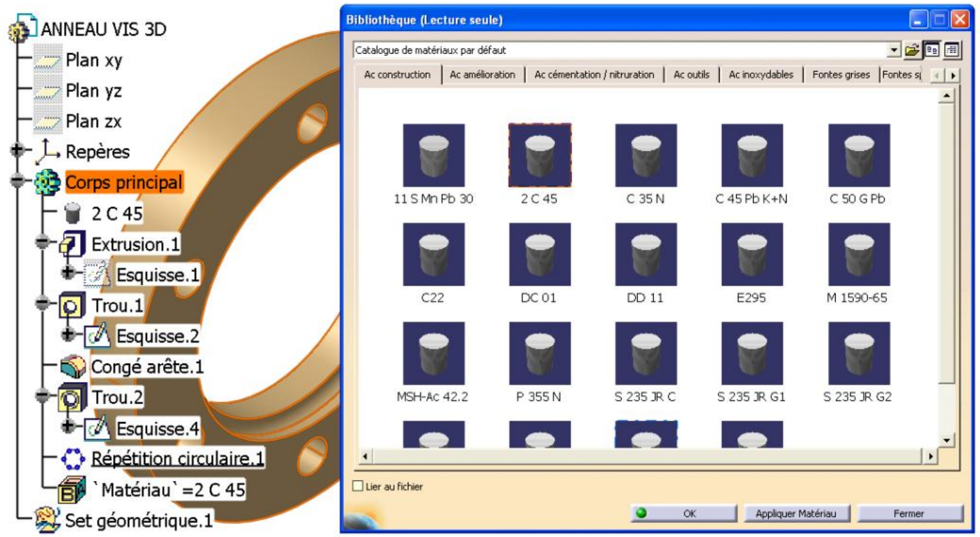
APPLICATION OF MATERIALS

Ac construction tab > 2 C 45

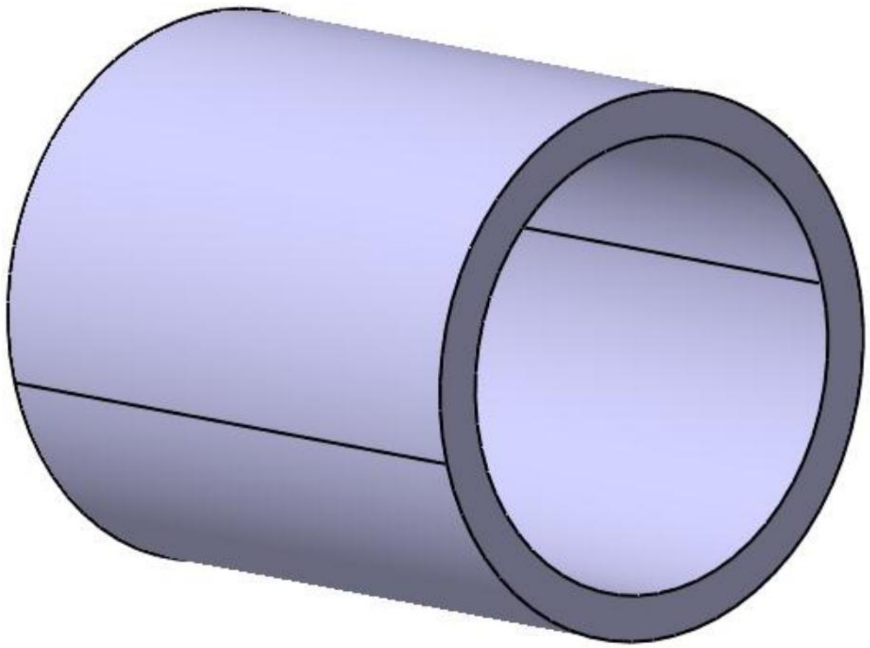
in the **Main Body** tree view (part outline turns red)

Apply Material > OK

File > **Save**



12. BASIC EXERCISE #8: CYLINDRICAL WEDGE



FINAL RESULT

Start menu > **Part Design** > enter the name of the part: **CYLINDRICAL SHIM** > OK
File menu > **Save** > **L:catia** > file name: **CYLINDRICAL WEDGE** > Save

SPAN Ø 30 mm

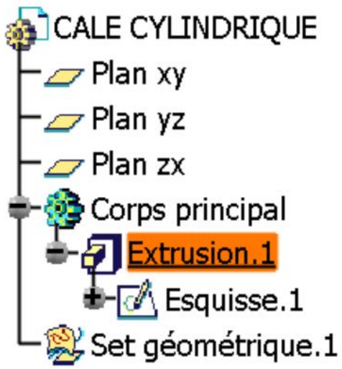
the **ZX** plane in the tree structure > **SKETCH**

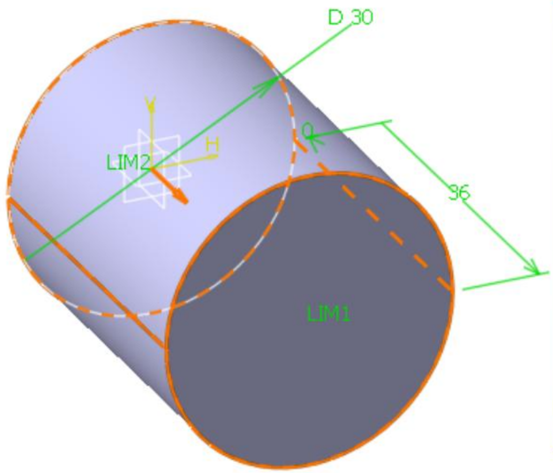
CIRCLE > the center on the origin > any diameter (the circle turns red)

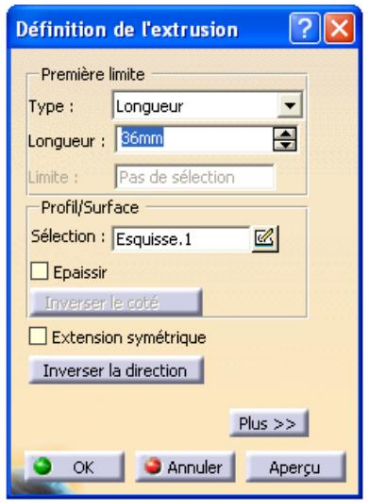
CONSTRAINTS > set dimension > **2x** on the diameter value > enter **30 > OK**

LEAVING THE WORKSHOP

EXTRUSION > Type : Length > enter **36** > **OK**





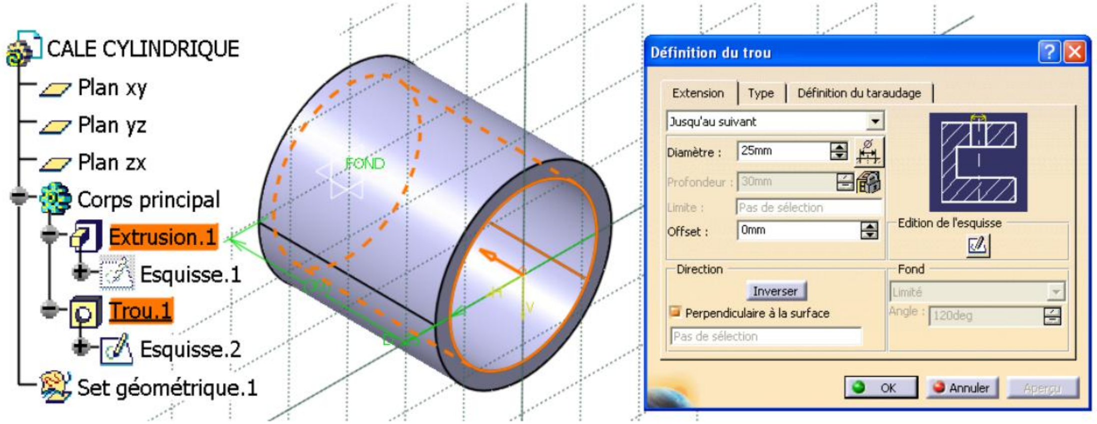


HOLE Ø 30 mm

HOLE

a flat face of the cylinder (the hole appears in red)

Extension tab : Up to next > **Diameter :** 25mm >
OK > **OK**



APPLICATION OF MATERIALS

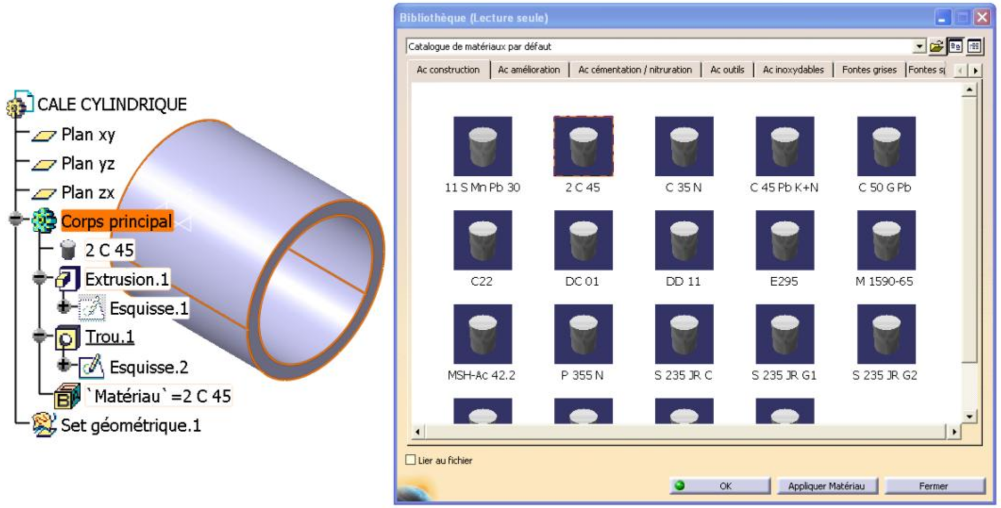
APPLICATION OF MATERIALS

Ac construction tab >

2 C 45

in the **Main Body** tree view (part outline turns red)

Apply Material > OK



File > **Save**

Note: The Cylindrical Shim can also be constructed in one step, using the **Thicken** option of the Extrude tool .

Do this exercise by opening a new file **(Start** > **Part Design)** which you will call **CYLINDRICAL WEDGE 2.**

CYLINDRICAL WEDGE 2

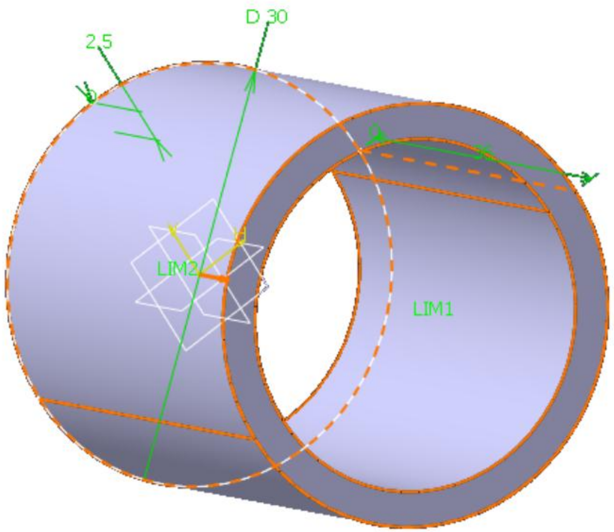
the **ZX** plane in the tree structure > **SKETCH**

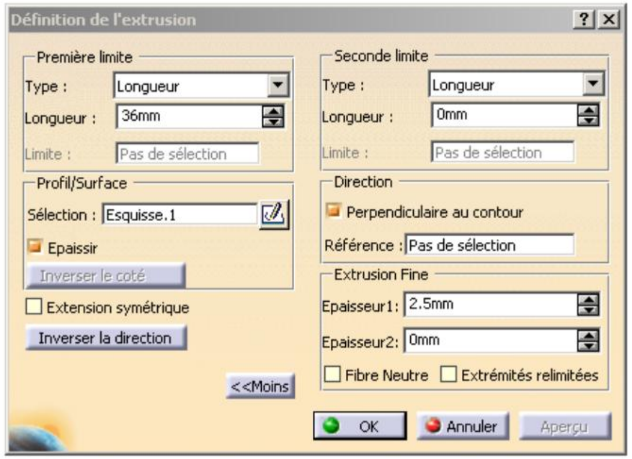
CIRCLE > the center on the origin >
any diameter (circle turns red)

CONSTRAINTS > set dimension > **2x**
on the diameter value > enter **30**
> okay

LEAVING THE WORKSHOP

EXTRUSION > Type : Length > enter **36** > activate the **Thicken tag >** the window expands > Thin **Extrusion > Thickness 1 >** enter **2.5mm > OK**

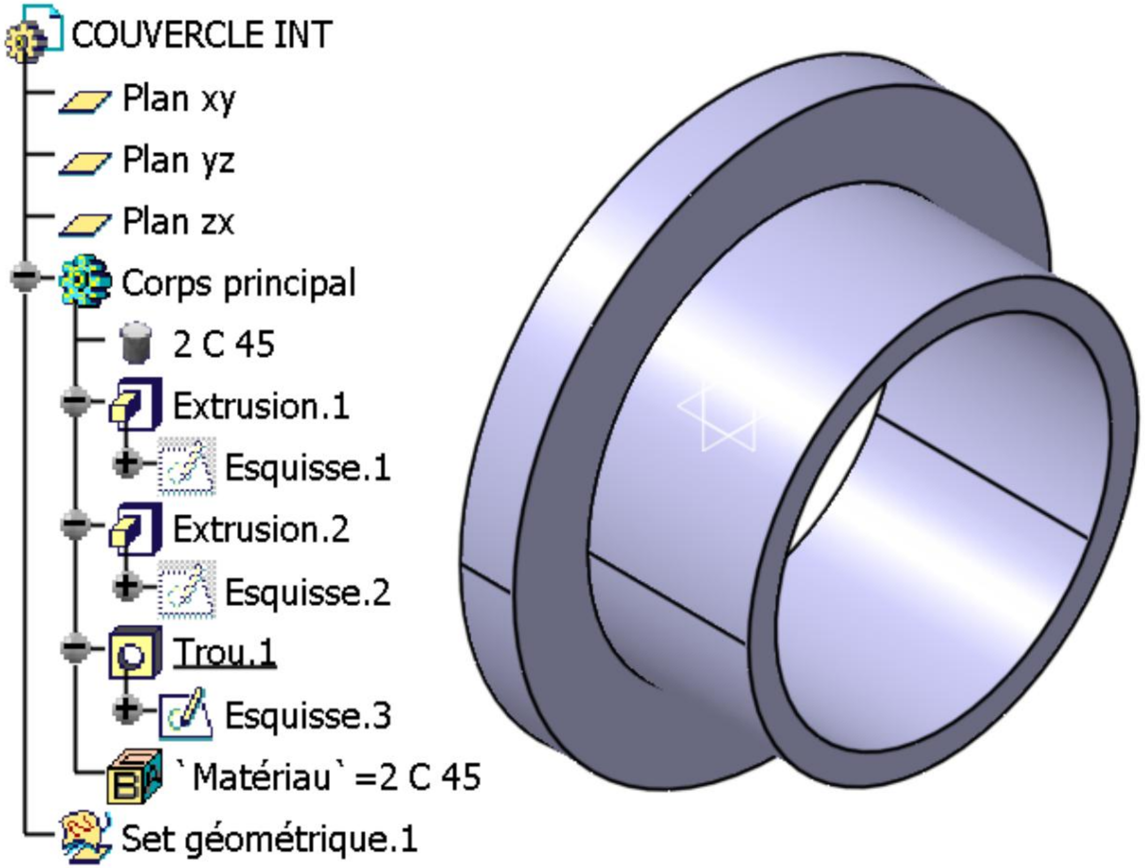




File > **Save**

Note: thickness 1 corresponds to inward thickening, thickness 2 corresponds to outward thickening.

13. BASIC EXERCISE 9: INT LID



FINAL RESULT

Start menu > **Part Design** > enter the name of the part: **COUVERCLE INT** > **OK**
File menu **> Save** > **L: catia** > file name : COUVERCLE **INT** > **Save**

SPAN Ø 40mm LENGTH 4mm

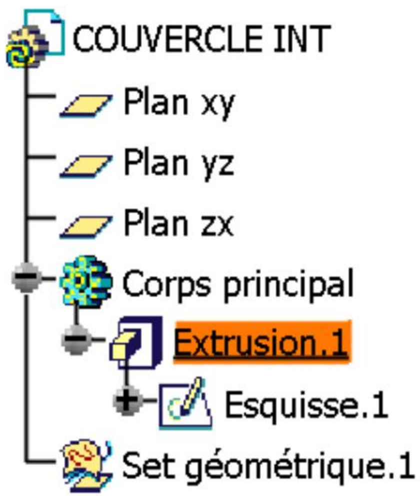
the **ZX** plane in the tree structure > **SKETCH**

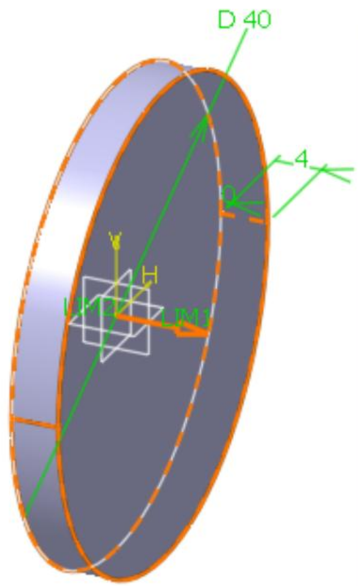
CIRCLE > the center on the origin > any diameter (the circle turns red)

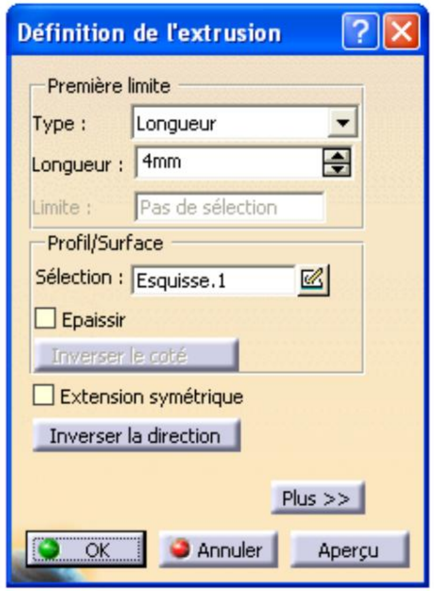
CONSTRAINTS > set dimension > **2x** on the diameter value > enter **40 > OK**

LEAVING THE WORKSHOP

EXTRUSION > Type : Length > enter **4** > **OK**







SPAN Ø 29mm LENGTH 12mm

a flat face of the cylinder

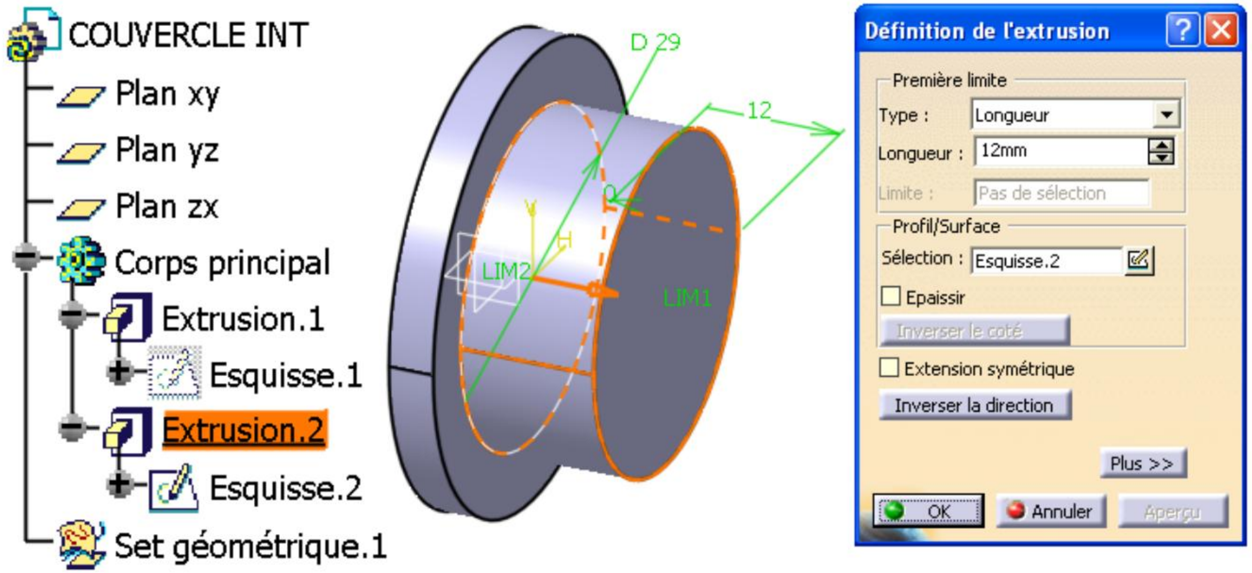
SKETCH

CIRCLE > the center on the origin > any diameter (the circle turns red)

CONSTRAINTS > set the dimension > **2x** on the diameter value > enter **29 > OK**

LEAVING THE WORKSHOP

EXTRUSION > Type: Length > enter **12** > **OK**

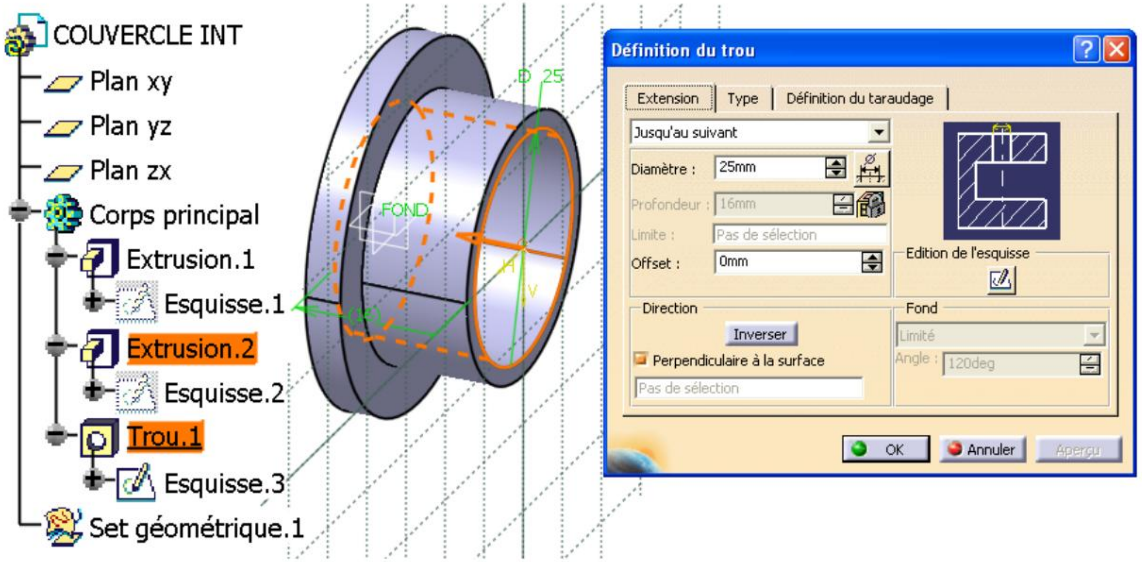


HOLE Ø 25mm

HOLE

the face of the **Ø29 staff** (the hole appears in red)

Extension tab : Up to next > **Diameter :** 25mm > **OK** > **OK**



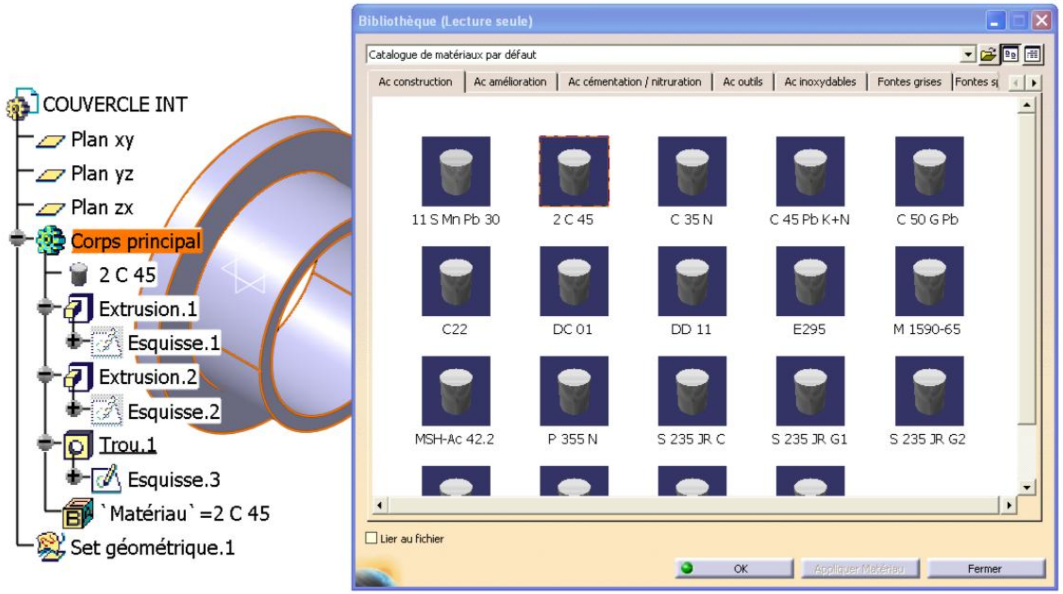
APPLICATION OF MATERIALS

APPLICATION OF MATERIALS

Ac construction tab > **2 C 45**

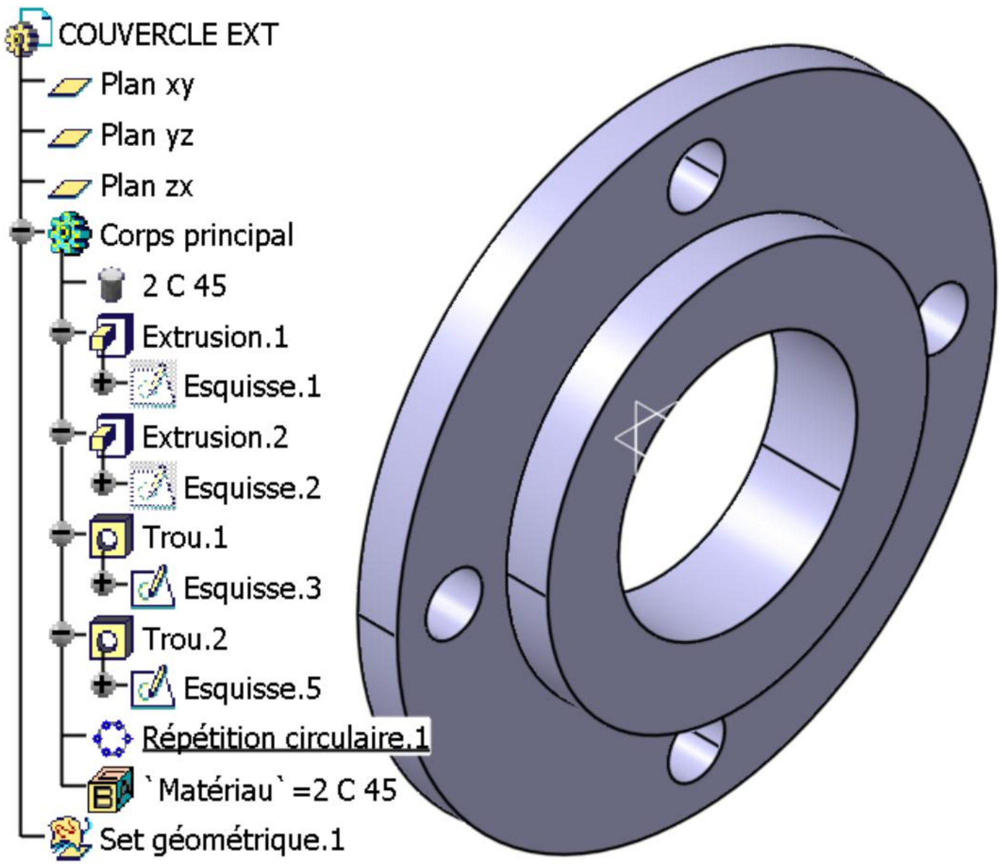
in the **Main Body** tree view (part outline turns red)

Apply Material > **OK**



File > Save

14. BASIC EXERCISE 10: EXT LID



FINAL RESULT

Start menu > **Part Design** > enter the name of the part: **COUVERCLE EXT** > **OK**
File menu **> Save** > **L: Catia** > file name: **COUVERCLE EXT** > **Save**

REACH Ø 74 mm LENGTH 4.9mm

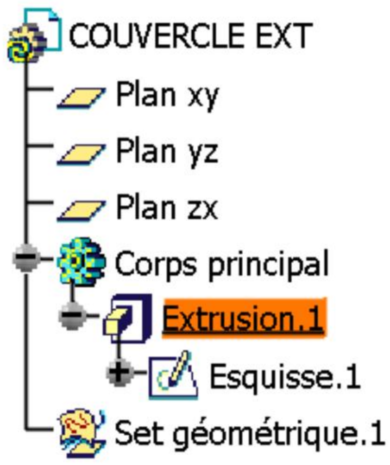
the **ZX** plane in the tree structure > **SKETCH**

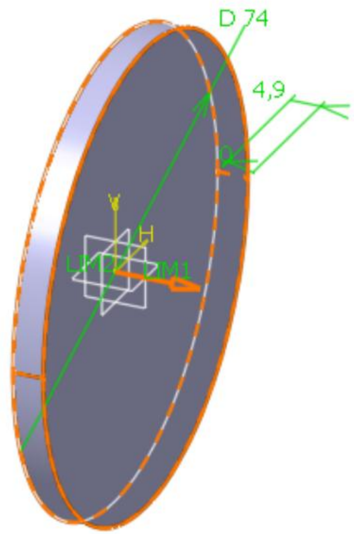
CIRCLE > the center on the origin > any diameter (the circle turns red)

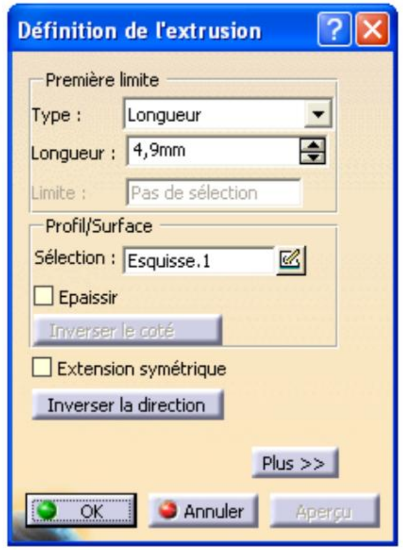
CONSTRAINTS > set dimension > 2x on the diameter value > enter 74 > **OK**

LEAVING THE WORKSHOP

EXTRUSION > Type: Length > enter **4.9** > **OK**







REACH Ø 47mm LENGTH 5.1mm

a flat face of the cylinder

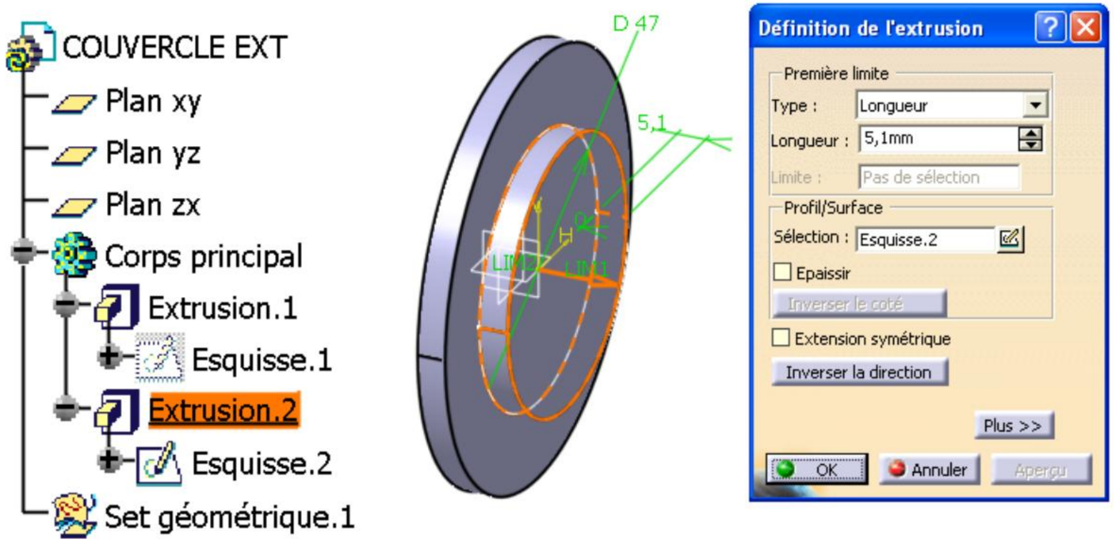
SKETCH

CIRCLE > the center on the origin > any diameter (the circle turns red)

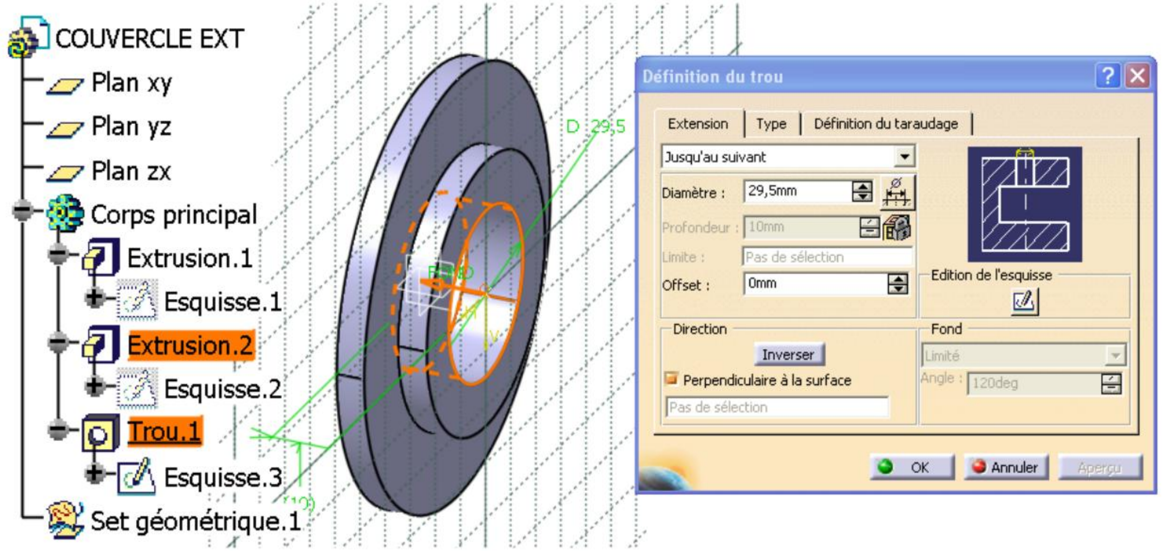
CONSTRAINTS > set the dimension > **2x** on the diameter value > enter **47 > OK**

LEAVING THE WORKSHOP

EXTRUSION > Type: Length > enter **5.1** (reverse direction if necessary) > **OK**



HOLE Ø 29.5mm

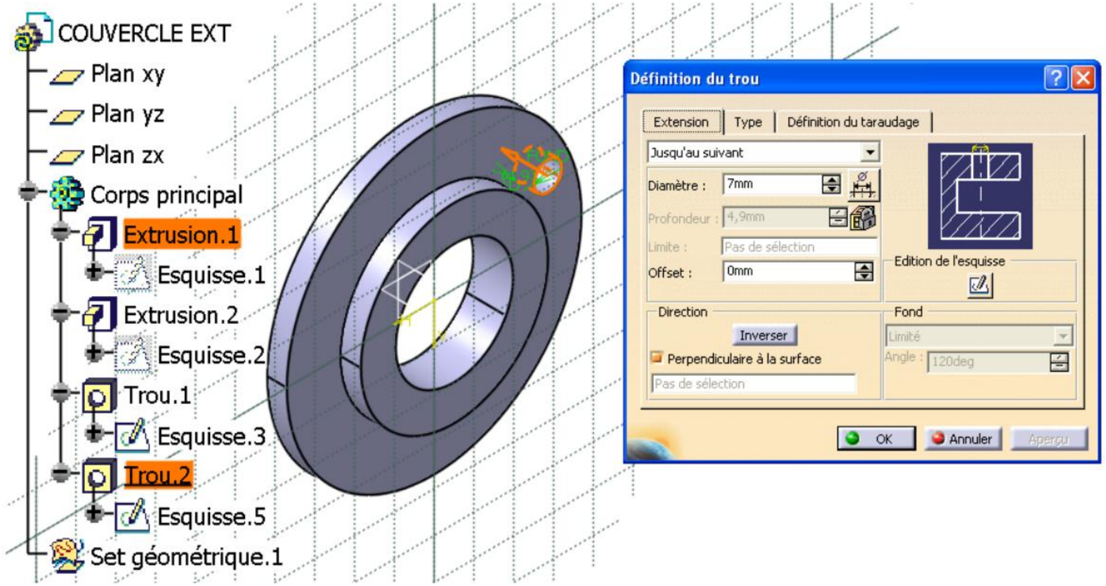


HOLE

the face of the **Ø47 staff** (the hole appears in red)

Extension tab : Up to next > **Diameter : 29.5mm** > **OK > OK**

4 THROUGH HOLES Ø 7mm

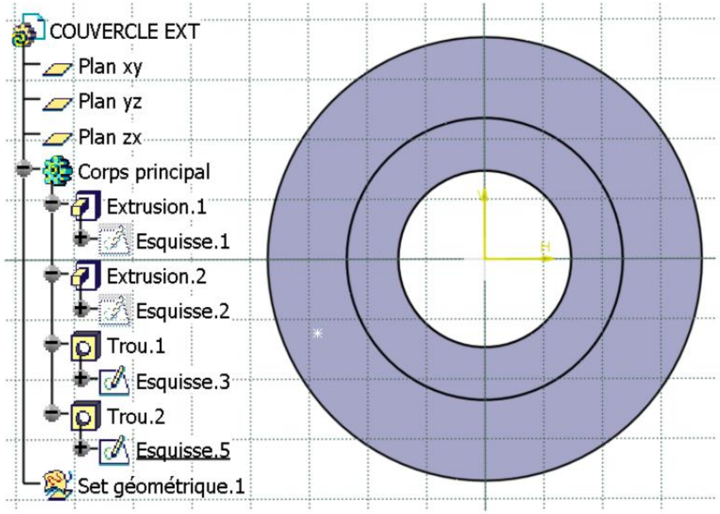


HOLE

the face of the **Ø 74 staff** (the hole appears in red)

Topological error message : **OK**

on the arrows linked to the hole and move the hole to 2 o'clock (on the staff face) ÿ **Extension:** Up to the next > **Diameter : 7** mm ÿ

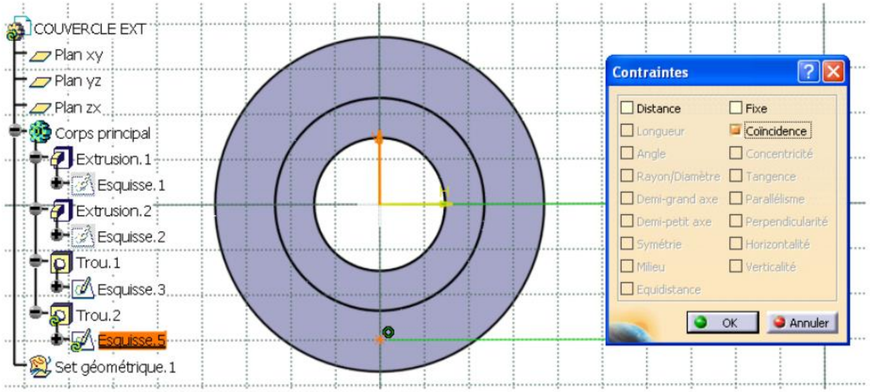


Type tab : **Simple** >

Extension tab > **Editing the sketch**

CONSTRAINTS > the horizontal axis **H** and on the point > put the odds > 2x on the dimension value > enter **30** > **OK**

on the vertical axis > keep the **Ctrl** key pressed and two on the point (the selected elements turn red) > release the **Ctrl** key

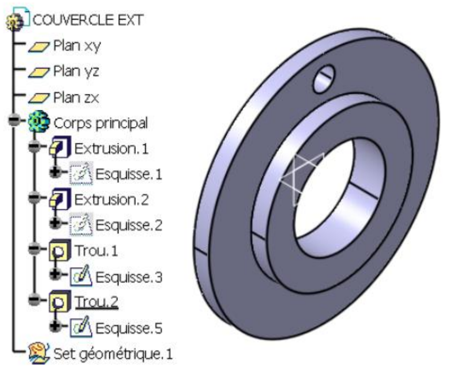


SELECTED CONSTRAINTS IN A DIALOG BOX

Coincidence > **OK**

LEAVING THE WORKSHOP

okay

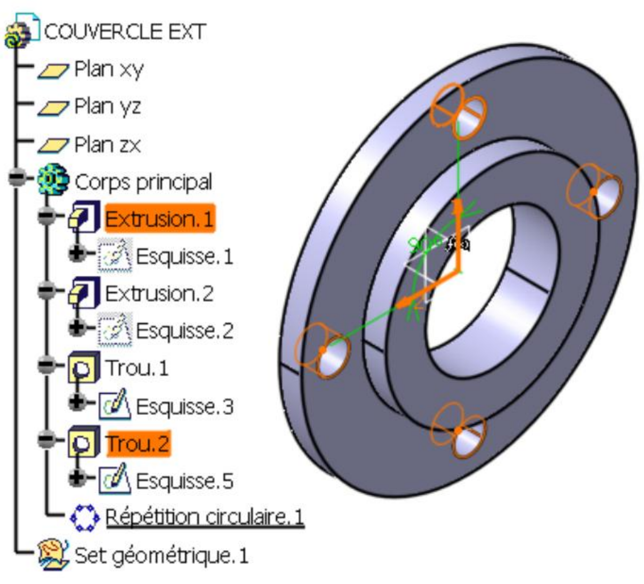


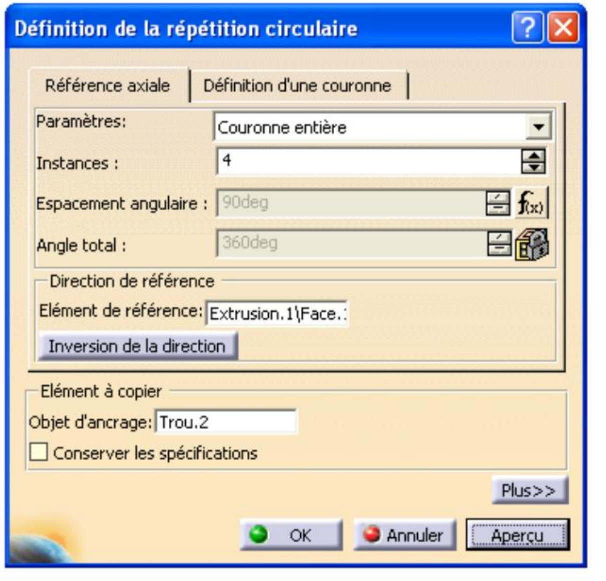
CIRCULAR REPEAT

Hole.2 (= last hole created) in the tree structure (the hole turns red)

CIRCULAR REPEAT

Axial reference tab : **> Parameters:** Entire crown > **Instances :** enter **4** > **Reference direction :** worn Ø 74 > the face of the activate: **Keep specifications** > **OK**





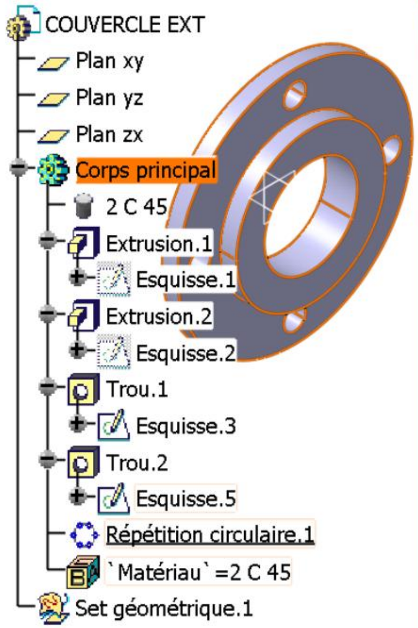
APPLICATION OF MATERIALS

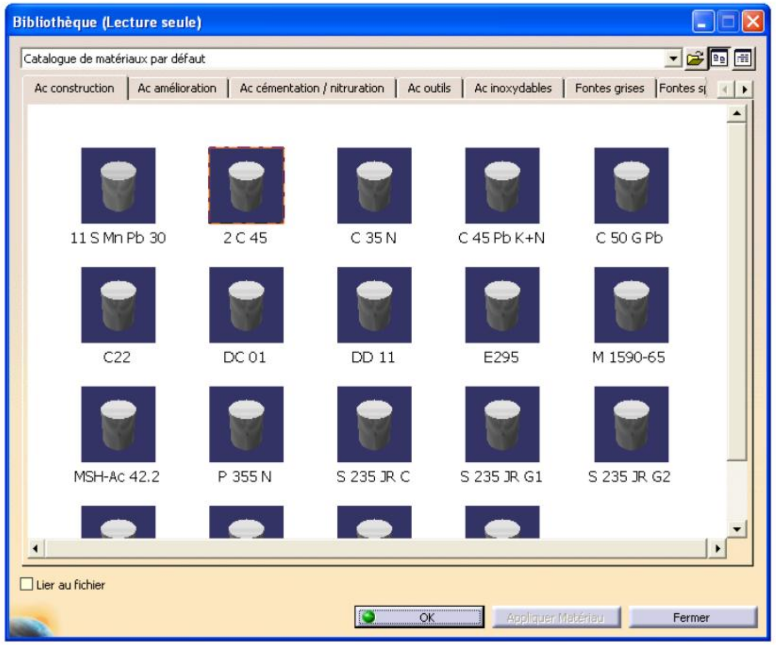
APPLICATION OF MATERIALS

Ac construction tab > **2C 45**

in the **Main Body** tree view (part outline turns red)

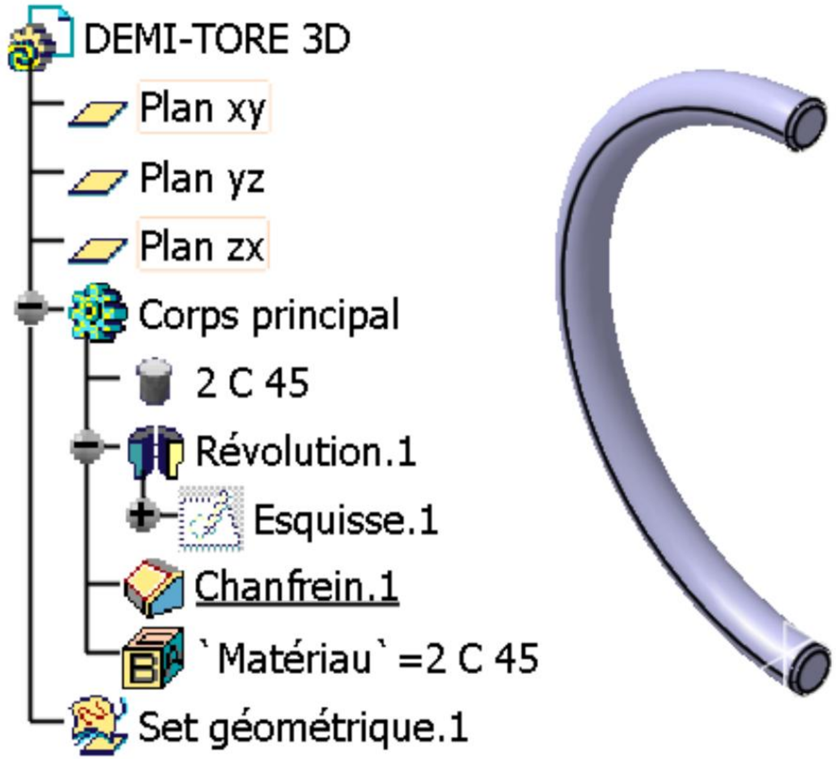
Apply Material > **OK**





File > Save

15. BASIC EXERCISE 11: HALF-TORUS



FINAL RESULT

Start menu > **Part Design** > enter the name of the part: **HALF TORE** > **OK**
File menu **> Save** > **L: Catia** > file name: **HALF TORE** > **Save**

HALF TORUS Ø2mm

the **ZX** plane in the tree structure > **SKETCH**

CIRCLE > the center on the origin > any diameter (the circle turns red)

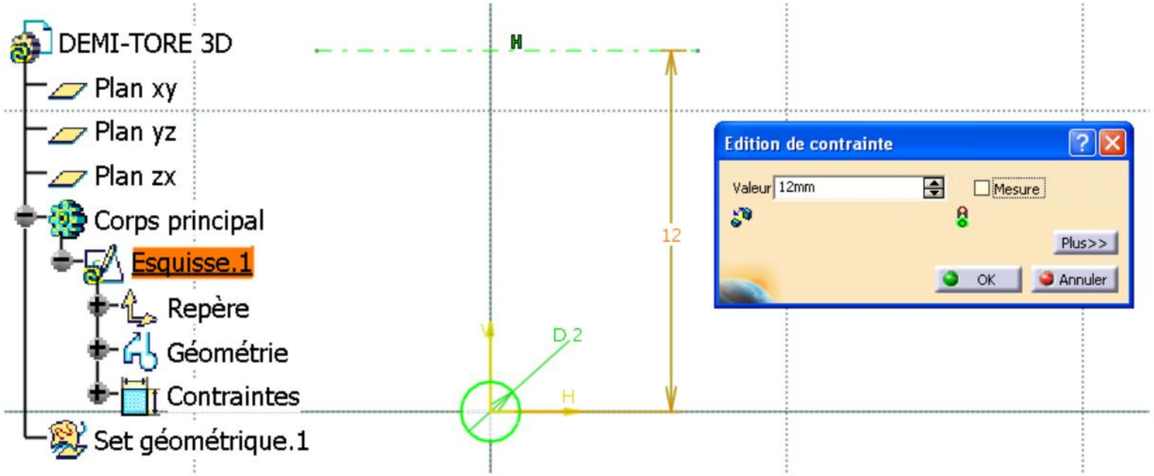
CONSTRAINTS > set dimension > **2x** on the diameter value > enter **2 >** OK

AXIS > draw a horizontal axis line above the circle.

beside the view

CONSTRAINTS > the **horizontal** axis H

and the axis line > set the dimension > **2x** on the value of the dimension > enter **12 > OK**



LEAVING THE WORKSHOP

REVOLUTION

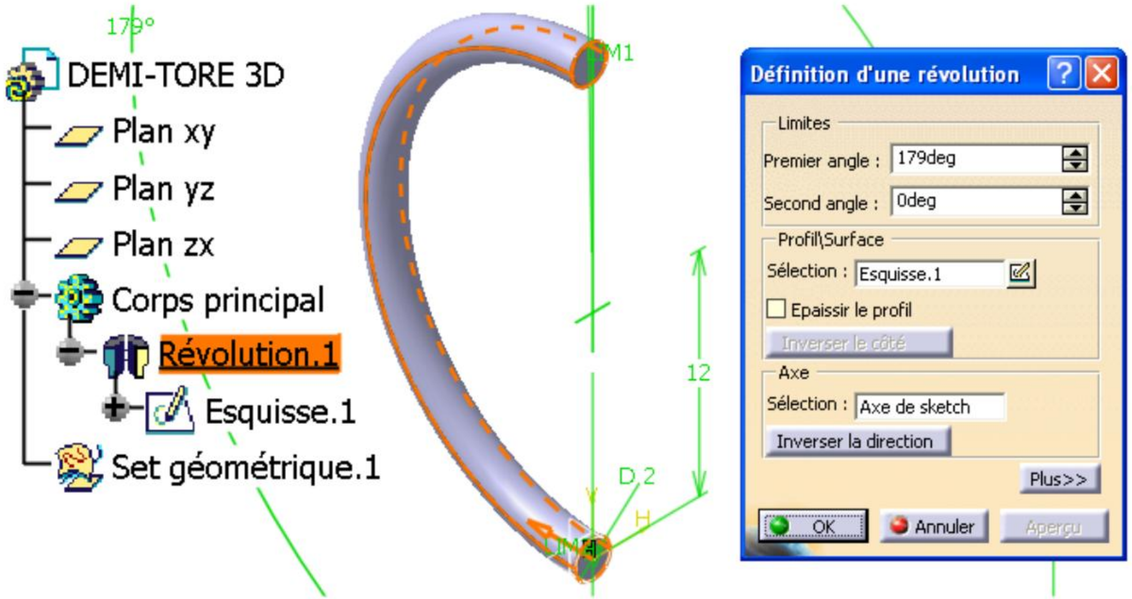
First angle: 179 deg >

Second angle: 0 deg >

Profile/Surface > Sketch.1

Axis > Sketch Axis
(Normally the axis of the sketch is selected automatically)

>OK



CHAMFER

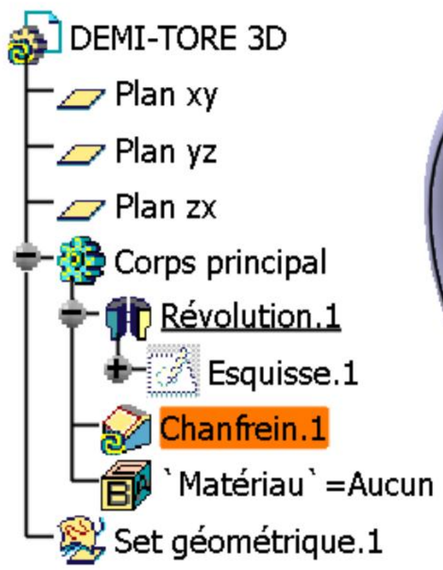
Mode: Length1/Angle >

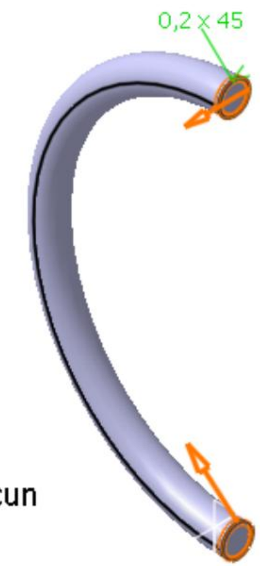
Length 1: enter **0.2 >**

Angle : between **45 >**

Objects to chamfer: the 2 ends of the half torus (Ctrl key pressed to select 2 elements)

> **okay**





APPLICATION OF MATERIALS

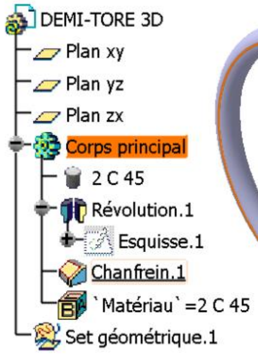
APPLICATION OF MATERIALS

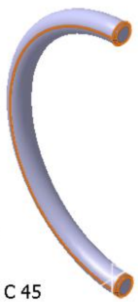
Ac construction tab > 2 C 45

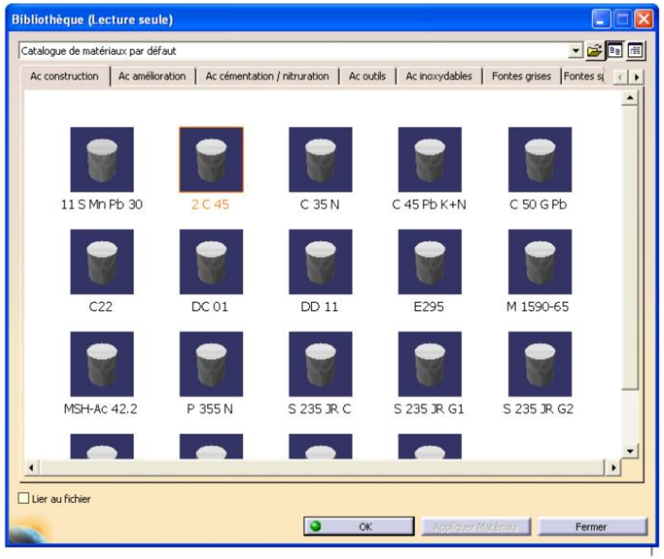
in the **Main Body** tree view (part outline turns red)

Apply Material > **OK**

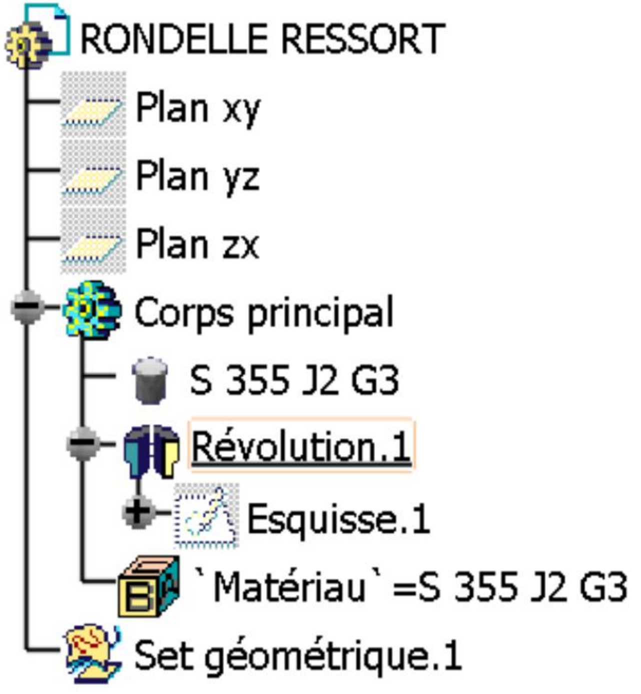
File > Save

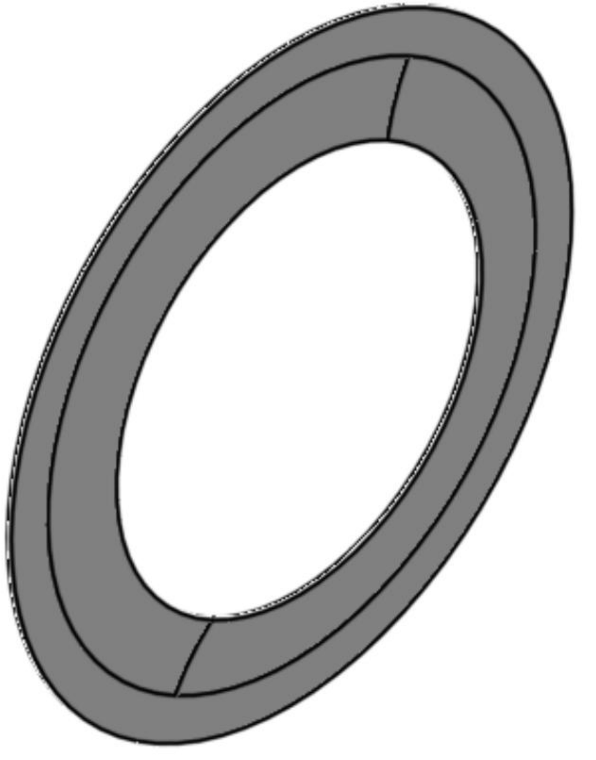






16. BASIC DRILL 12: SPRING WASHER





FINAL RESULT

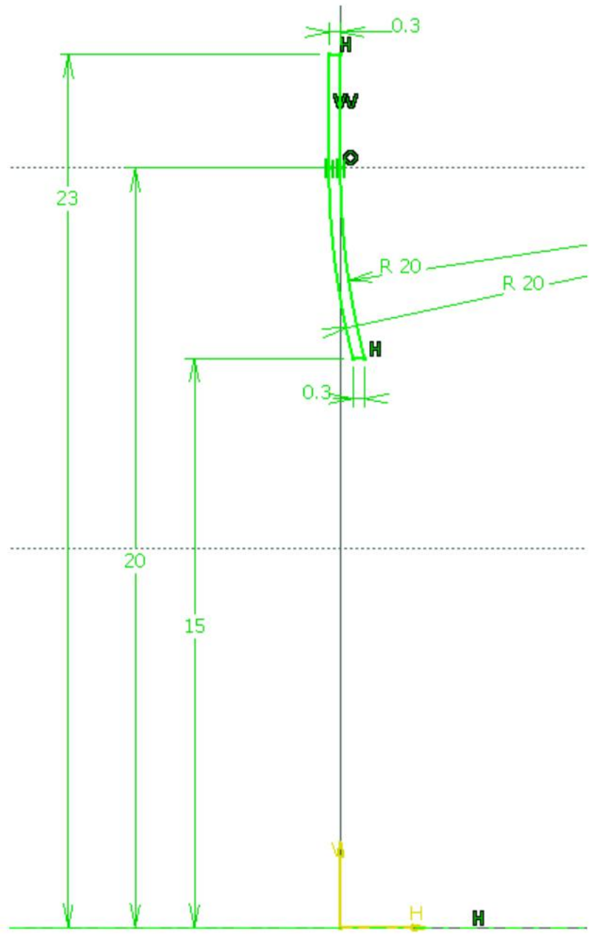
Start menu > **Part Design** > enter the name of the part: **SPRING WASHER** > **OK**
File menu **> Save** > **L: Catia** > file name: **SPRING WASHER** > **Save**

the **ZX** plane in the tree structure > **SKETCH**

AXIS > draw the horizontal axis line, passing through the origin.

Using the **construction tools,** draw the profile of the **spring washer** respecting the dimensions given in the image on the right

LEAVING THE WORKSHOP

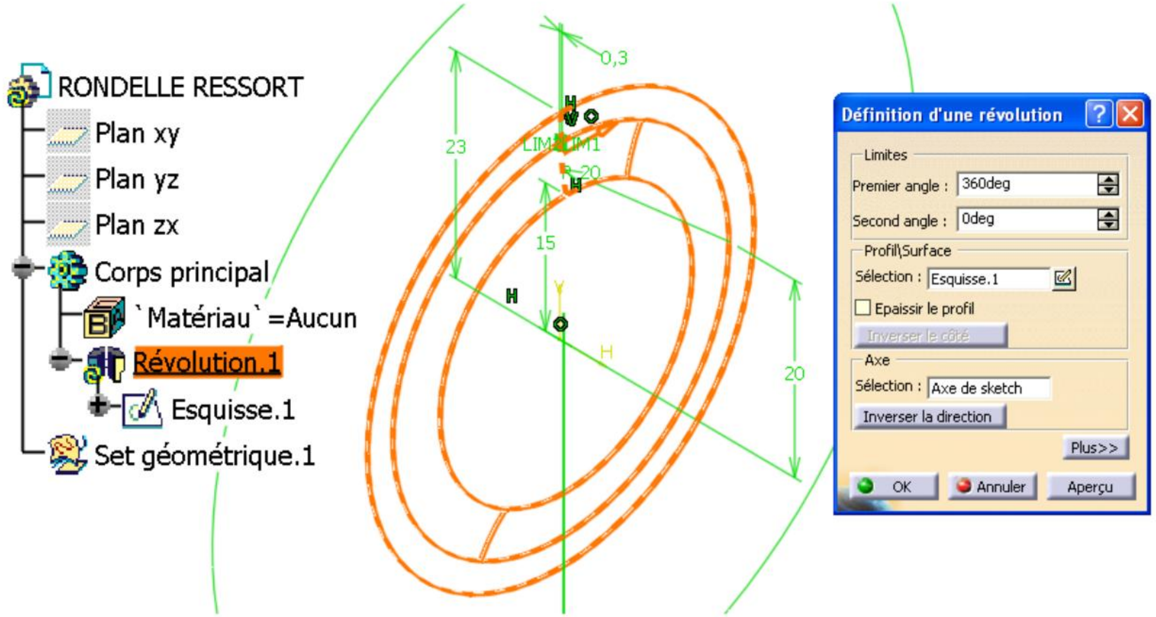


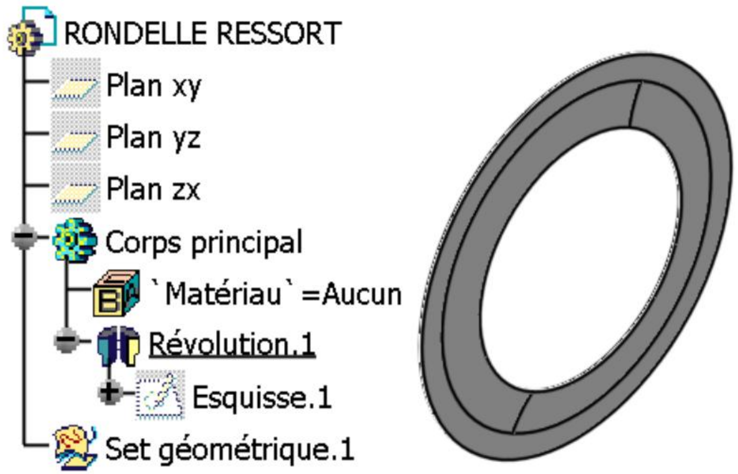
REVOLUTION

First angle: 360 deg >

Second angle: 0 deg >

Select : Sketch Axis > **OK**



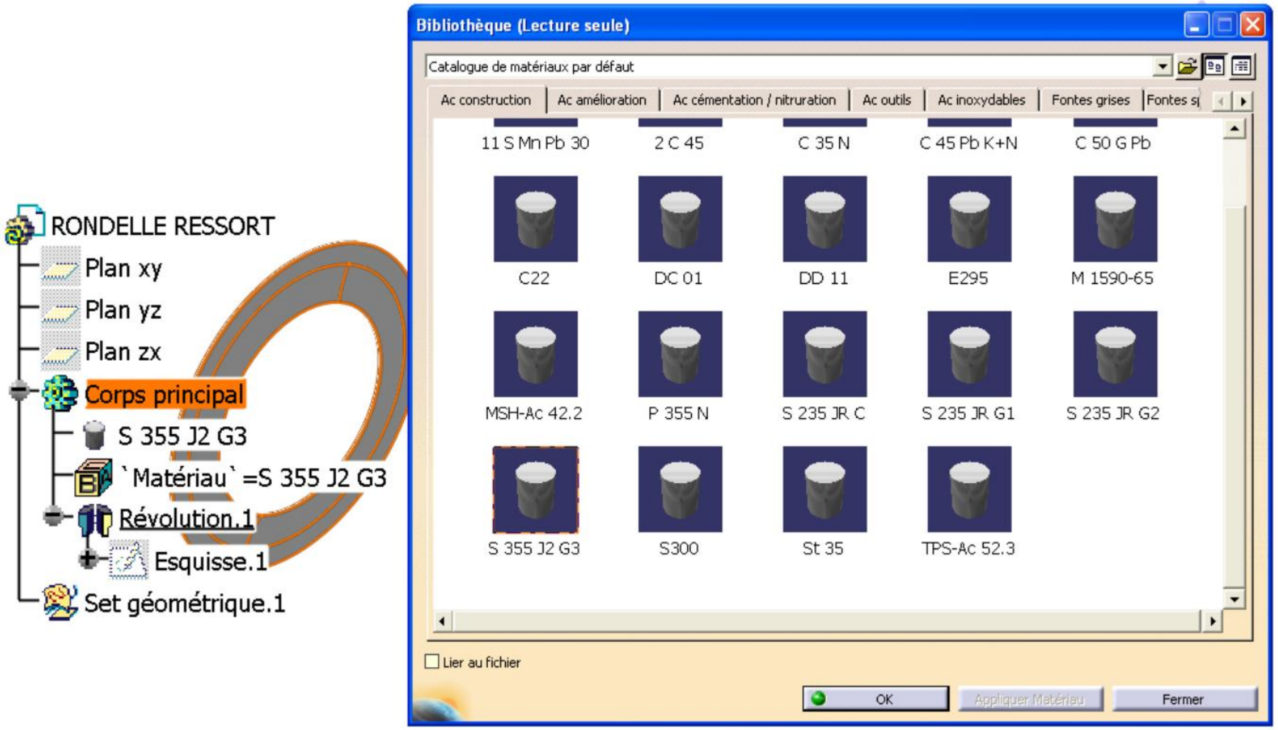


APPLICATION OF MATERIALS

Ac construction tab > **S 355 J2 G3**

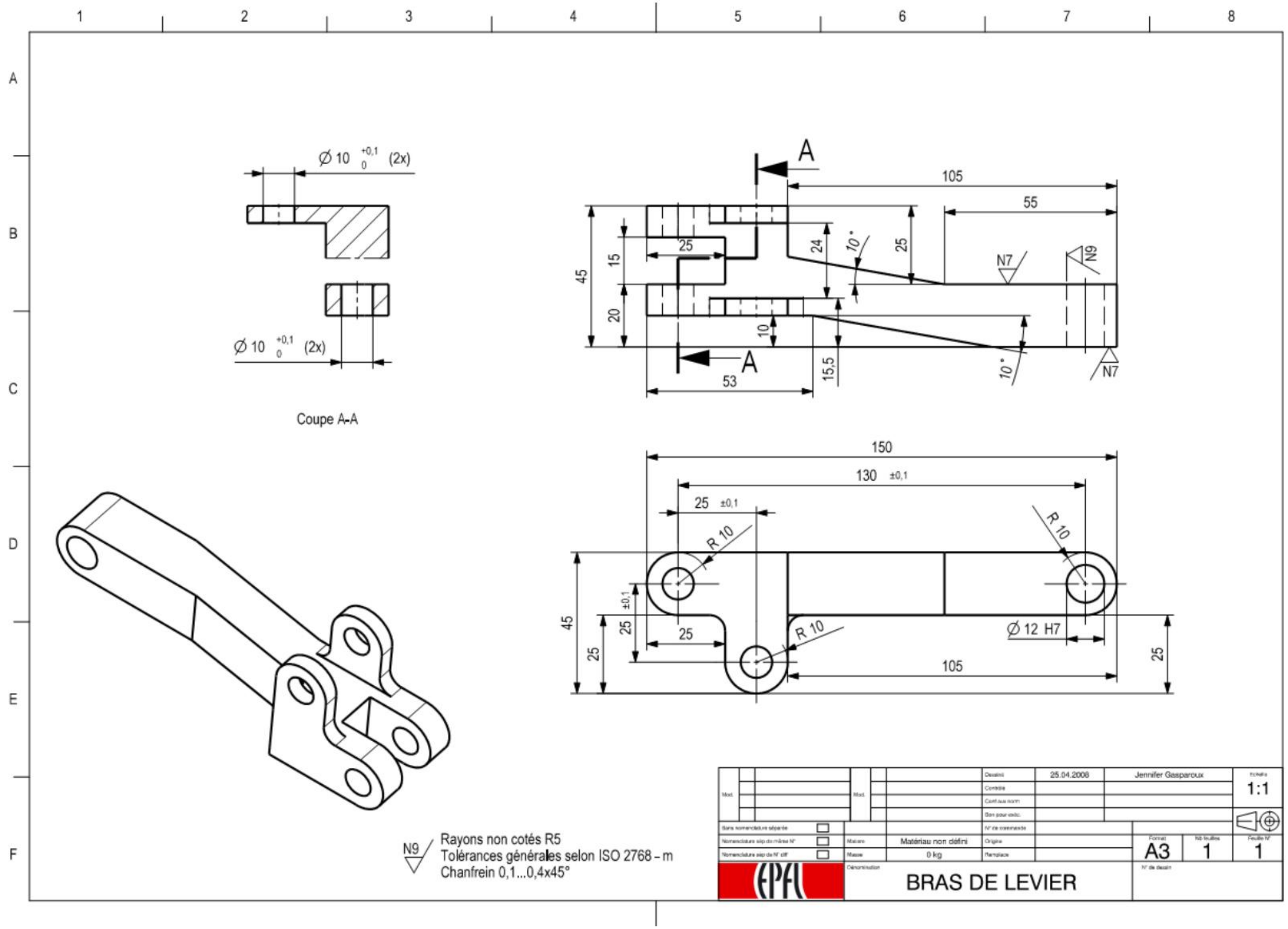
in the **Main Body** tree view (part outline turns red)

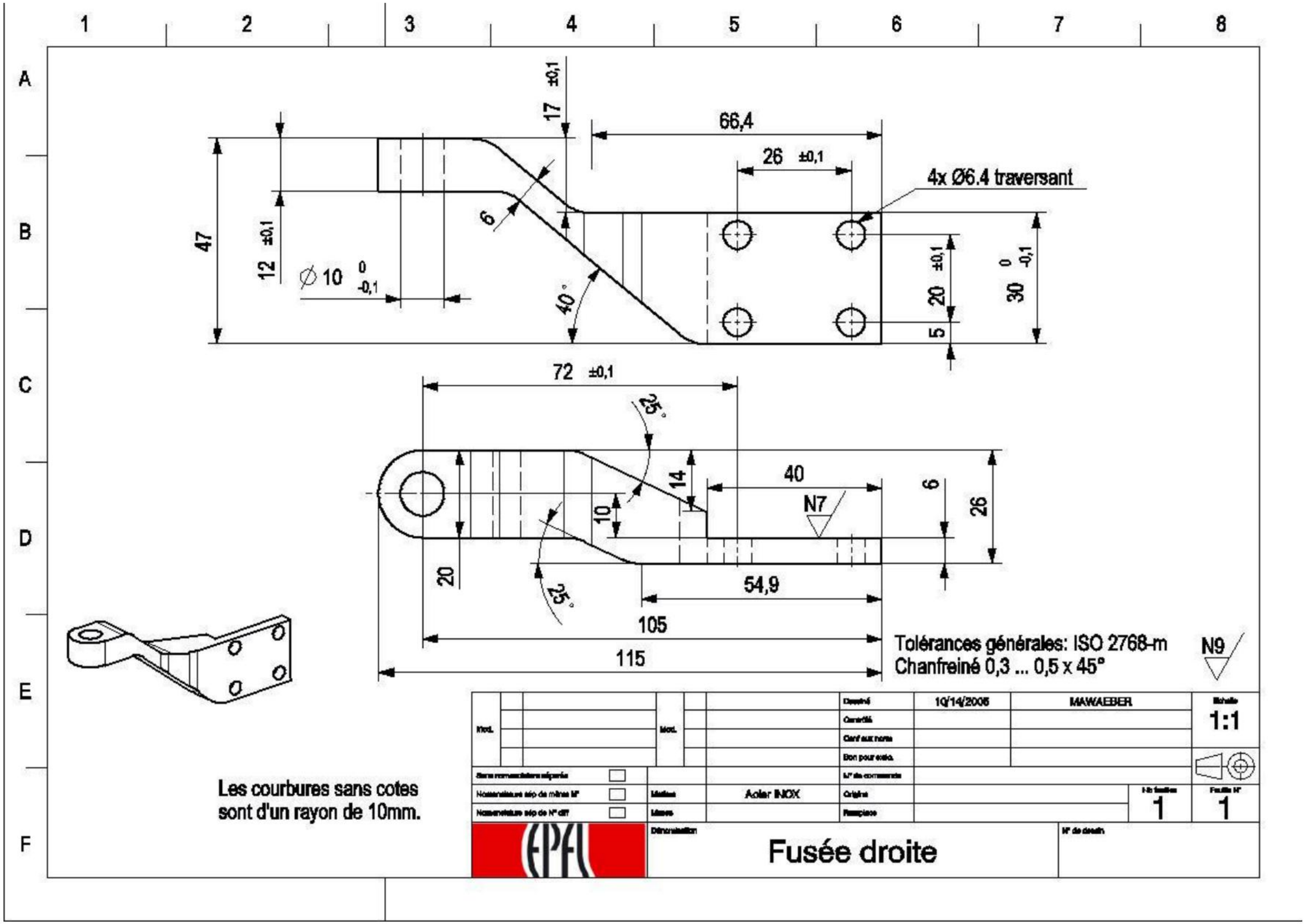
Apply Material > **OK**



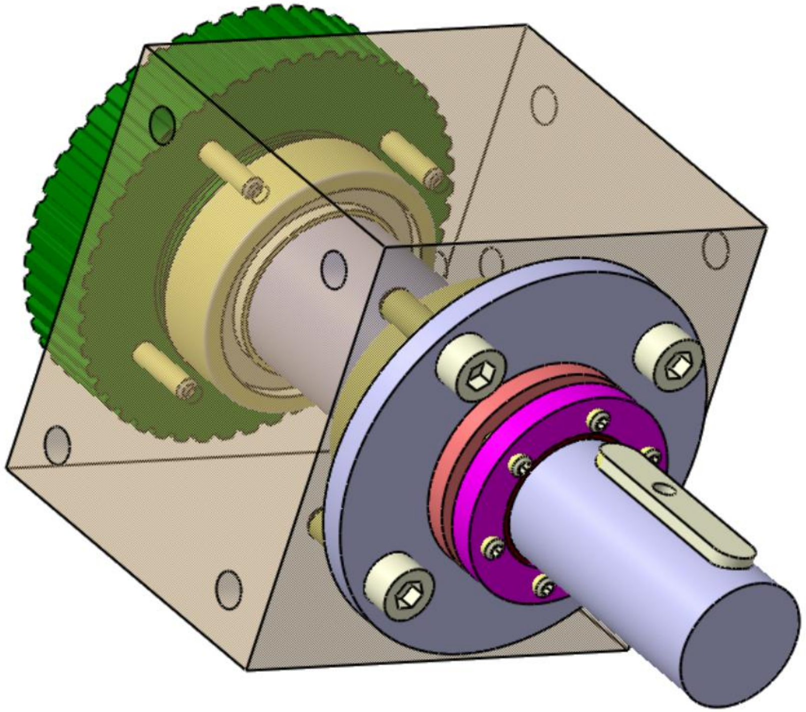
File > Save

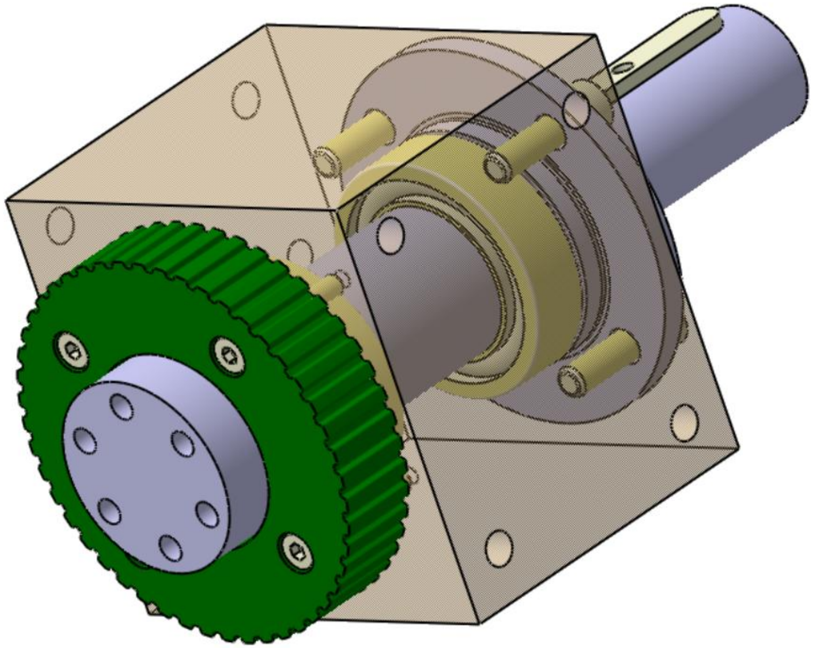
17. EXAMPLE OF TEST 2




17 ±0,1
66,4
26 ±0,1
4x Ø6.4 traversant
47
12 ±0,1
Ø 10 0 -0,1
6
40°
20 ±0,1
30 0 -0,1
5
72 ±0,1
25°
14
40
N7
10
6
26
20
25°
54,9
105
115
Tolérances générales: ISO 2768-m
Chanfreiné 0,3 ... 0,5 x 45°
N9
Les courbures sans cotes
sont d'un rayon de 10mm.
10/14/2005
MAWAEBER
1:1
Acier INOX
1
1
Fusée droite

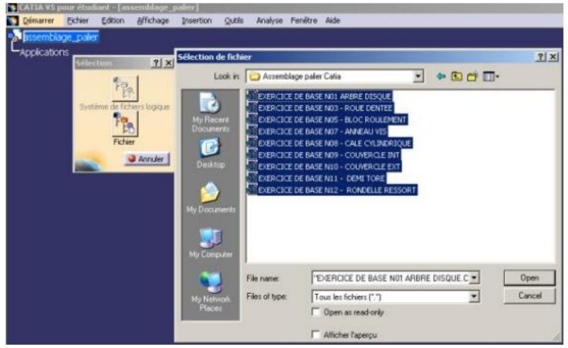
18. BASIC EXERCISE 13: 3D BEARING ASSEMBLY





FINAL RESULT

Start menu > **Assembly Design** > enter the name of the part: **ASSEMBLAGE_PALIER** > OK
File menu > **Save** > (default location, or as indicated) > file name: **ASSEMBLAGE_PALIER** > Save

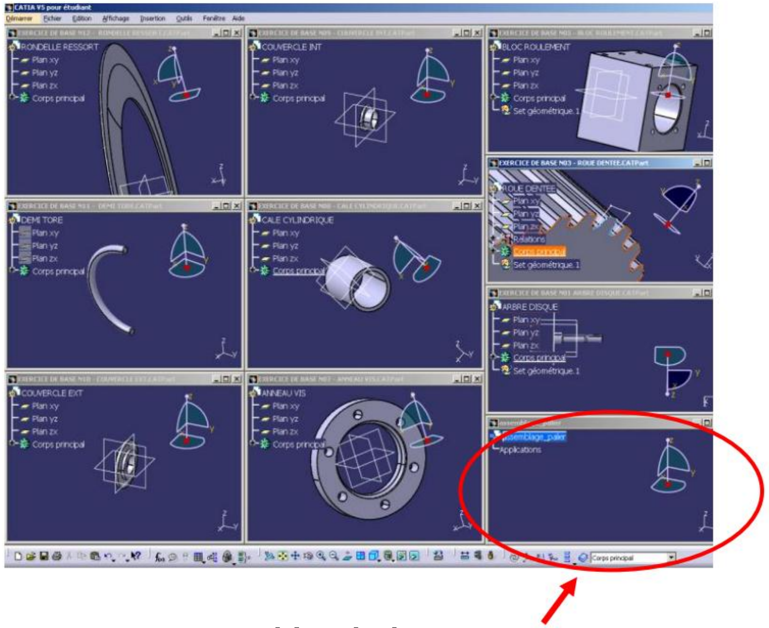


File > **open** > select all parts of the assembly while holding the key Ctrl pressed **(9 parts: gear wheel, spring washer, half torus, inner cover, outer cover, cylindrical wedge, bearing block, disc shaft, screw ring)**

Open

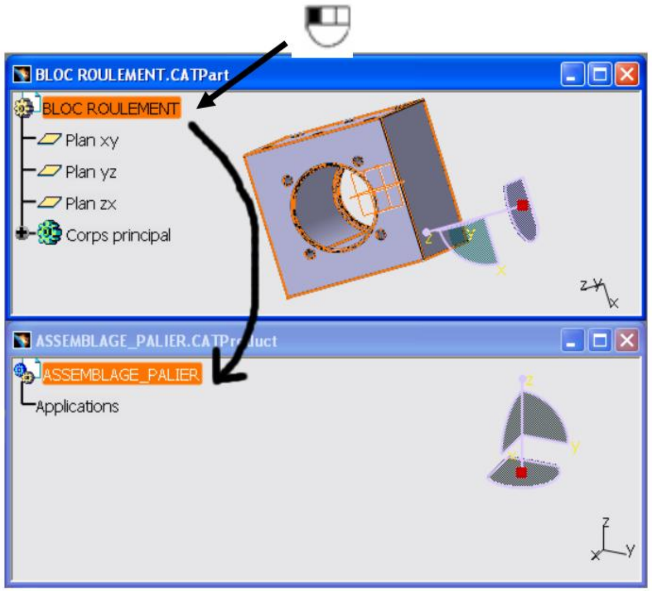
Window > **Horizontal Tile**

All assembly components are visible



Assembly window

Inserting parts into the assembly:



To move each part into the assembly:

Example for the BEARING BLOCK :


BEARING BLOCK in tree view (selected item turns red)
> Holding the selection, drag the element into the assembly

Repeat the operation for all the components

Once the components have been inserted into the assembly, it is no longer necessary to keep the windows open: **close** the component windows, keep only the landing assembly window.

Component positions :

After insertion, the parts of the assembly are superimposed on each other.

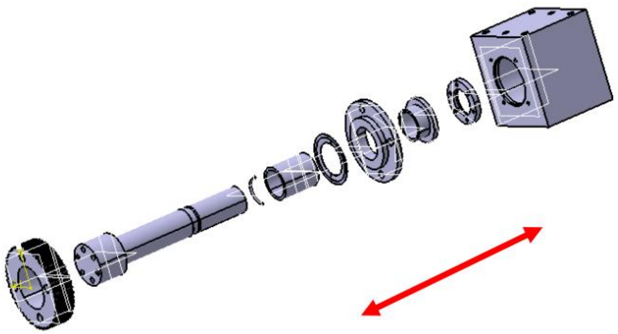
Using the manipulation function, move the components to distinguish them from each other:

Handling

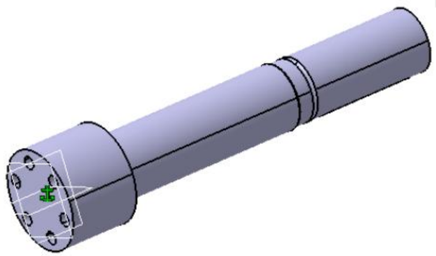
> choice of a **translation** axis

> Move the pieces along the chosen axis

>OK



Fixing the basic component:



It is **necessary** to fix the main part:

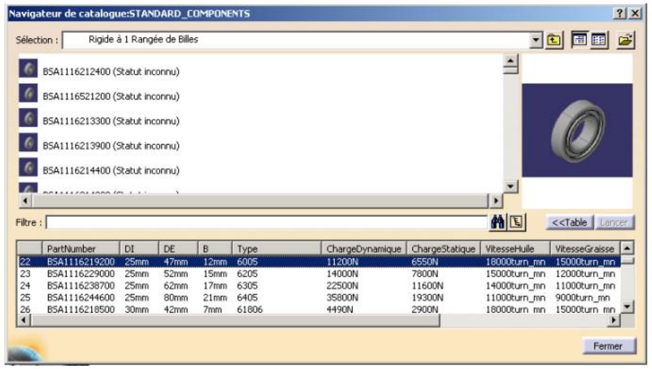
Fix > **select :** DISK SHAFT

(An anchor appears on the drawing)

Inserting catalog parts:

> **Catalog > 2x** **Bearings > 2x** **Rigid single row**

marbles 0



In the list at the bottom, presenting the characteristics of the bearings, choose **(2x**)

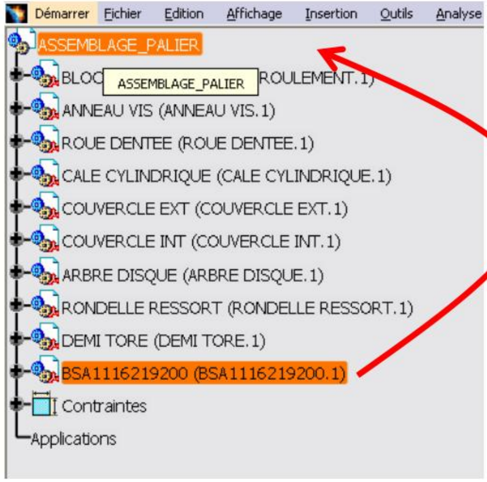
The **BSA1116219200** bearing (reference no. 22 in the catalog)

This is a 6005 type bearing: ÿ int: 25mm; ÿ outer: 47mm; width: 12mm

>OK

> Close catalog

Duplicating a part:



Assembly requires 2 bearings. To duplicate the component :

> BSA1116219200 in the tree view (selected item turns red)

> Drag the element into ASSEMBLAGE_PALIER while holding down the Ctrl key. The bearing reference should appear twice in the tree structure.

The duplicated element is often superimposed on the main element, the Manipulation tool allows you to separate the different elements.

Handling >

> choice of a movement axis

> Move the bearing along the chosen axis > OK

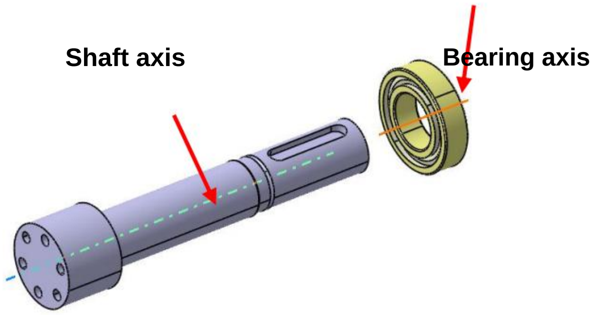
Assembly of components:

Before starting, it is possible, for better visibility, to hide the planes associated with each part:

Press the **Ctrl** + **F** keys > In **Name,** enter: Plan* > > Select (all planes turn red)
> **View > Hide/show > Hide/show** (this function hides all selected items) > **OK**

Assembly of the two bearings and the cylindrical shim on the shaft

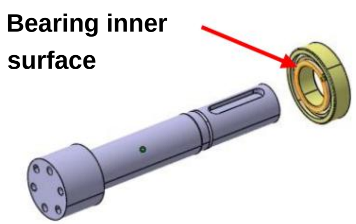
The tools that will be used are:
> **Coincidence** constraint
> **Contact** constraint

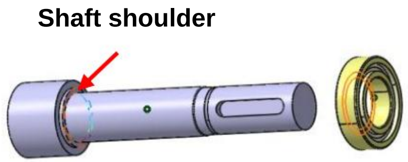


For example, for the first bearing:

> **> Close Wizard** message > bearing axis > shaft axis

Consequence: the bearing is aligned with the axis of the shaft

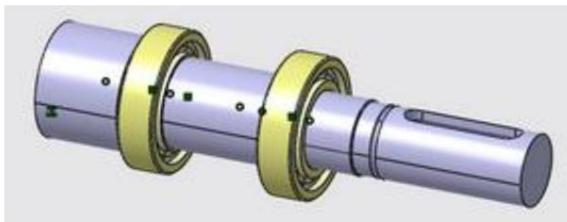




> **> Close Wizard** message > will be in the rolling surface which contact with the shaft (inner ring surface)

> the surface of the shaft that will be in contact with the bearing.

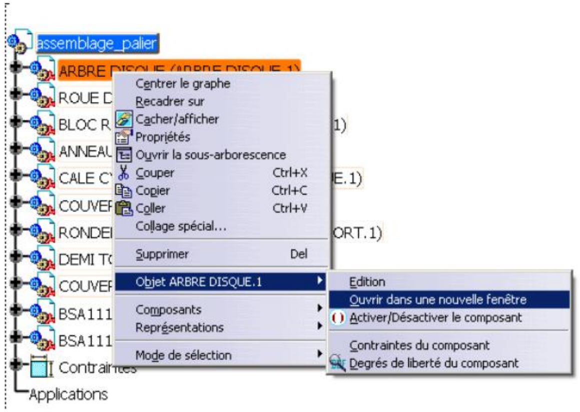
Consequence: The bearing comes into contact with the shaft shoulder



> Repeat the operation for the cylindrical shim and the second bearing.

Assembly of the half-torus in the groove:

It is not possible to create contacts between toroidal surface (In our case: contact: half-torus and groove of the shaft.)
The approach consists in creating a central point at the level of the throat of the shaft, then in distancing a plane belonging to the half-torus at this same point.



Point creation:

> DISK TREE(in tree)
> **Disc Tree Object.1** > **Open in a new window**

In the new window:

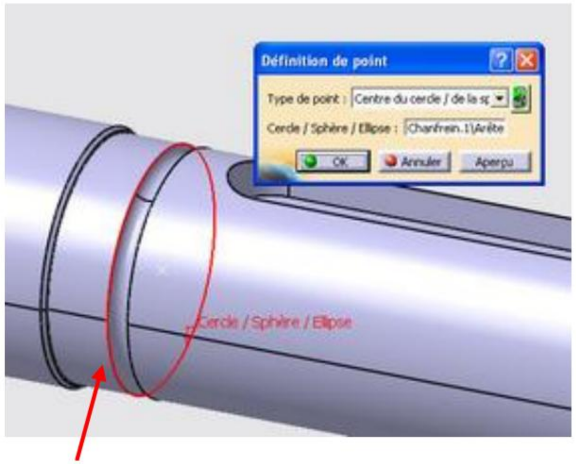
> **POINT**

> **on the edge of the groove**

Stitch type : **Center of circle > OK**
A dot should appear in the center of the throat

> **Save** the change (File – Save)

> **Close** disk tree window

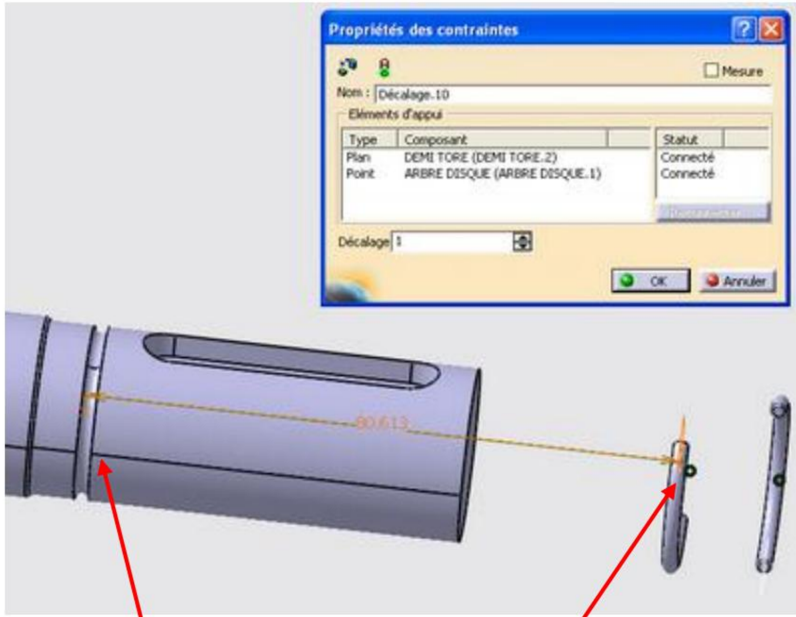


Edge of the throat

> **Duplicate** the half-torus as done previously for the bearing > **Move** the two half-toruses so as to distinguish them > **Create coincidence constraints** between the half-torus axis and the axis of the shaft. If necessary, rotate one of the 2 half-tori around its axis to prevent it from merging with the other.

> Following the same procedure as on page 148, show the planes of the half-tori perpendicular to their axis.

> Using the distance function  : **Constrain the plane perpendicular to the axis of the half torus to a distance of 1 (or -1)** from the point created previously (depending on the side of the groove chosen)
> **Repeat the operation** for the second half-torus



Point

Plane perpendicular to axis

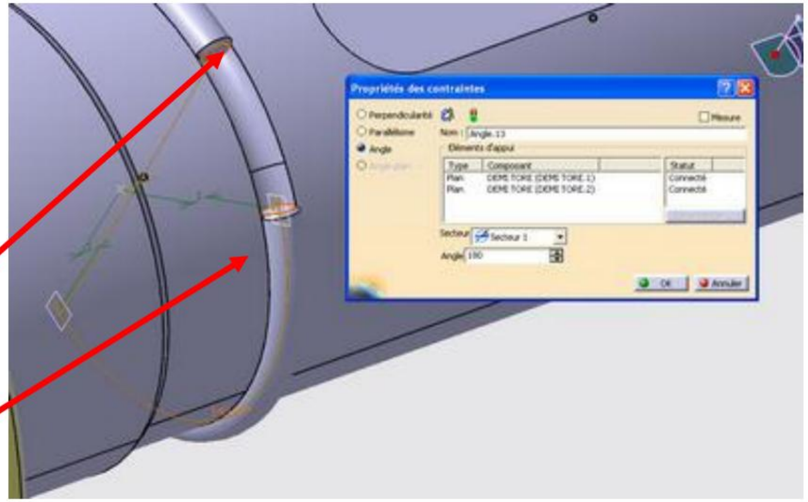
As we saw during the construction of the half-torus, each half-torus covers 179°. To prevent them from colliding, we can constrain them angularly:

>   **>** Between the faces of the two half tori > Angle: **180°**
> OK > OK

Note: This operation is equivalent to gluing the two surfaces together.

Half-torus: Side 1

Half-torus: Face 2

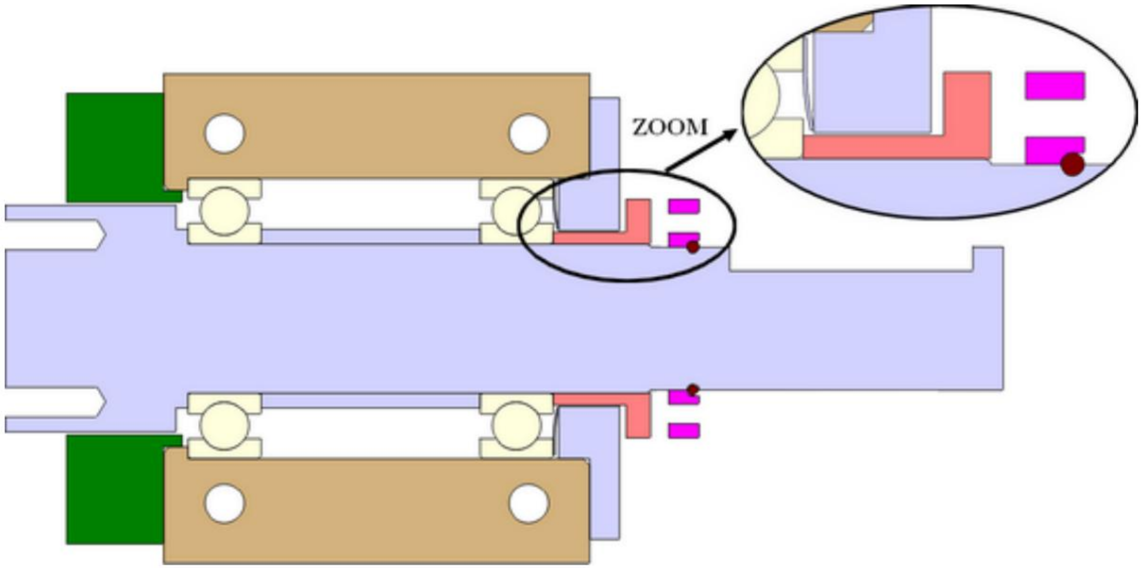


Assembly of the other components:

The assembly of the other components is done in the same way as the bearings and the cylindrical shim, using the tools seen previously.

Build the next assembly

Note: The illustration below shows the relative positioning of the parts in relation to the others



For the RING VIS part, you must create a point on the axis of the part and put a distance constraint between this point and the point already created in the center of the groove.

Note : for clarity, it is possible to modify the color of the parts and make them transparent:

 BEARING BLOCK (in the tree structure) >  **Properties** >  **Graph** tab > Choose a color > activate **Transparency** >  **Apply** > **OK**

M4 x 25 mm screw assembly

Inserting the screws:

> **Catalog >** In the **Selection drop-down list,**

go up to **3D_components > 2x VIS**

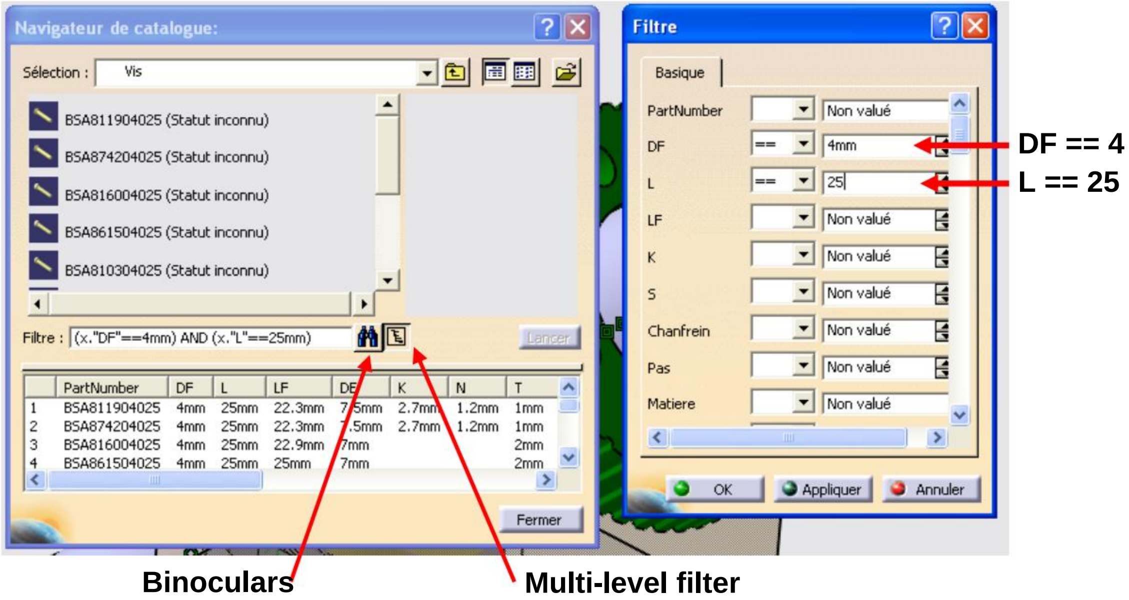
Search for screws using characteristics:

> **Multi-level filter >** **Binoculars**

In the **filter** window

>DF == 4 (screw diameter)
>L == 25 (screw length)

>OK



> 2x reference screws **BSA816004025** (3 e in the list) **> OK > close the Catalog window**

Screw assembly:

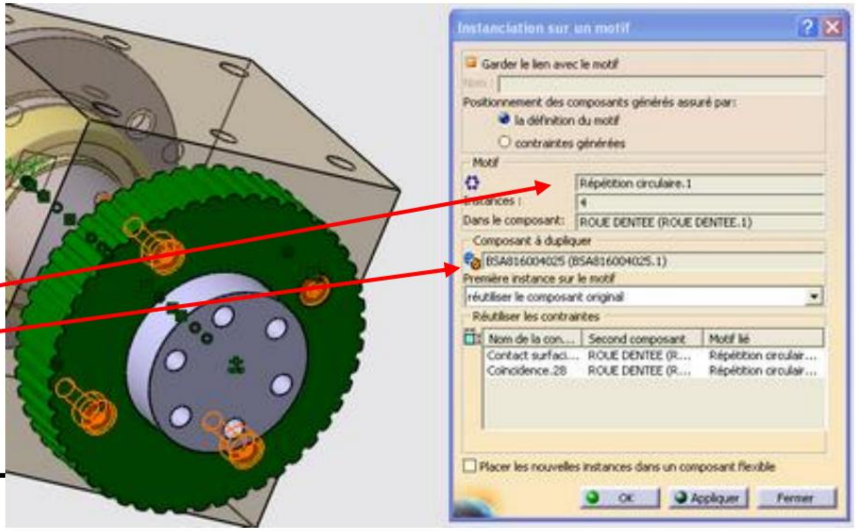
> **Constrain a screw** using the constraints of coincidence and contact with respect to the hole accommodating the M6 screws (in the cogwheel)

Copy of the 4 screws in a circular pattern

> **REUSE A PATTERN**

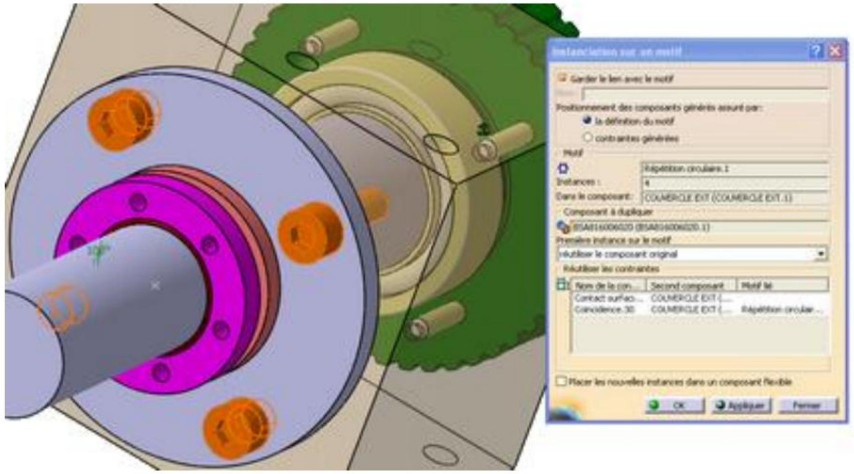
> Pattern : a circular pattern hole made for the wheel

> Component to duplicate : screws **BSA816004025**



> Apply > OK

M6 x 20 mm screw assembly

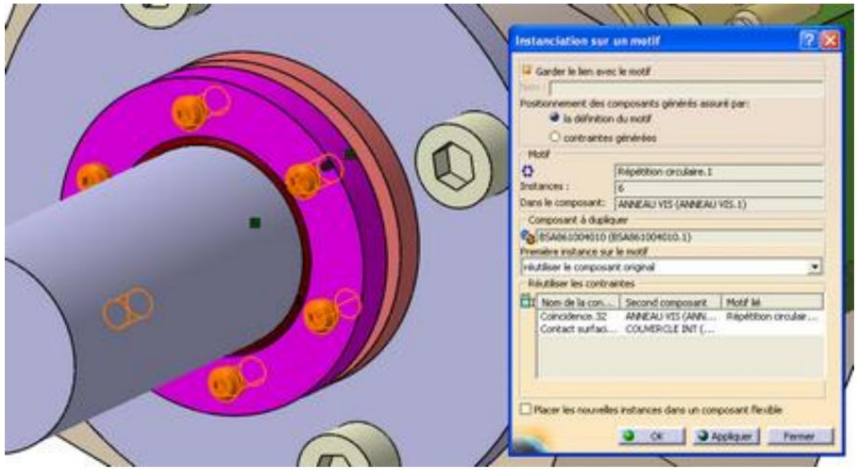


Refer to the figure on the right for the positioning of the M6 x 20 mm screws

Perform the same operations as those seen previously

The screws to choose are: **BSA816006020** (10th in the list)

M4 x 10 mm screw assembly



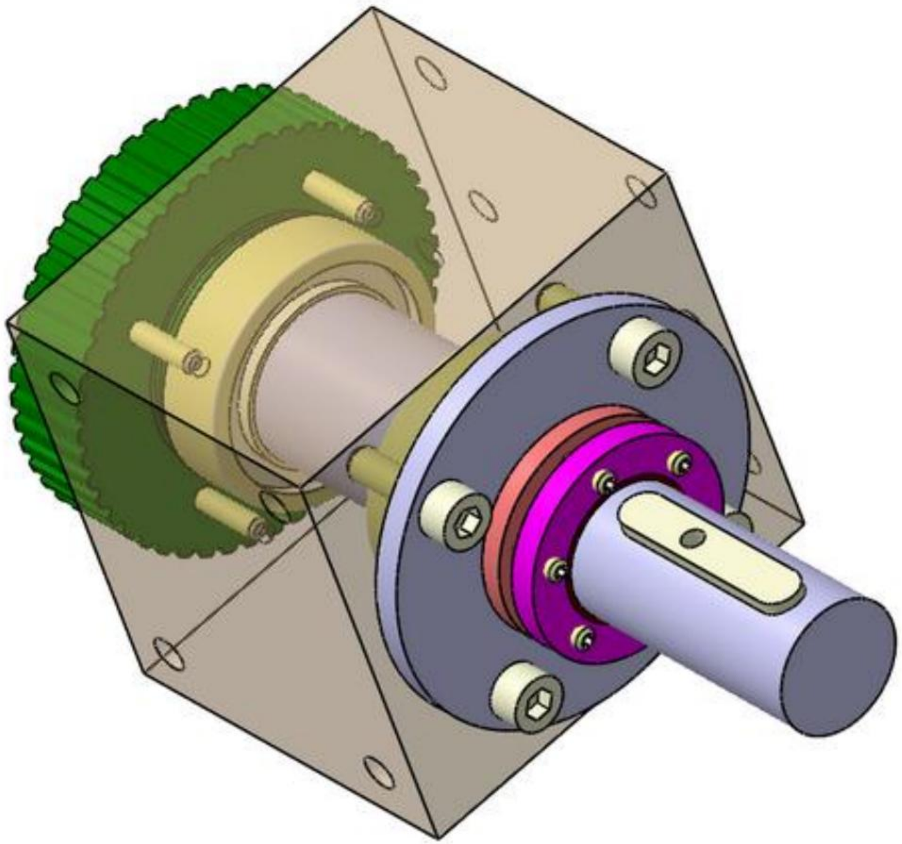
Refer to the figure on the right for the positioning of the M4 x 10 mm screws

Perform the same operations as before

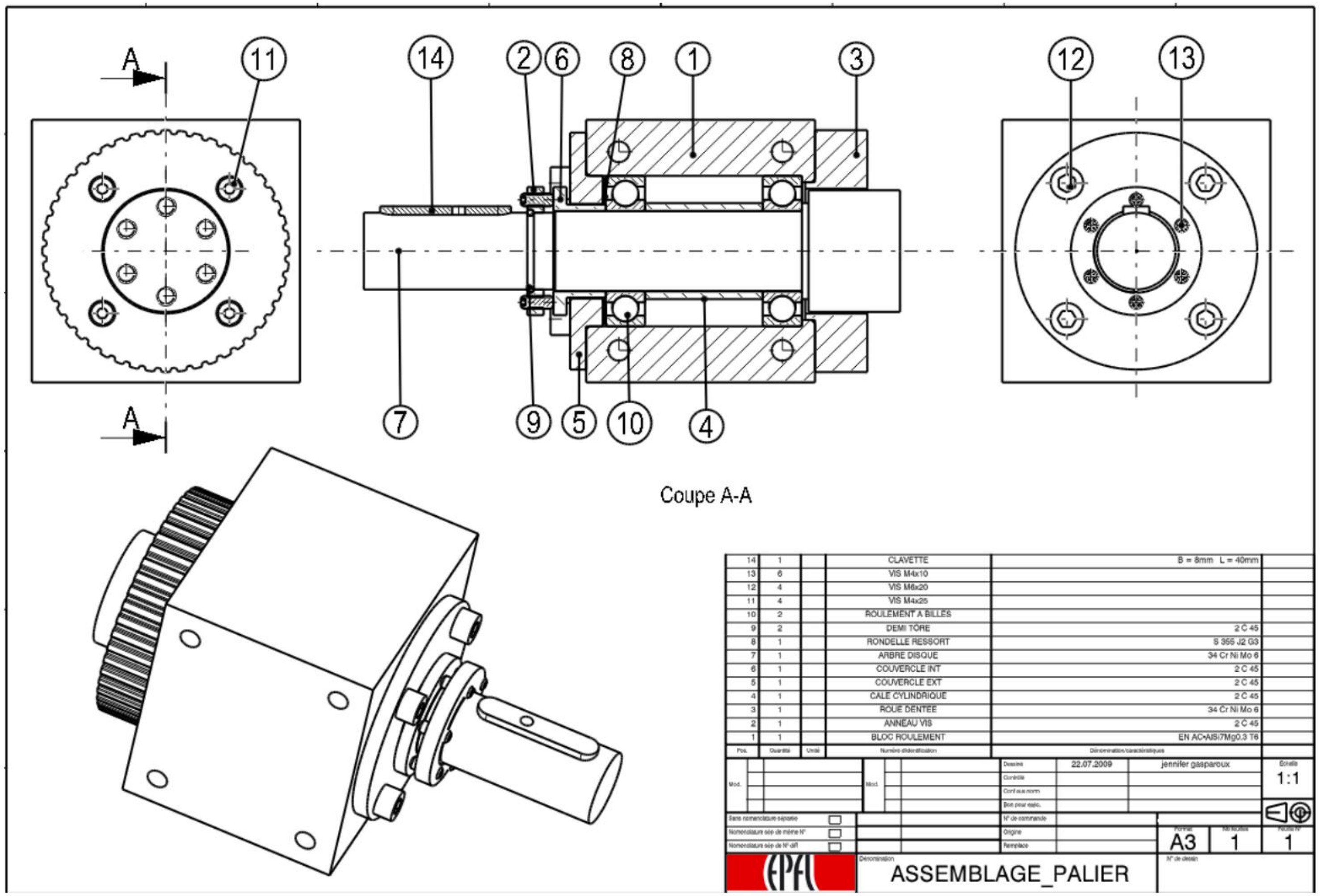
The screws to choose are: **BSA861004010** (14th in the list)

Key Assembly

Choose from the catalog the key (width 8 mm and length 40 mm) reference: **BSA1026063000**
Assemble the key to the DISK SHAFT using all the tools seen previously

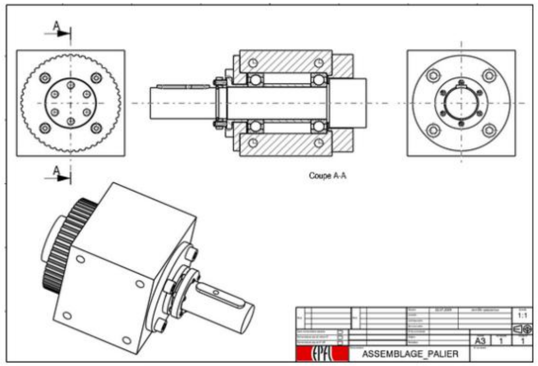


19. BASIC EXERCISE #14: 2D BEARING ASSEMBLY



FINAL RESULT

Start Menu > **Drafting** > Standard: **DETAIL ISO** > Shape Style: **A3 ISO** > Orientation: **Landscape** > OK
File menu > **Save** > (default location, or as indicated) > file name: **ASSEMBLAGE_PALIER** > Save



Perform assembly drawing.

Note: In section views, shafts should not be cut:

> DISK TREE (in the tree structure of the 3D window) > **Properties** > tab **Drawing** > Enable **Uncut in section views** > **Apply** > **OK**

To insert the parts list:

> LIST OF PIECES

To edit this list:

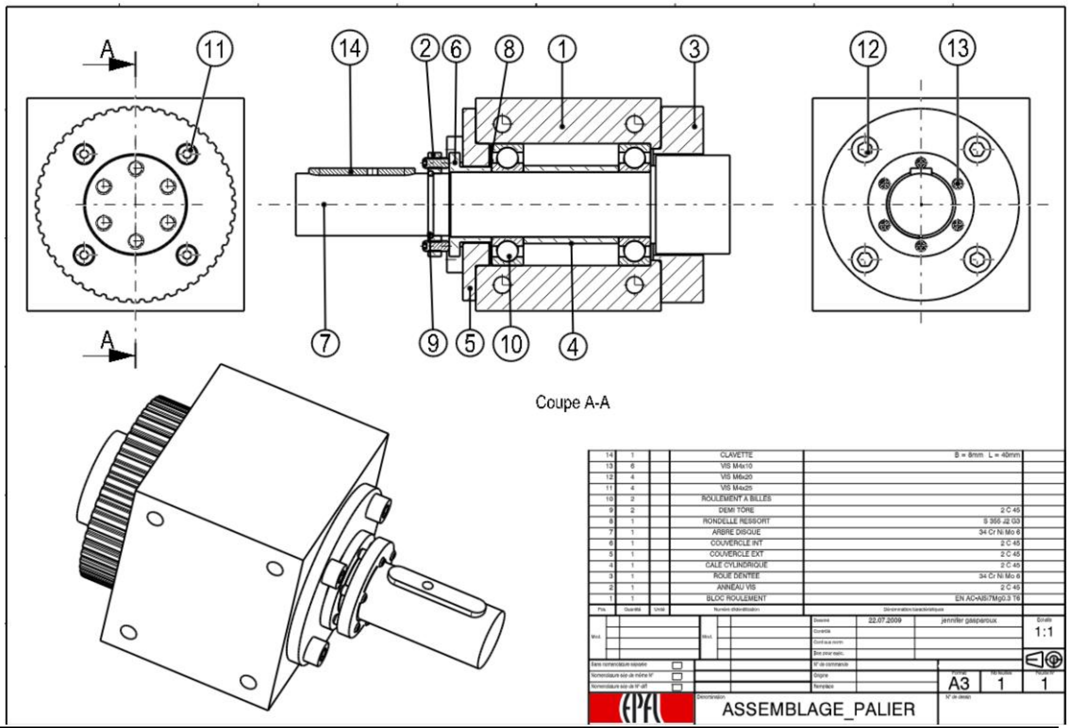
> Edit menu > **Layer Background**

To return to the drawing:

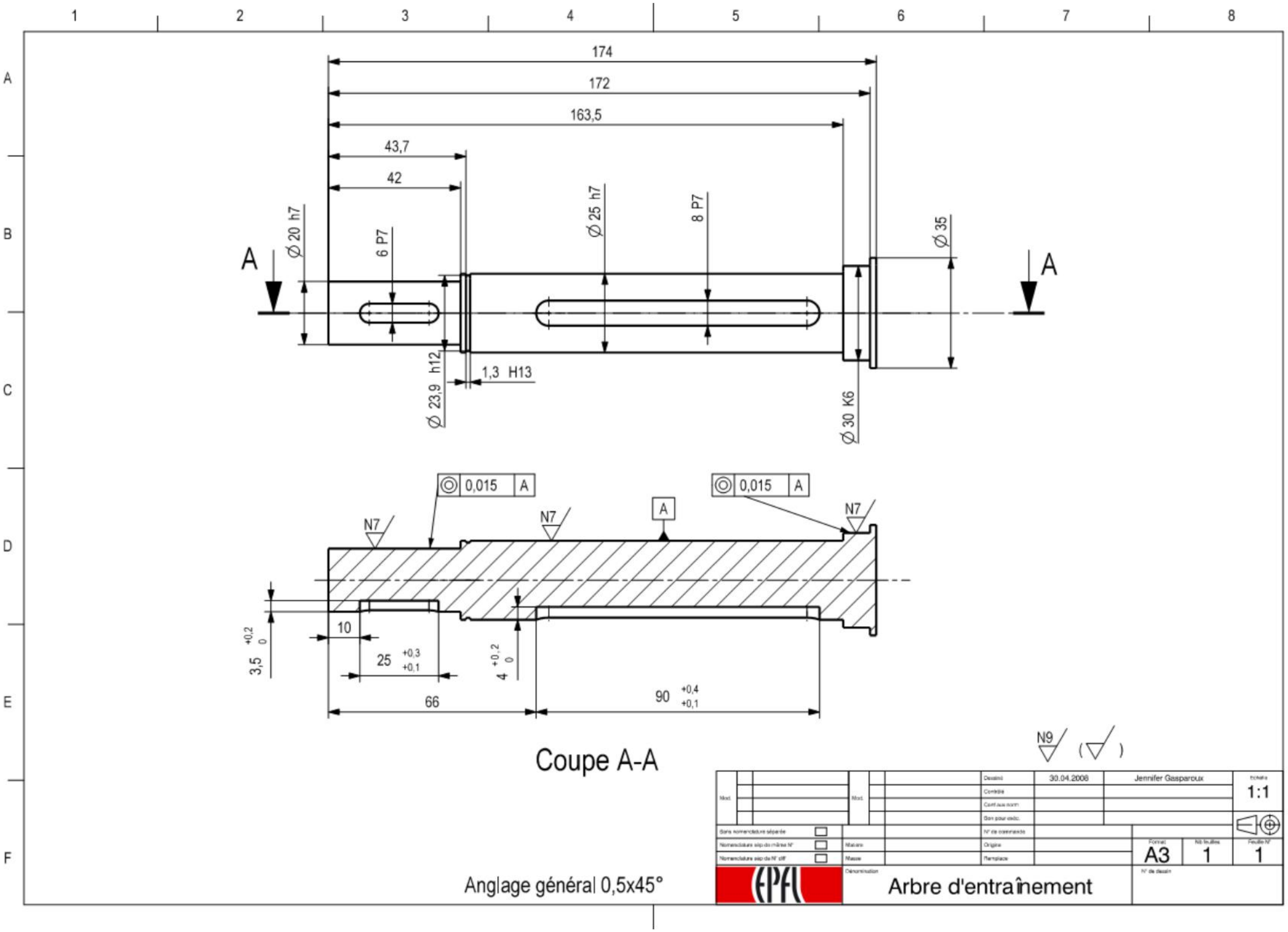
> Edit menu > **View layer**

To number the parts:

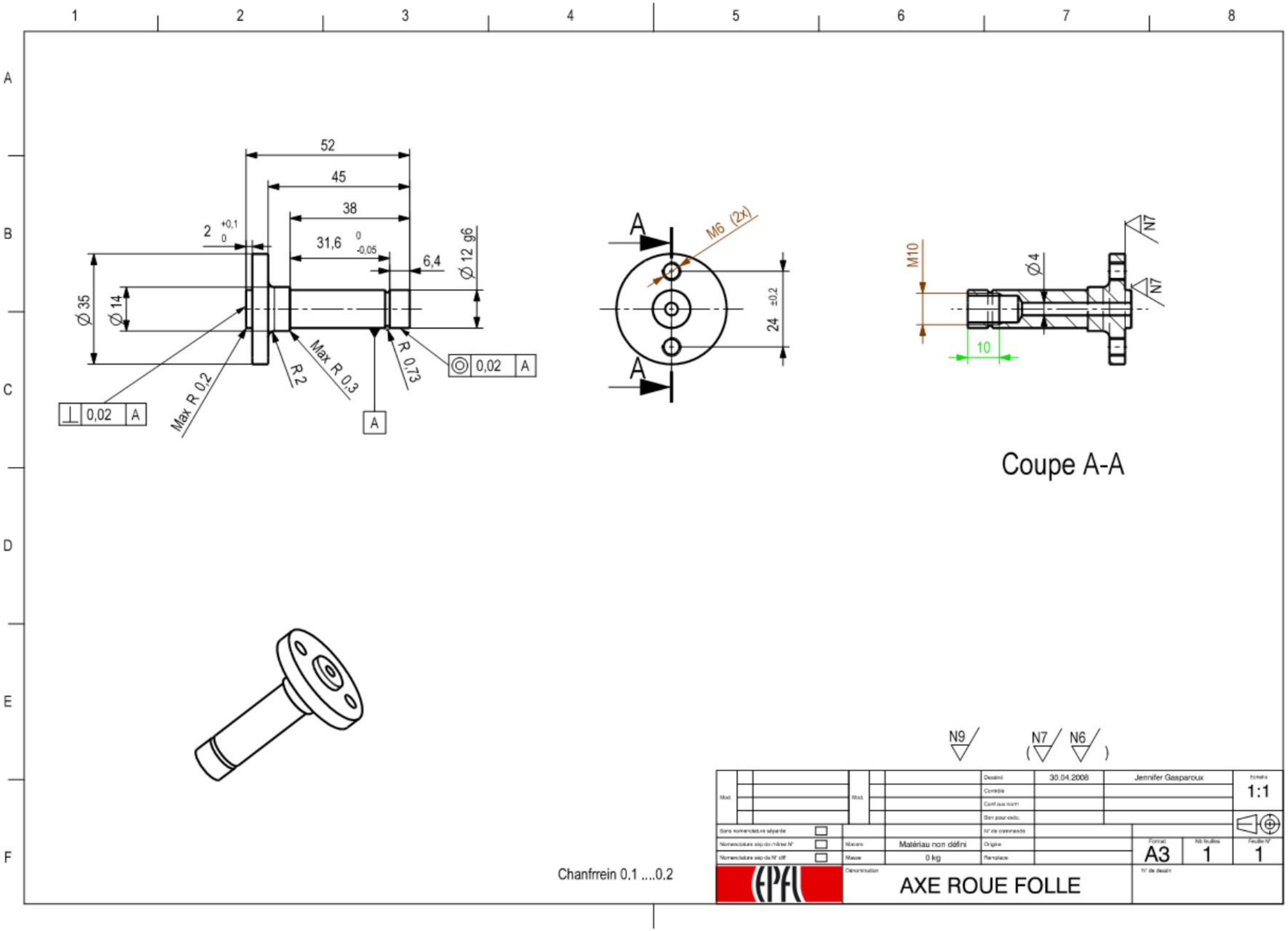
> **PART NUMBER** (behind TEXT) (of course the part number must match the number in the parts list)

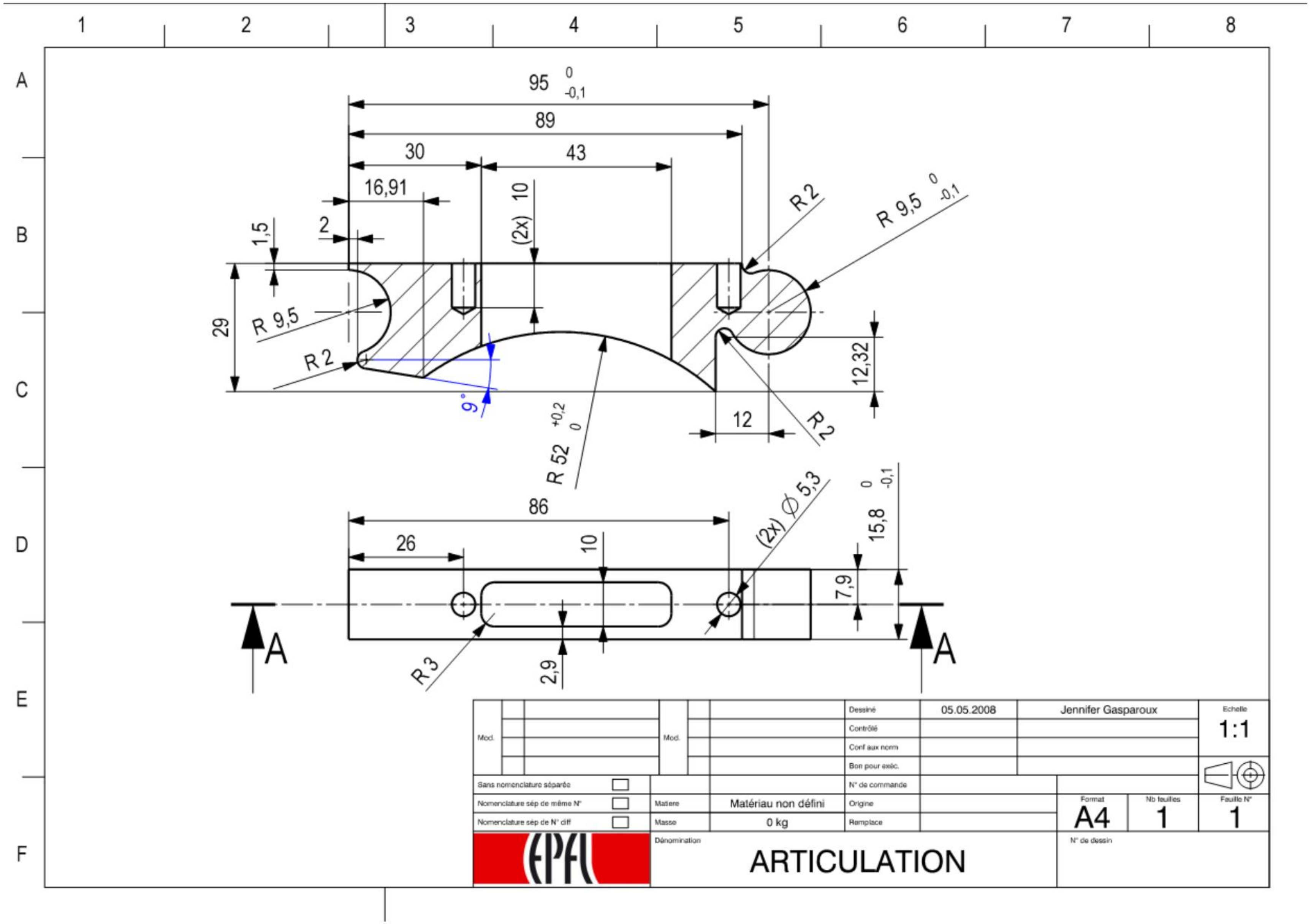


20. ADDITIONAL EXERCISES

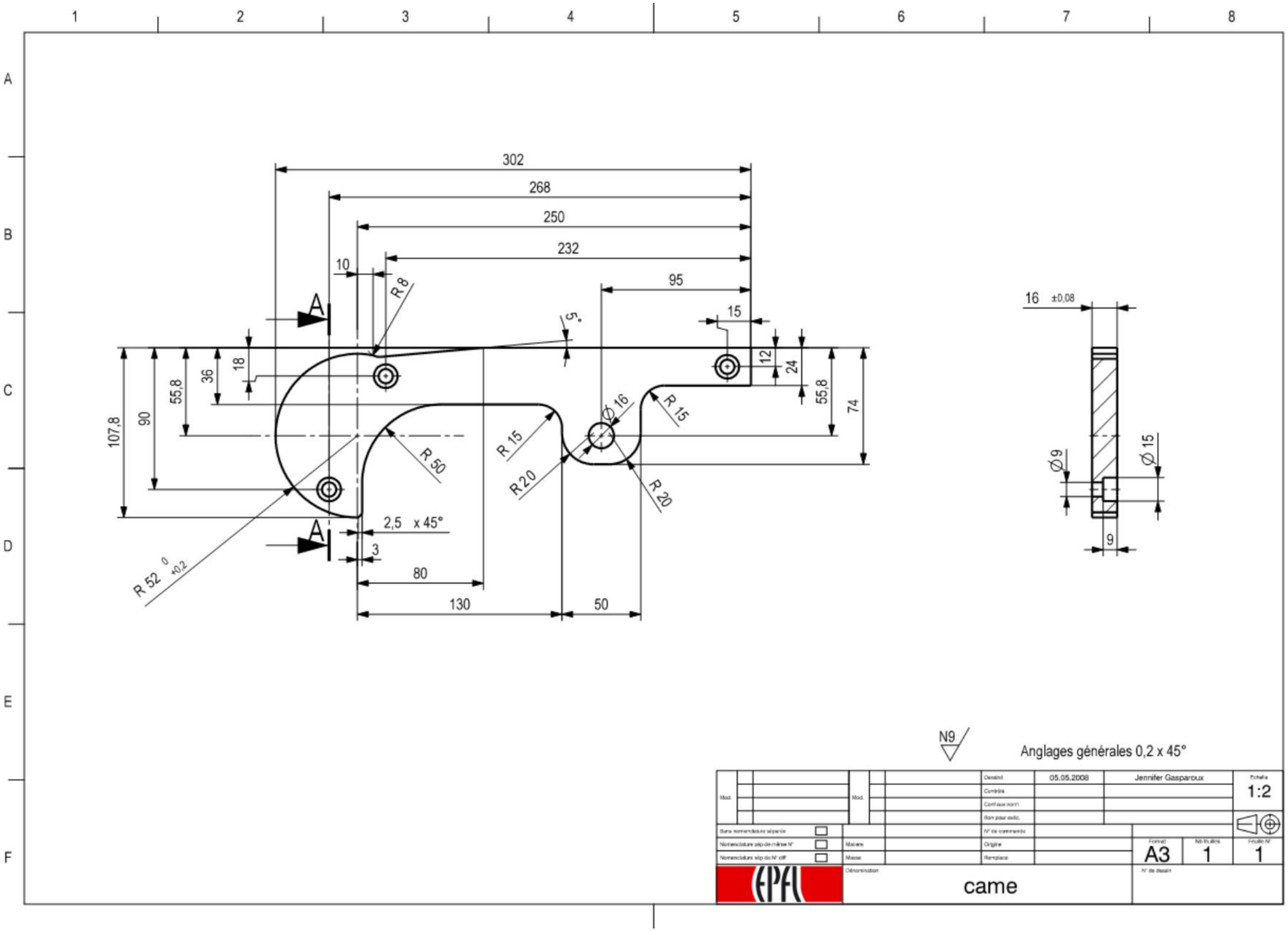


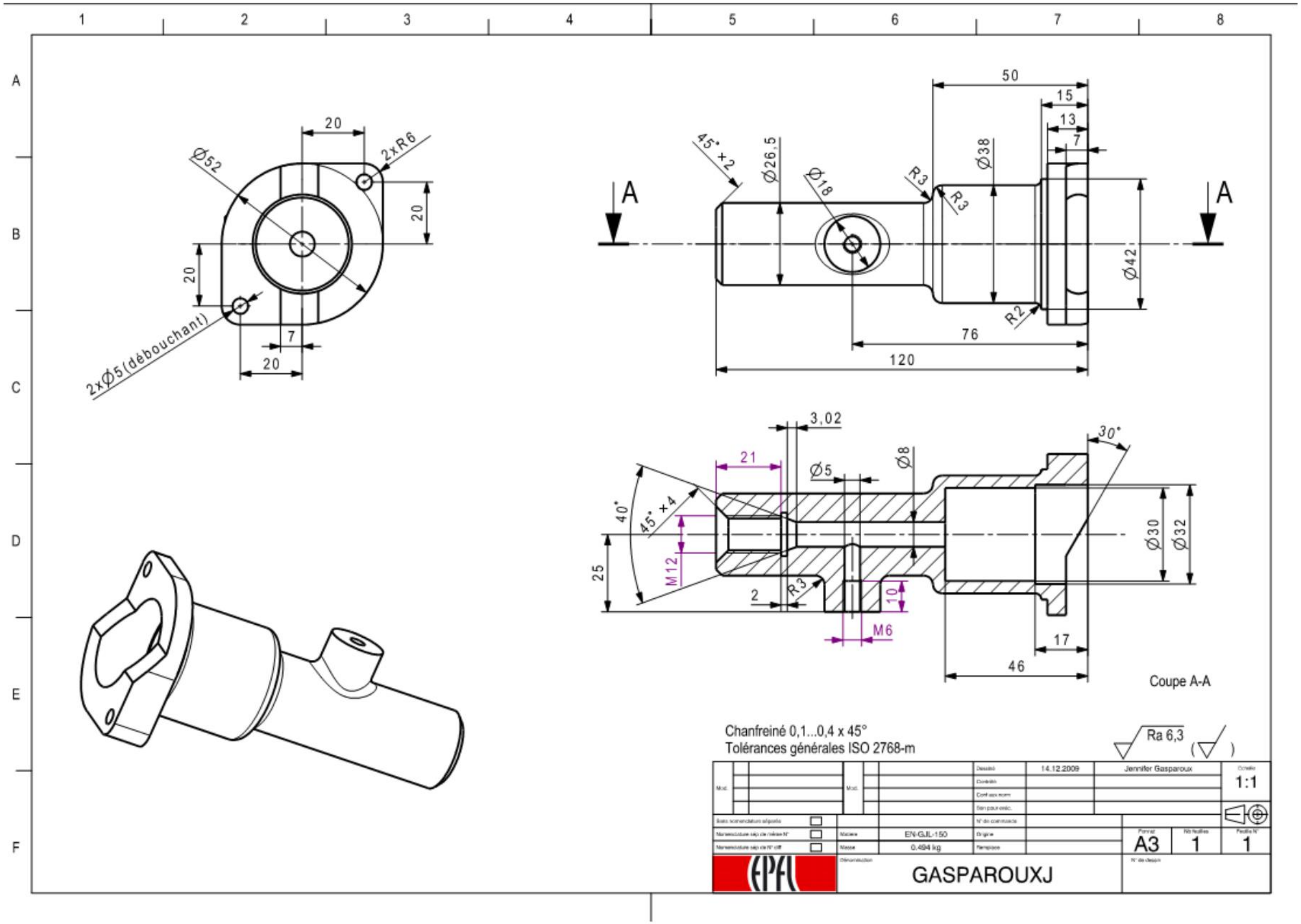
174
172
163,5
43,7
42
Ø 20 h7
6 P7
Ø 25 h7
8 P7
Ø 35
A
A
Ø 23,9 h12
1,3 H13
Ø 30 K6
0,015 A
0,015 A
N7
N7
N7
A
10
25 +0,3 +0,1
3,5 +0,2 0
4 +0,2 0
66
90 +0,4 +0,1
Coupe A-A
N9
Anglage général 0,5x45°
30.04.2008
Jennifer Gasparoux
1:1
A3
1
1
Arbre d'entraînement

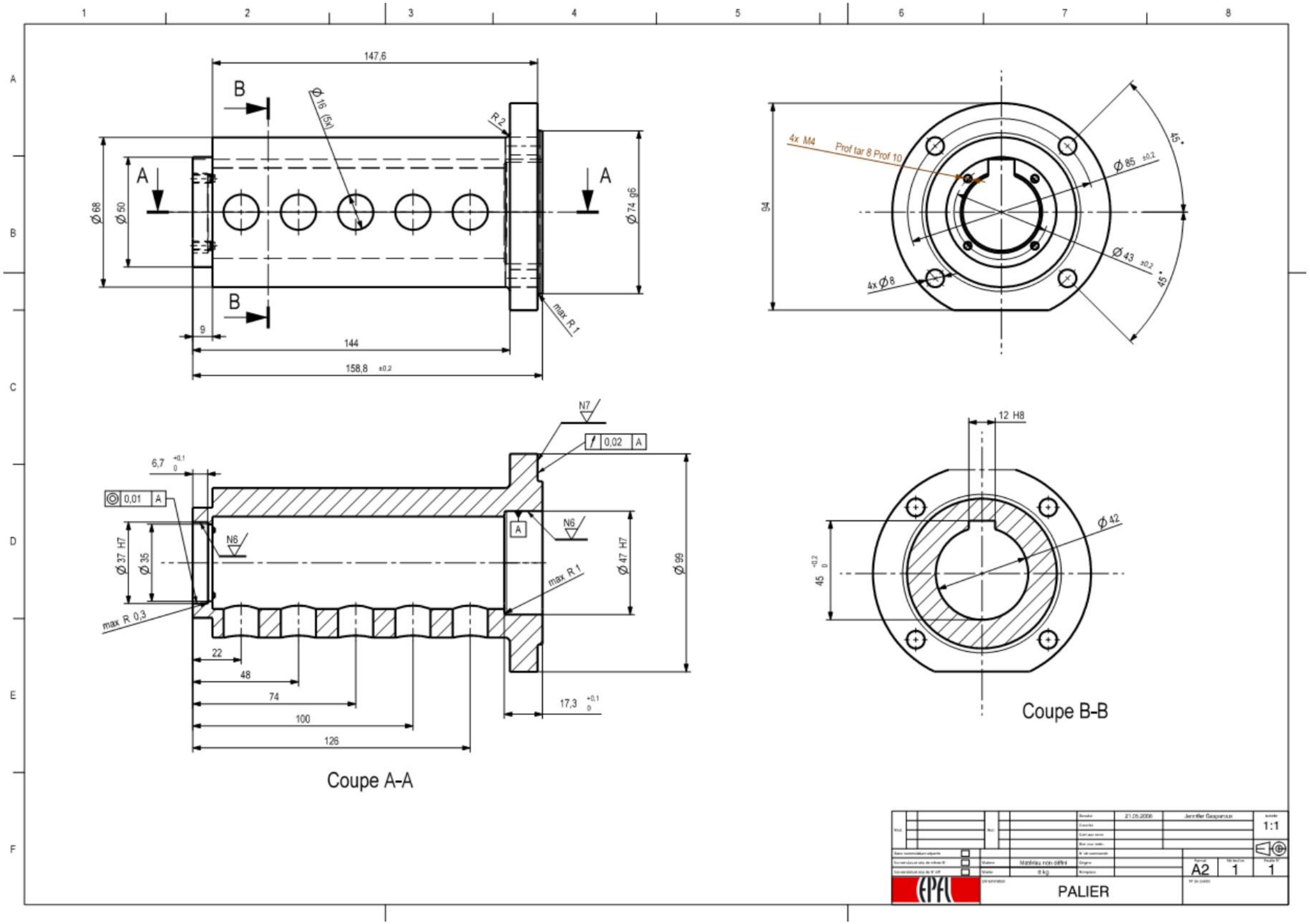

52
45
38
31,6
6,4
Ø 35
Ø 14
Ø 12 g6
Max R 0,2
R 2
Max R 0,3
R 0,73
0,02 A
0,02 A
A
M6 (2x)
24 ±0,2
M10
Ø 4
10
N7
Coupe A-A
N9
N7
N6
Chanfrrein 0.1 ....0.2
30.04.2008
Jennifer Gasparoux
1:1
Matériau non défini
0 kg
A3
1
1
AXE ROUE FOLLE



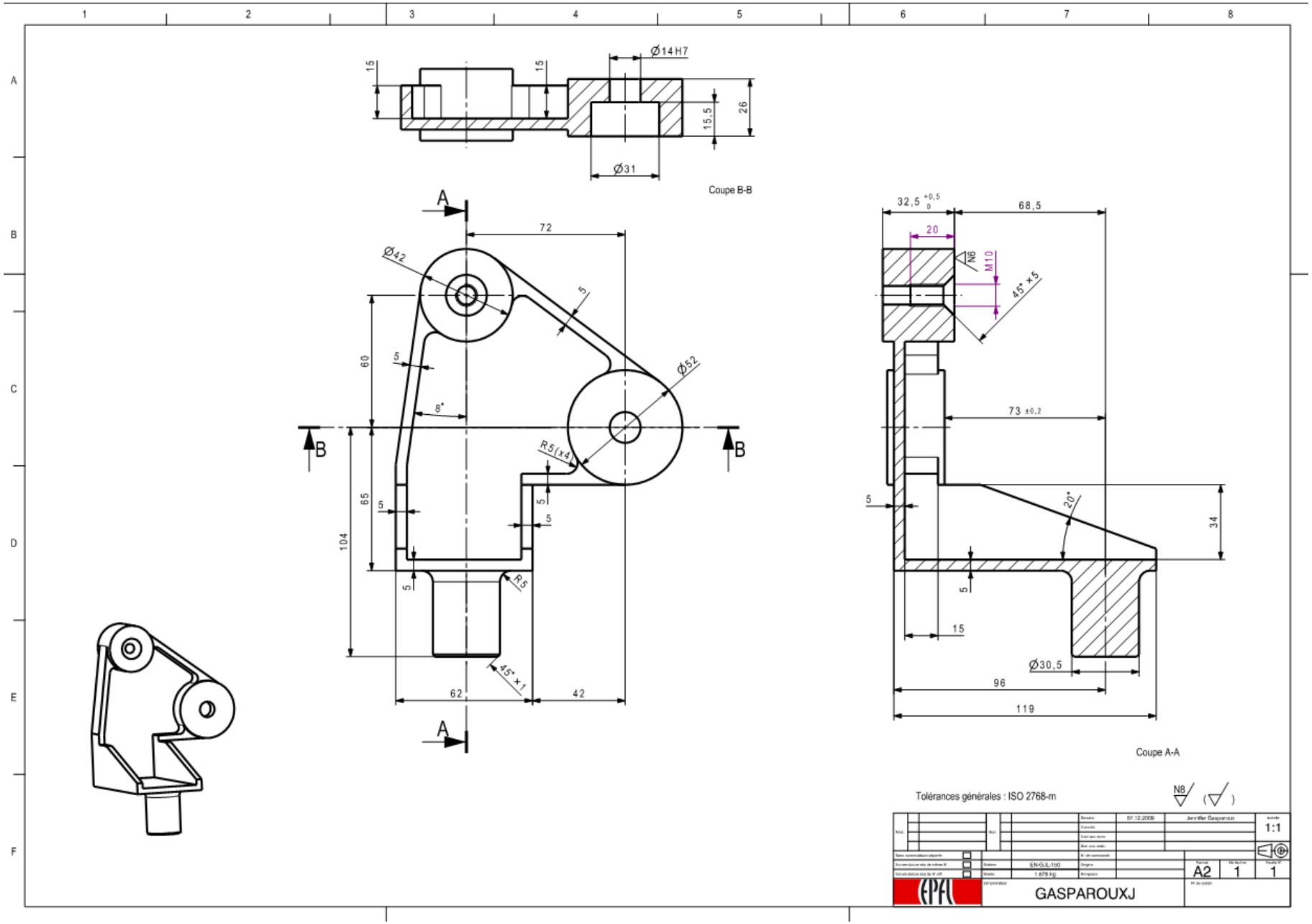
95 0 -0,1
89
30
43
16,91
2
1,5
(2x) 10
R 2
R 9,5 0 -0,1
29
R 9,5
R 2
9°
R 52 +0,2 0
12
12,32
R 2
86
26
10
(2x) ⌀ 5,3
15,8 0 -0,1
7,9
R 3
2,9
A
A
Dessiné
05.05.2008
Jennifer Gasparoux
Echelle
1:1
Contrôlé
Conf aux norm
Bon pour exéc.
Mod.
Mod.
Sans nomenclature séparée
Nomenclature sép de même N°
Nomenclature sép de N° diff
N° de commande
Matiere
Matériau non défini
Origine
Masse
0 kg
Remplace
Format
A4
Nb feuilles
1
Feuille N°
1
Dénomination
ARTICULATION
N° de dessin


302
268
250
232
95
10
R 8
5°
15
12
24
18
36
55,8
90
107,8
55,8
74
A
A
Ø 16
R 15
R 15
R 50
R 20
R 20
2,5 x 45°
3
80
130
50
R 52 0 +0,2
16 ±0,08
Ø 9
Ø 15
9
N9
Anglages générales 0,2 x 45°
05.05.2008
Jennifer Gasparoux
1:2
A3
1
1
came


20
Ø52
2xR6
20
20
2xØ5(débouchant)
7
20
50
15
13
7
45° x2
Ø26,5
Ø18
R3
R3
Ø38
A
A
Ø42
R2
76
120
3,02
21
Ø5
Ø8
30°
40°
45° x4
M12
25
2
R3
10
M6
Ø30
Ø32
17
46
Coupe A-A
Chanfreiné 0,1...0,4 x 45°
Tolérances générales ISO 2768-m
Ra 6,3
14.12.2009
Jennifer Gasparoux
1:1
EN-GJL-150
0.494 kg
A3
1
1
GASPAROUXJ

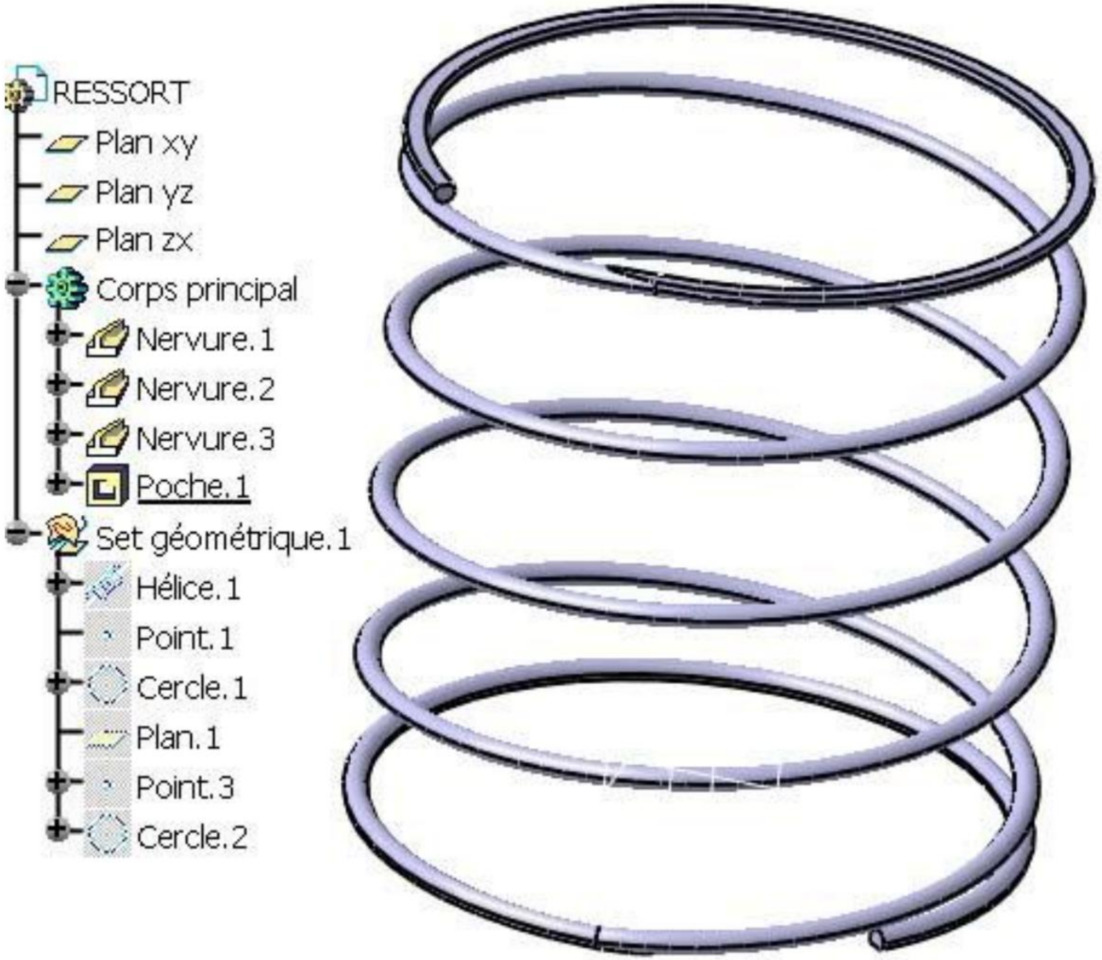


147,6
B
Ø 16 (5x)
R 2
A
A
Ø 68
Ø 50
Ø 74 g6
max R 1
9
144
158,8 ±0,2
4x M4
Prof tar 8 Prof 10
94
Ø 85 ±0,2
Ø 43 ±0,2
4x Ø8
45°
45°
N7
0,02 A
6,7
0,01 A
N6
N6
Ø 37 H7
Ø 35
Ø 47 H7
Ø 99
max R 1
max R 0,3
22
48
74
100
126
17,3
Coupe A-A
12 H8
Ø 42
45
Coupe B-B
1:1
A2
1
1
Matériau non défini
PALIER



Ø14 H7
15
15
15,5
26
Ø31
Coupe B-B
A
72
Ø42
5
60
5
8°
Ø52
B
B
R5(x4)
65
104
5
5
5
5
R5
45° ×1
62
42
A
32,5 +0,5 0
68,5
20
N6
M10
45° ×5
73 ±0,2
5
20°
34
5
15
Ø30,5
96
119
Coupe A-A
Tolérances générales : ISO 2768-m
N8
1:1
EN-GJL-100
A2
1
1
GASPAROUXJ

21. ADDITIONAL EXERCISE 1: COMPRESSION SPRING



Technical characteristics: • Ø of wire: 0.5 mm • Ø of winding: 14 mm • length L under load: 15.3mm • number of active spins Na = 4 • pitch p = 3.75mm

FINAL RESULT

Start menu > **Generative Shape Design** > enter the name of the part: **SPRING** > **OK**
File menu **> Save > L: Catia** > file name: **SPRING** > **Save**

HELIX :

  **PROPELLER** (behind  **CURVE)**

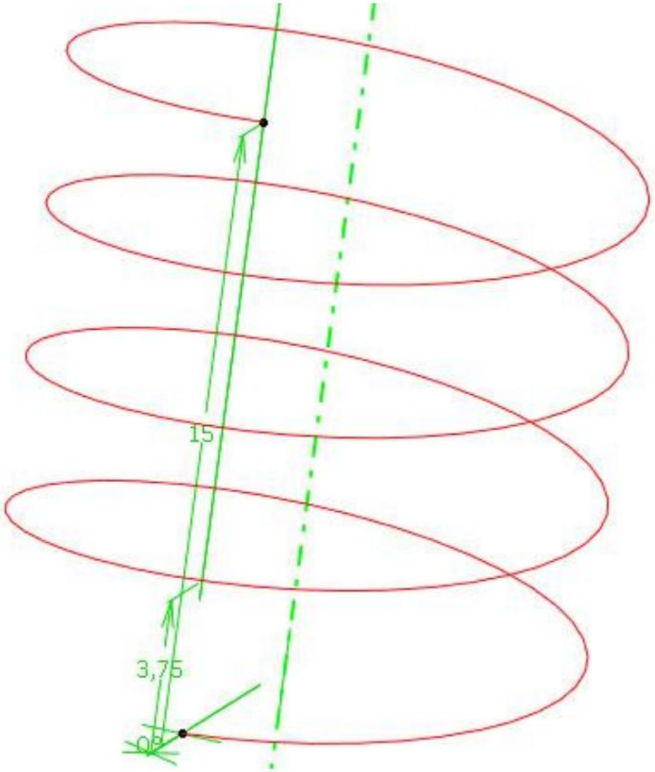
 Departure Point >  Create the point

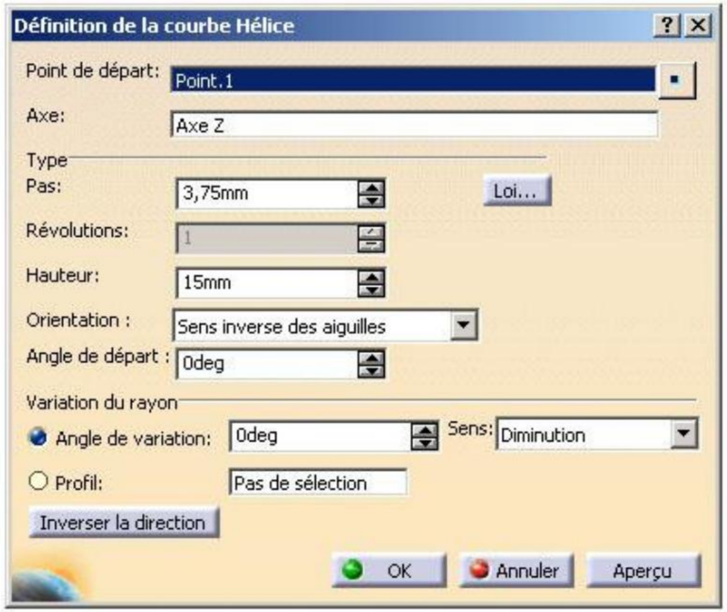
Coordinates : X=7mm
Y=0mm
Z=0mm

 okay

 Axis > Z axis  **> Pitch :** 3.75mm > **> Height :** 15mm >

 **okay**





Note: A spring is always drawn in its mounted configuration in an assembly: compressed for a compression spring, stretched for an extension spring.

Start > **Part Design**

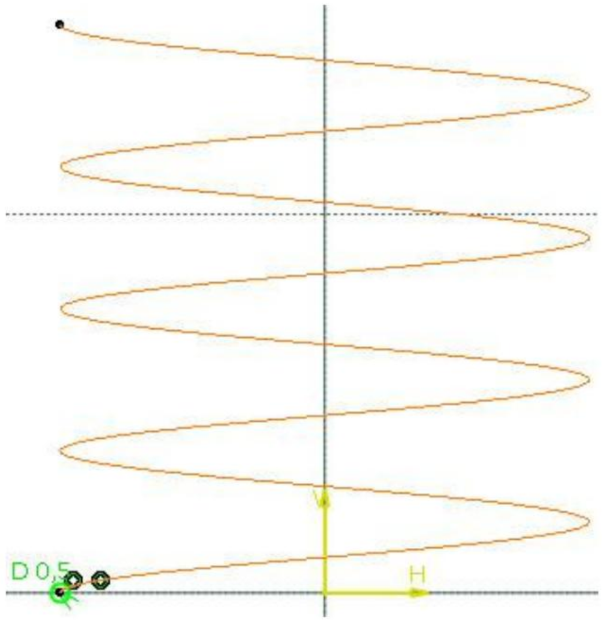
the **ZX** plane in the tree structure > **SKETCH**

CIRCLE > the center on the H axis > any diameter (the circle turns red)

CONSTRAINTS > set dimension > **2x** on the diameter value > enter **0.5 > OK**

center of the circle > CTRL the lower end of the helix > **SELECTED CONSTRAINTS IN A BOX OF DIALOGUE >** **Coincidence > OK**

LEAVING THE WORKSHOP



RIB:

Start > **Part Design**

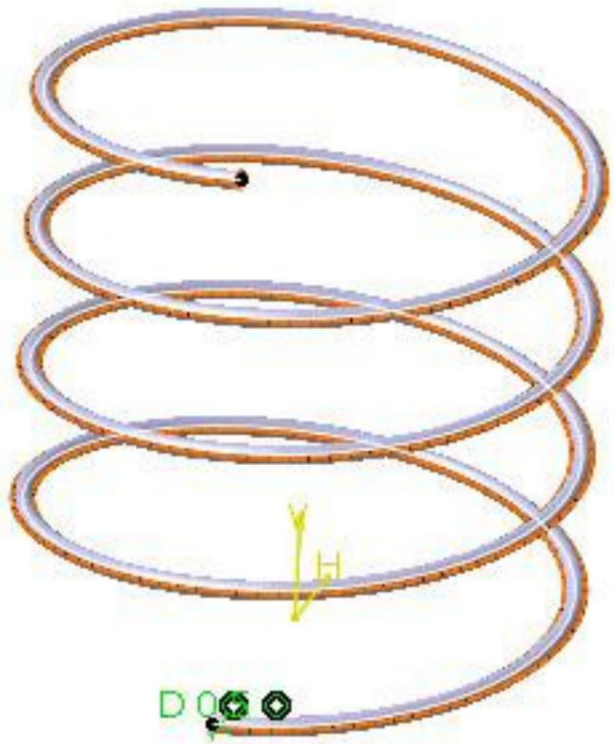
RIB

Message **Warning** > **OK**

Profile: Sketch.1 (default)

Guide curve: Propeller.1

okay



Start > **Generative Shape Design**

CIRCLES:

CIRCLE

Circle Type: Center – Radius

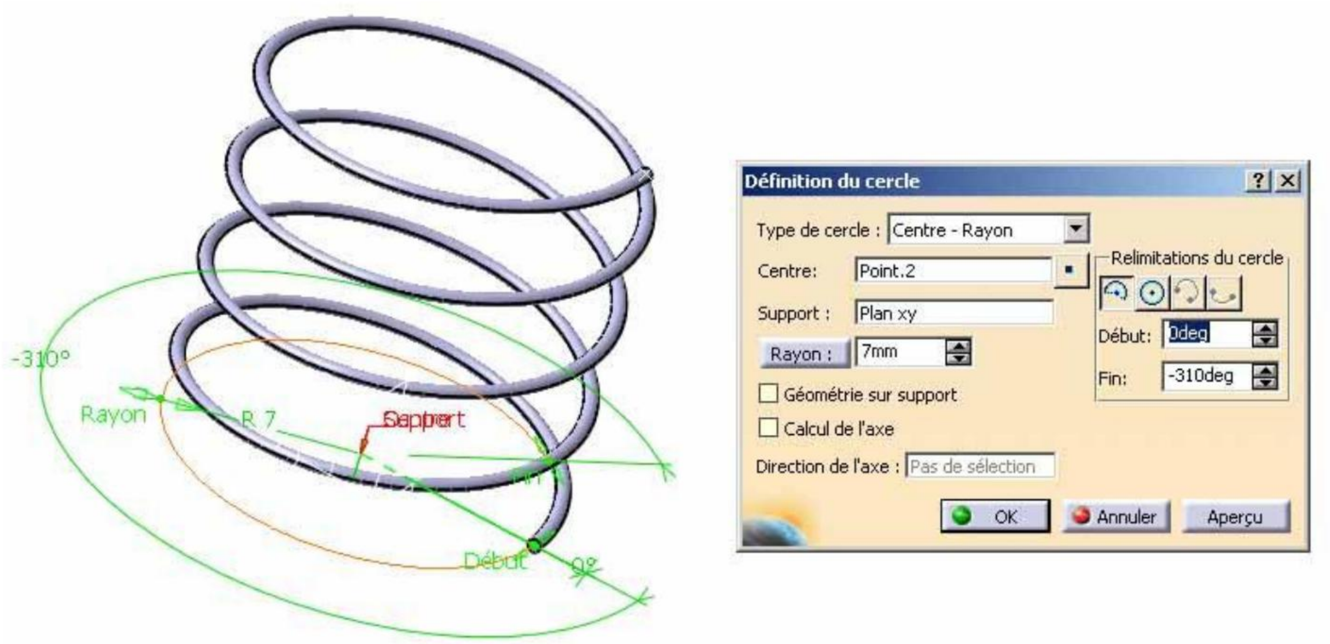
Center :

Center > Create the point

Coordinates : X=0mm
Y=0mm
Z=0mm

okay

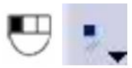
Support: XY plane
Radius: 7mm
Start : 0deg
End : -310deg

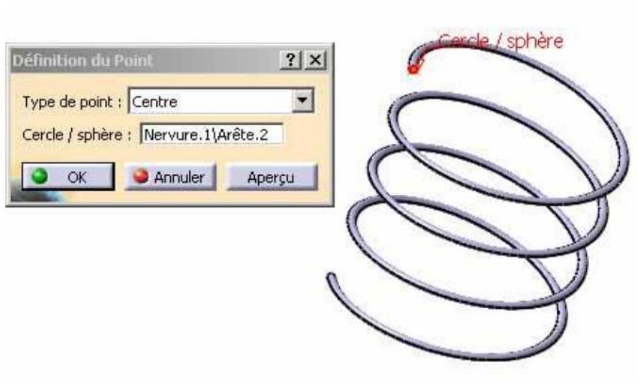


okay

To do the same on the upper part of the helix, you need two points and a plane, necessary for sketching the circle.

Creation of the first point:

 **POINT**

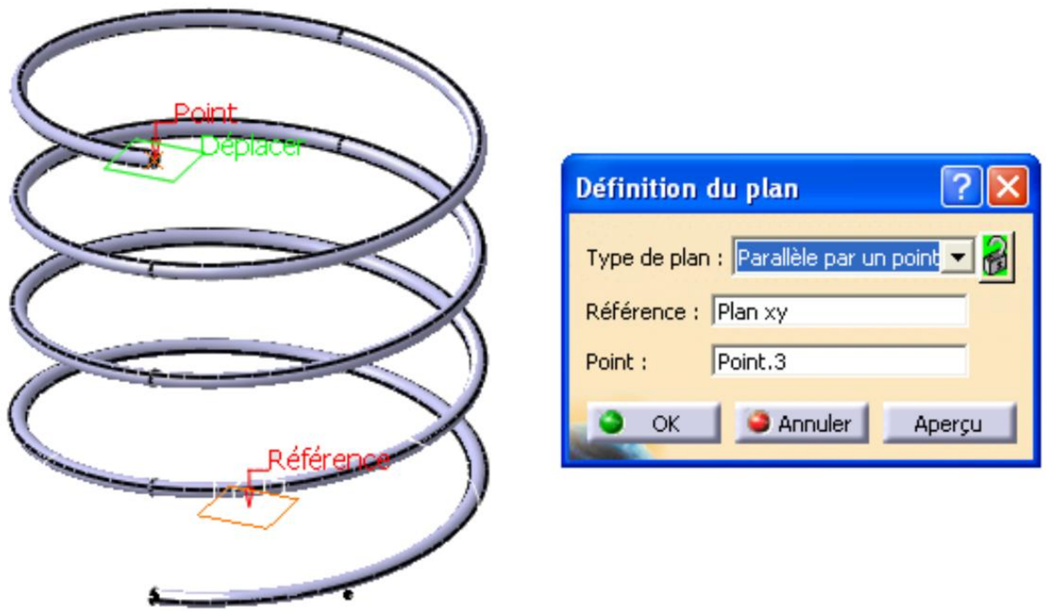


Stitch type: Circle center
Circle / sphere: Upper end of the helix

 **okay**

Plan creation:

 **PLAN**

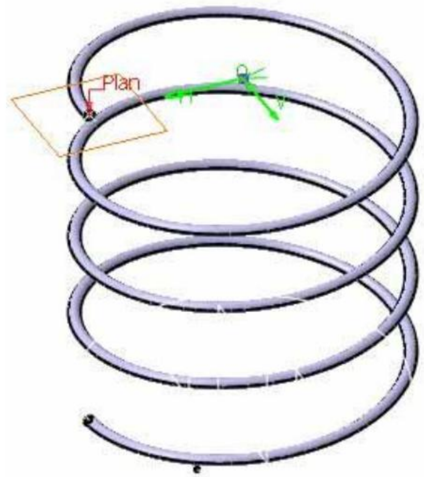


Type of plane: Parallel through a point **Reference:** XY plane **Point:** Point.3 (= last point created)

 **okay**

Creation of the second point (= center of the circle):

  **POINT**

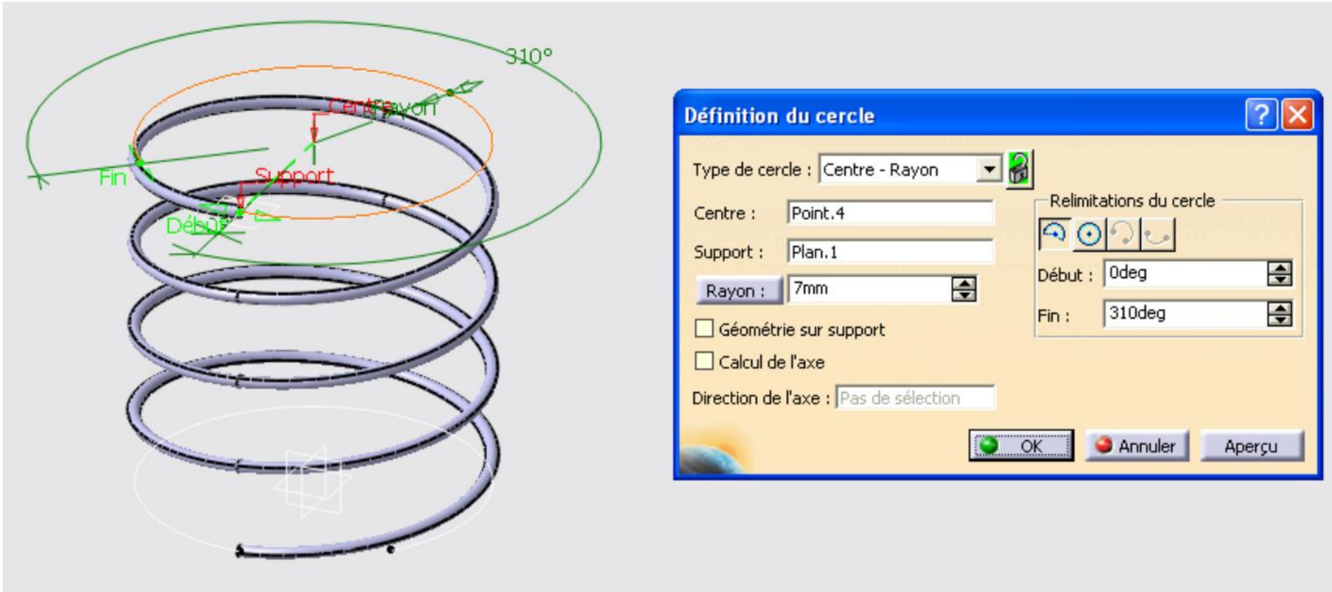


Point type: On plane
Plane: Plan.1 (= last plane created)
H= 0mm
V= 0mm

 **okay**

Creation of the circle:

  **CIRCLE**



Type of circle: Center – Radius
Center: Point.4 (=last point created)
Support: Plan.1 (= last plan created)
Radius: 7mm
Start: 0deg
End: 310deg

 **okay**

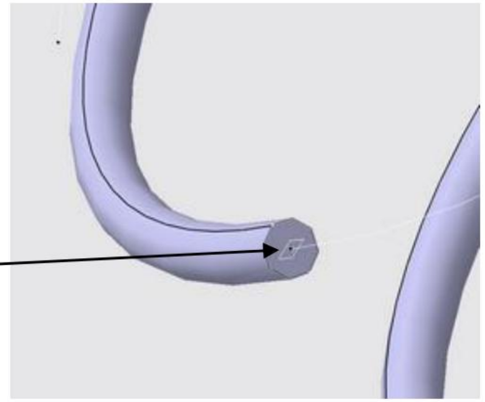
Start Menu > **Part Design**

RIB:

upper circular surface of the propeller **>** **SKETCH**

PROJECTION OF 3D ELEMENTS > upper circular surface of the propeller

LEAVING THE WORKSHOP

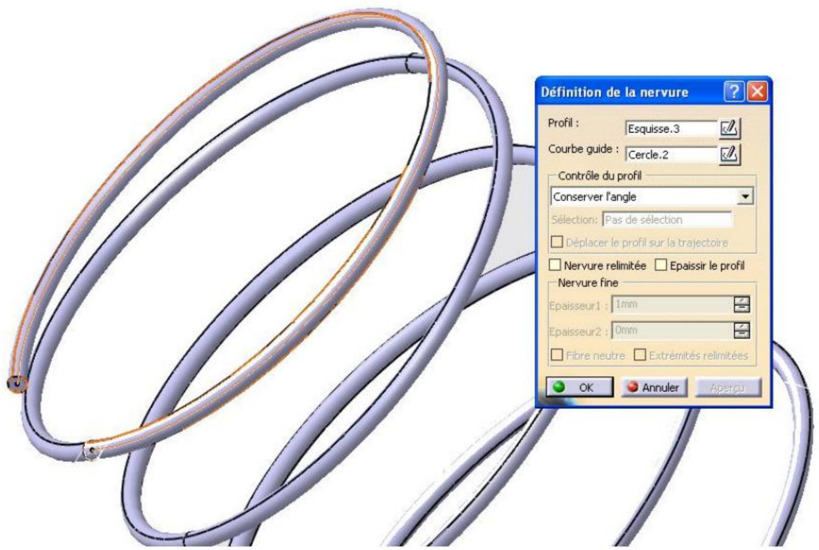


The sketch thus created appears in white, it resumes the outline of the upper circular surface of the propeller

Sketch.3 (last sketch created) > **RIB**

Profile: Sketch.3 (last sketch created)
Guide curve: Circle.2

okay



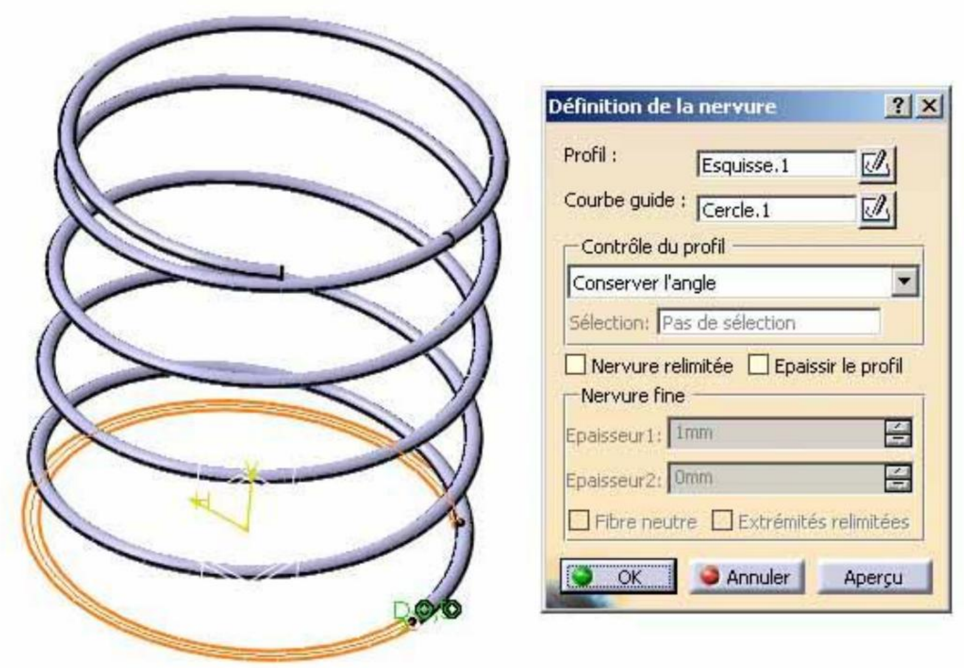
Do the same for the lower part of the spring:

Sketch.1 in tree >

RIB

Profile : Sketch.1
Guide Curve: Circle.1

okay

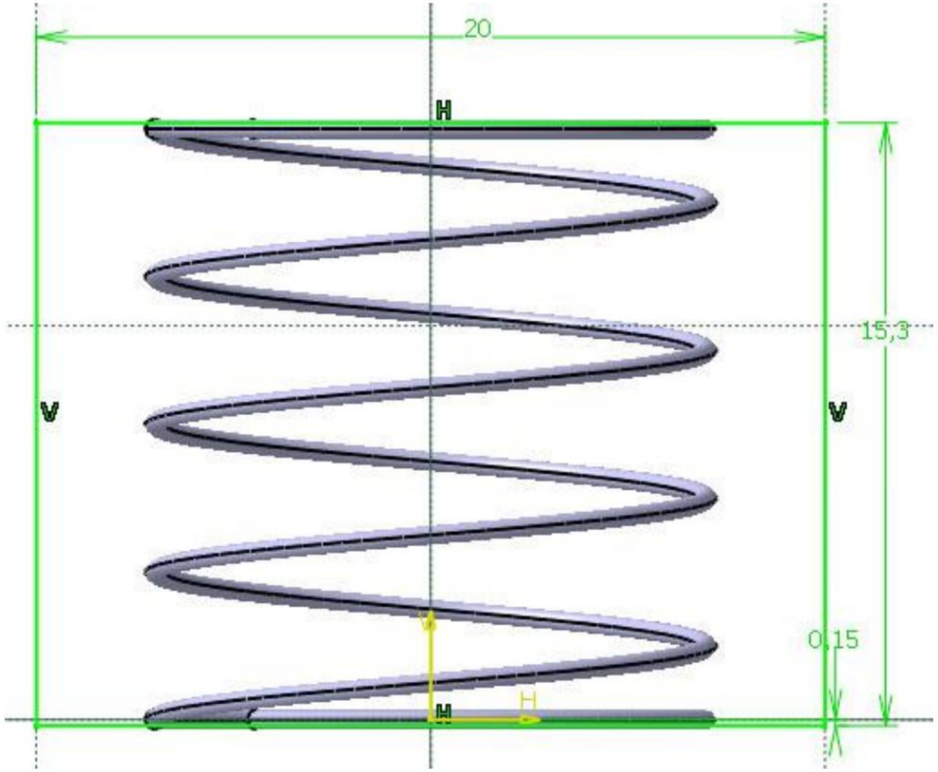


CHOPPED OFF :

 the **YZ** plan >  **SKETCH**

 **Rectangle >** according to the sketch opposite

 **LEAVING THE WORKSHOP**



POACHED

First limit:

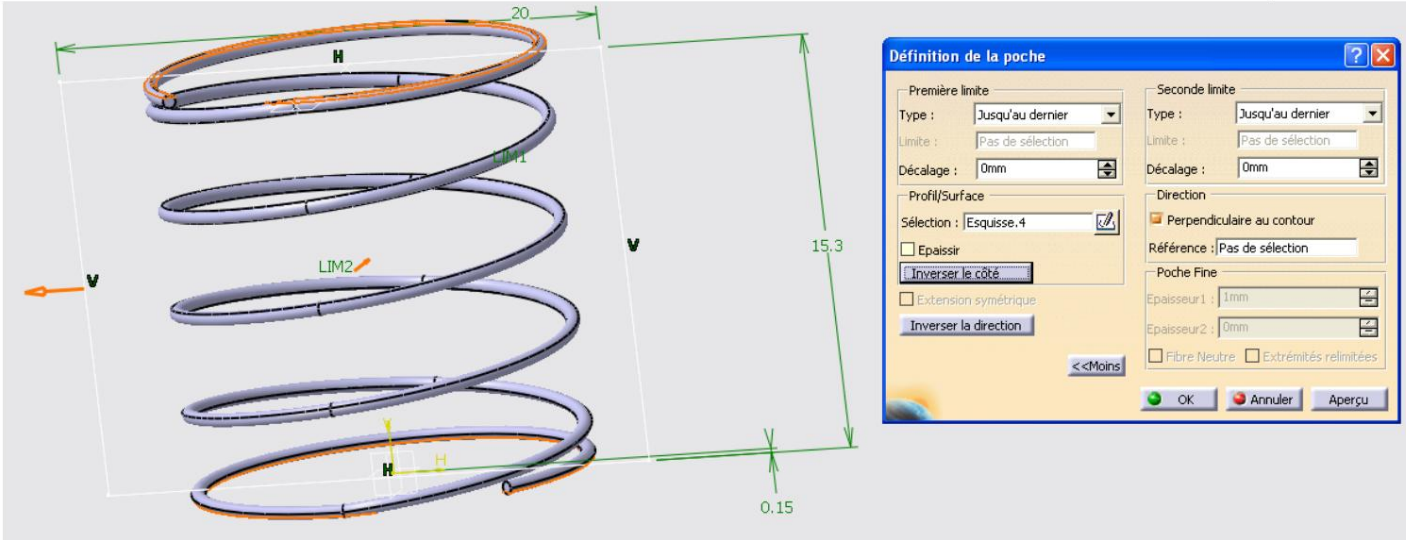
Type: to the last

More >>

Second Limit

Type: to the last

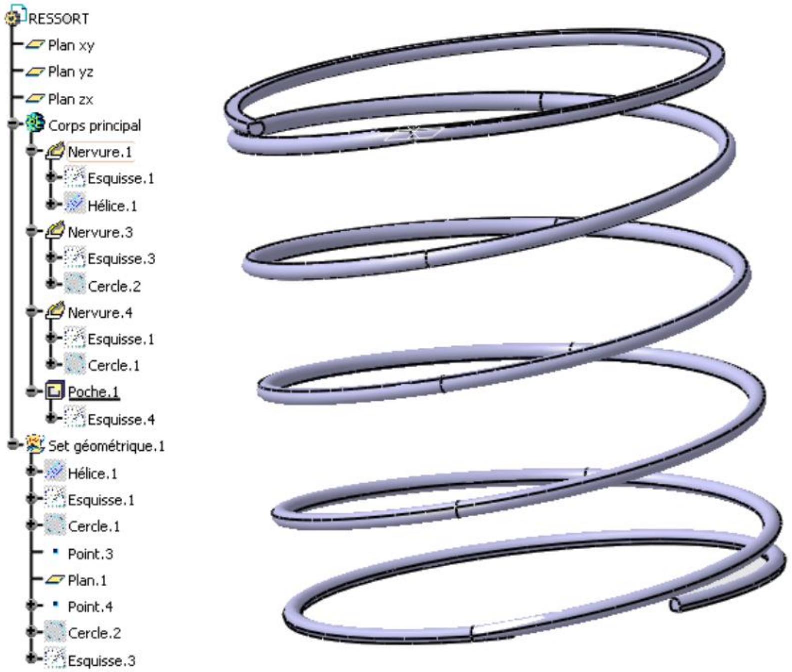
Selection: Sketch.4 (default)



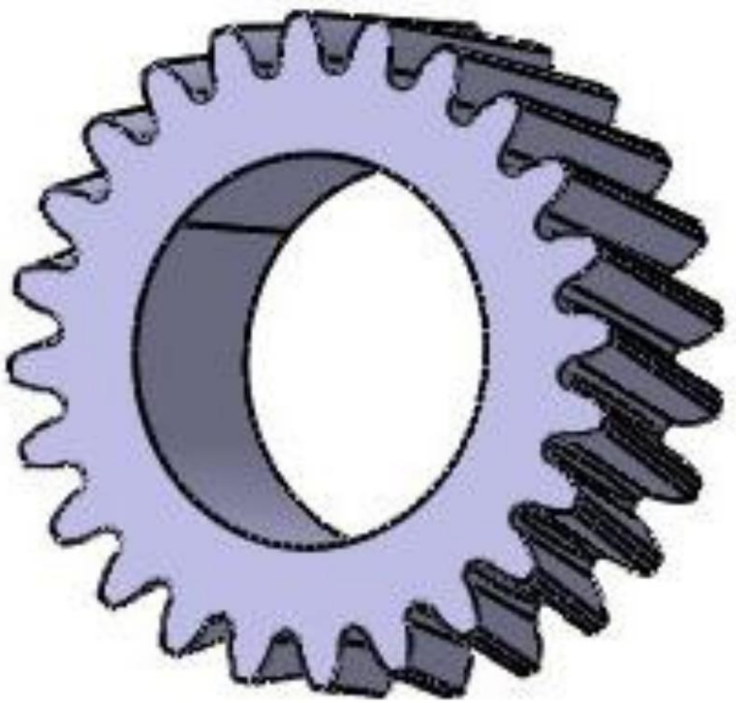
Reverse side

okay

File > Save



22. COMPLEMENTARY EXERCISE 2: SPRIGHT GEAR WITH HELICAL TOOTHING

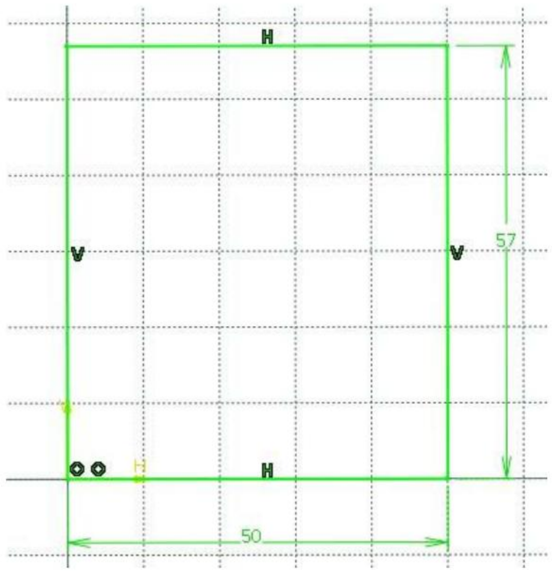


Technical characteristics: • type of tooth: spur gear with helical teeth • number of teeth Z=24 • Pitch Ø d=127.32mm • Pitch **pitch p=16.65 • Tooth width s=8.5mm •** Head **Ø da=137.7mm • Ø foot df=114mm • helix angle ÿ=11°**

FINAL RESULT

Start Menu > **Part Design** > enter part name: **SPROCKET** > **OK**
File menu > Save > **L: Catia** > file name: **SPROCKET** > **Save**

CYLINDER



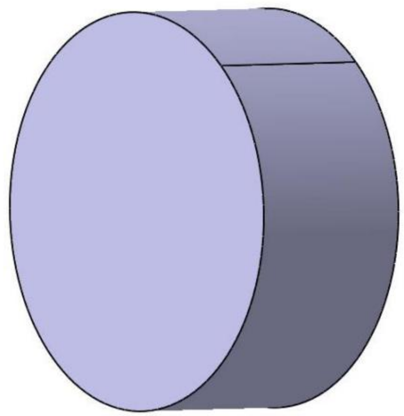
the **ZX** plane in the tree structure > **SKETCH**

Draw the sketch according to the figure opposite

LEAVING THE WORKSHOP

REVOLUTION
Axis > Selection: horizontal axis **H**

okay



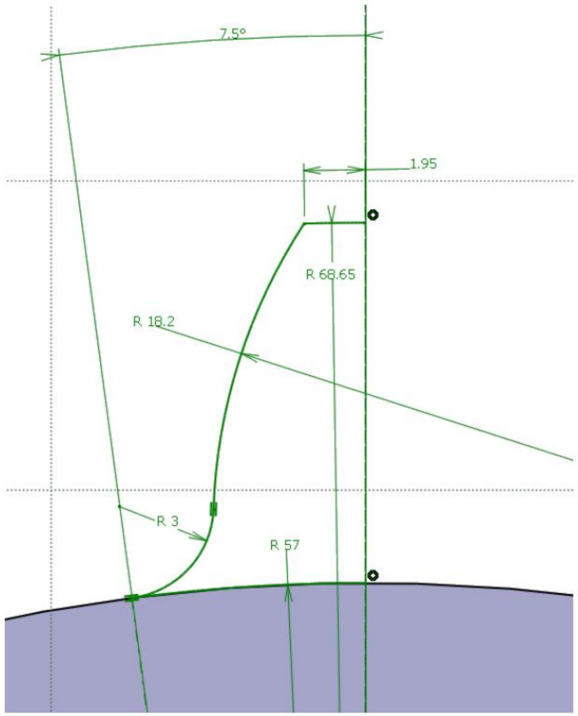
TOOTH

One of the faces of the cylinder > **SKETCH >** Draw the sketch of the tooth (according to the figures below)

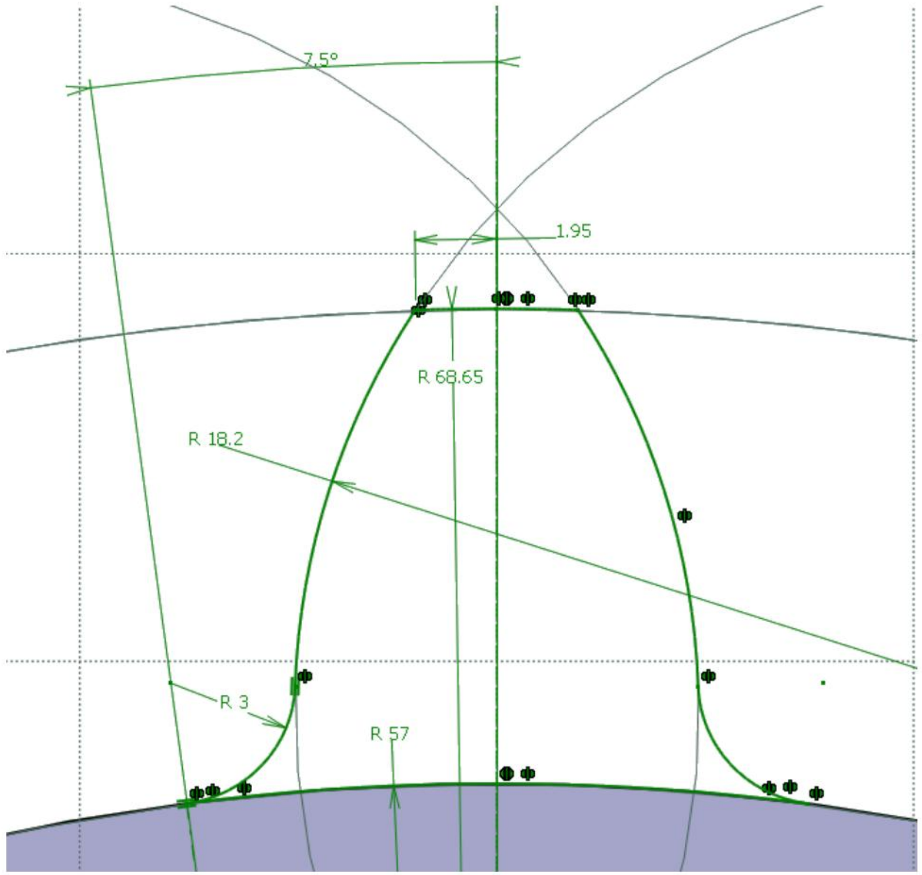
Method :

1. draw the left half of the tooth 2. create a vertical axis coinciding with the V axis
3. mirror this axis

Note: the tooth profile proposed here is an approximation of a real profile (circle involutes)



SYMMETRY



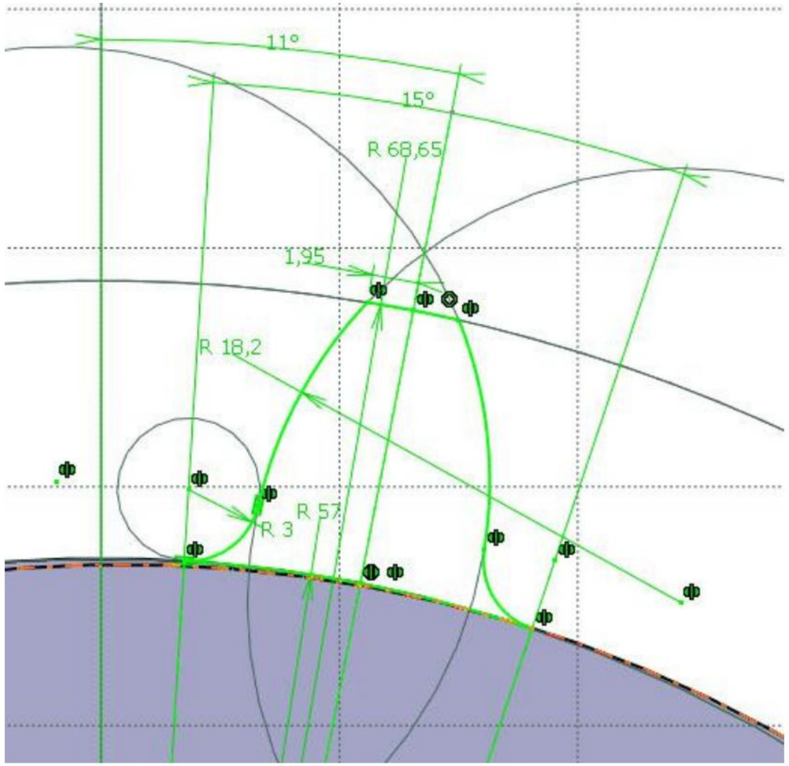
LEAVING THE WORKSHOP

COPY OF TOOTH SKETCH:

> Sketch.2 (in tree view) > **CTRL + C** > the second side of the cylinder > **CTRL + V**

Note: A second sketch identical to the first appears on the second face of the cylinder.

> 2x Sketch.3 (in tree view)



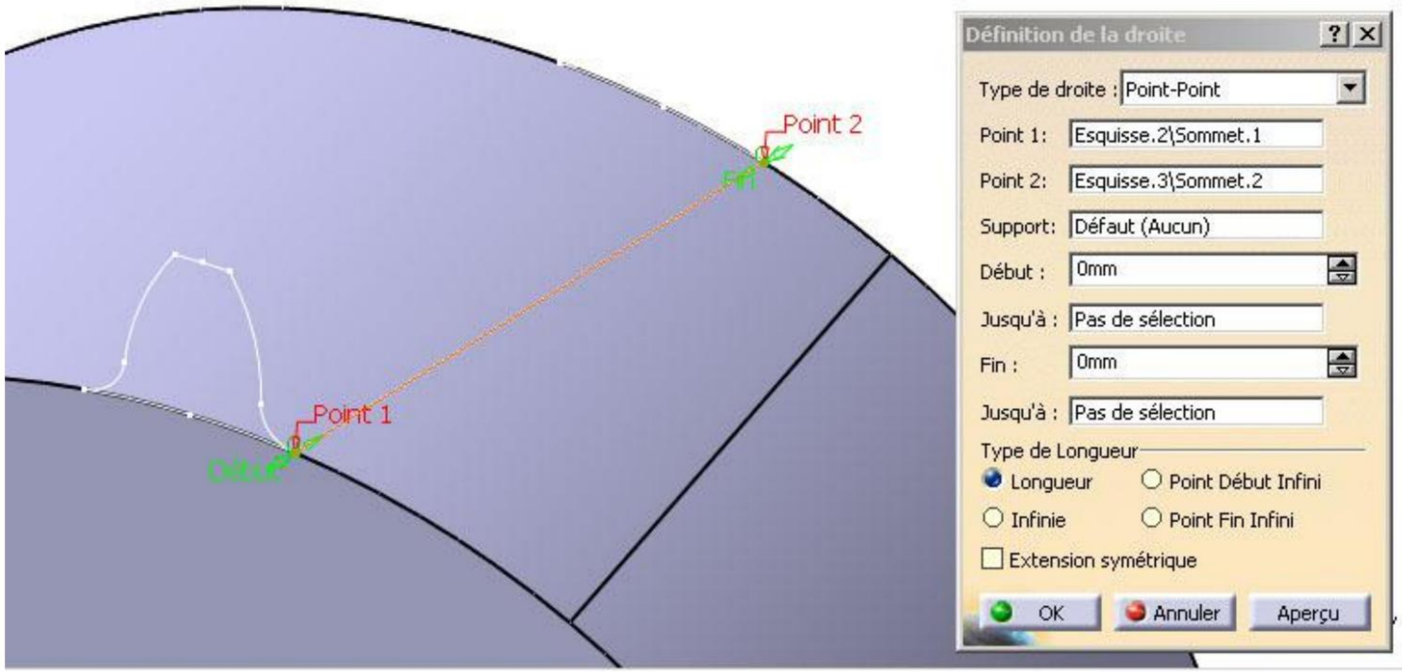
Assistance :

- Remove the verticality of the axis of symmetry, give an angle of 11° with respect to the axis V.
- Beware of automatic constraints that fix points on the V axis: delete • Check that the sketch is entirely constraint: in green

LEAVING THE WORKSHOP

CREATING A GUIDE

 LAW



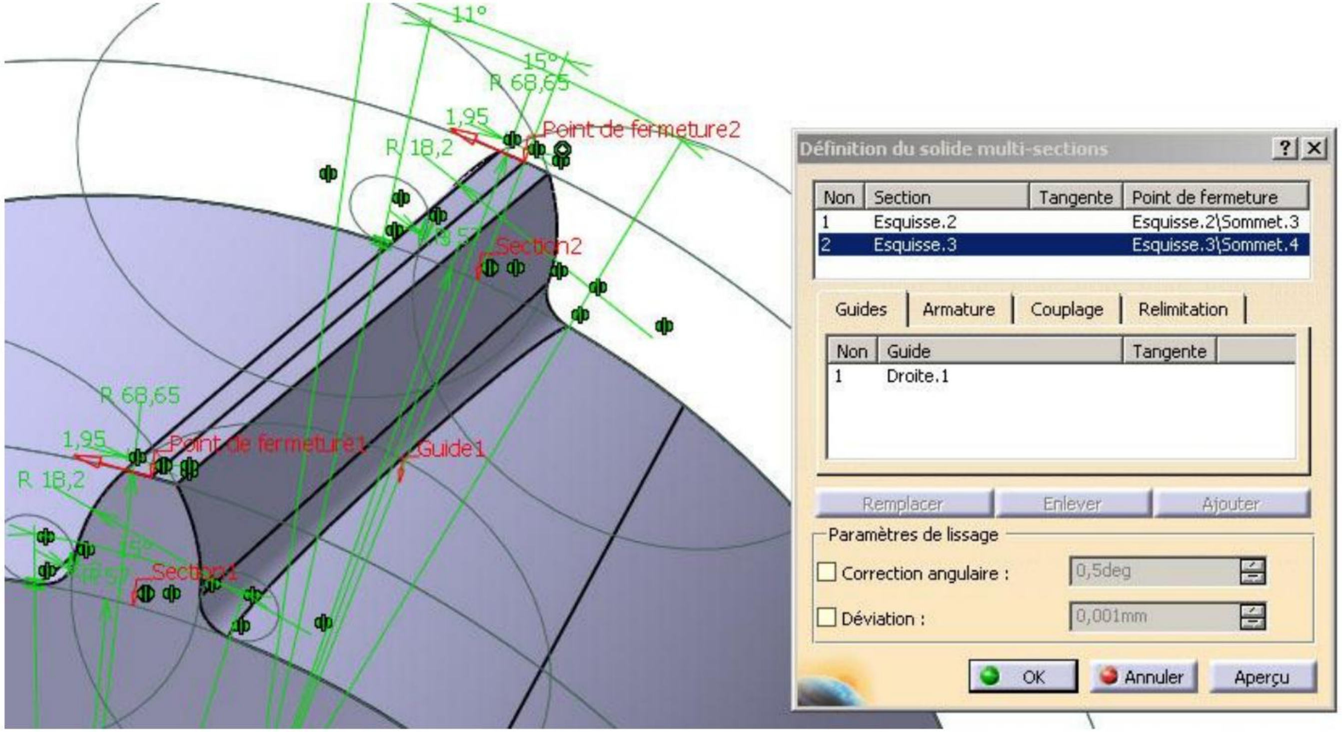
 **okay**

CREATING A MULTI-SECTION SOLID

SOLID MULTI-SECTIONS

Sketch.2 > Sketch.3

guide > Right.1



Note 1: Check that closing point 1 is opposite closing point 2. If not, follow the next procedure.

closing point2 > to replace

"the point of sketch.3 being in front of the closing point1, on on sketch.2"

Note 2: Check the arrow direction of both sketches, at the closing point. The two arrows should point to the same direction. If not > an arrow (it changes direction)

>OK

CIRCULAR REPEAT

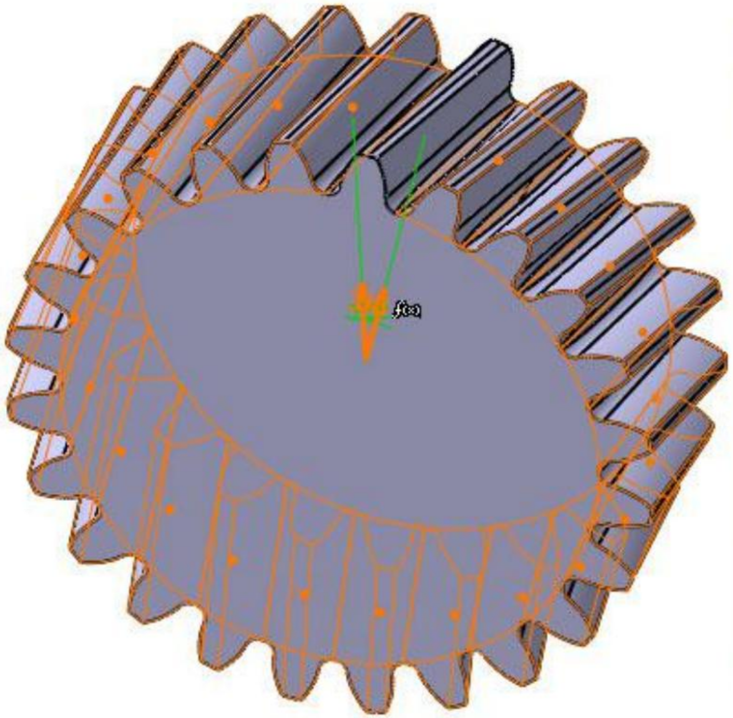
CIRCULAR REPEAT

Reference element > one side of the cylinder

Component > Multi-section solid

Parameters: full crown

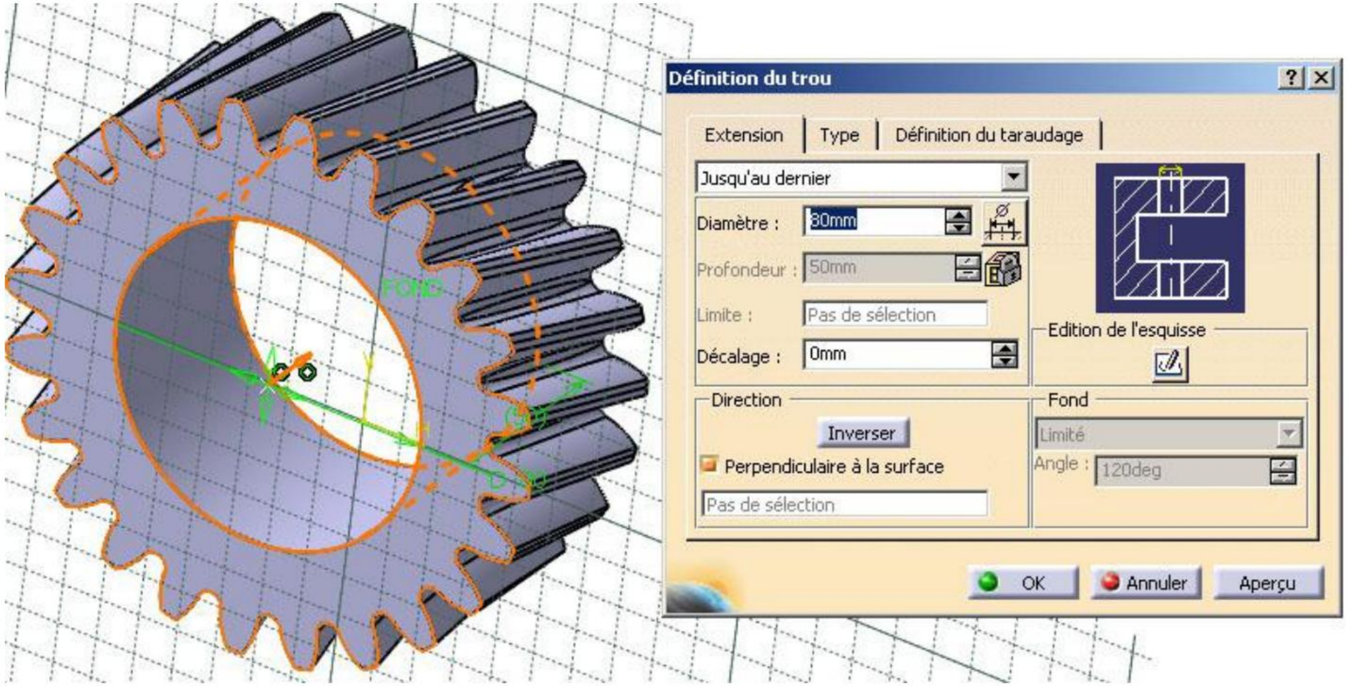
Instances: 24



>OK

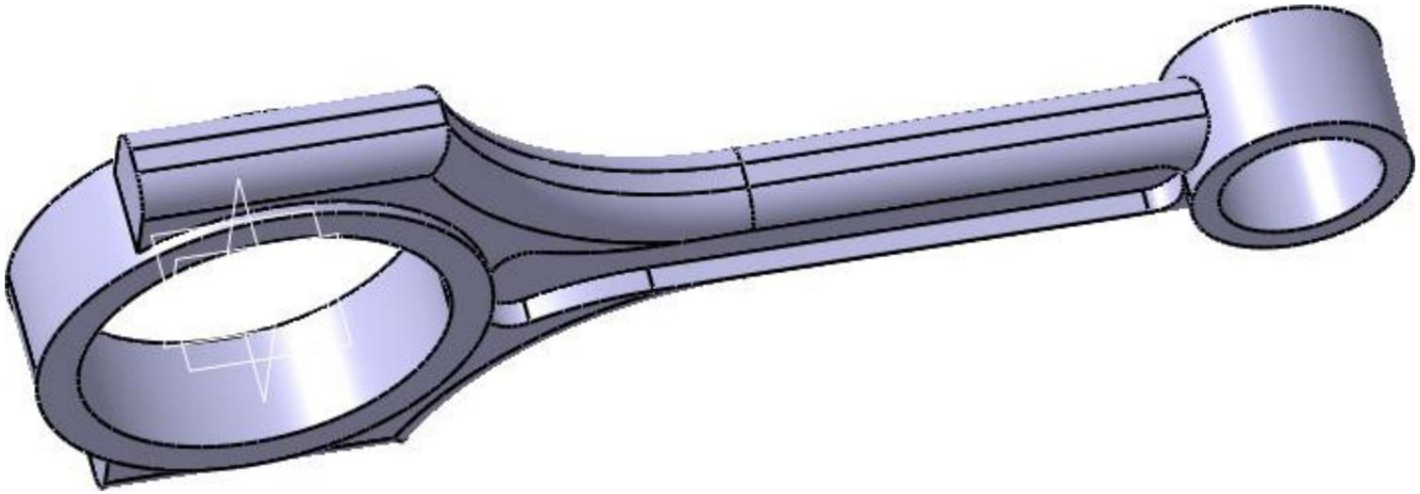
HOLE

Make a through hole, diameter 80 mm, centered on the pinion.



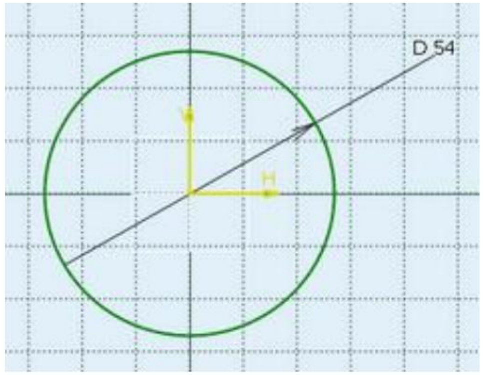
File > Save

23. COMPLEMENTARY EXERCISE N°3: CONNECTING ROD



This exercise aims to present a new approach and tools. The part in question can be obtained with all the tools seen previously.

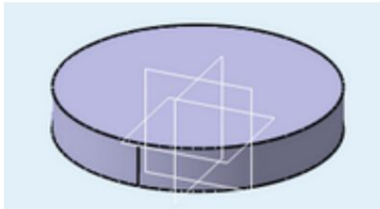
Start menu > **Part design** > Enter the name of the part: **ROD** > OK
File menu > Save > (default location, or as indicated) > file name: **BIELLE** > **Save**



In the **XY plane,** draw a circle of diameter 54.

  **LEAVING THE WORKSHOP**

  **EXTRUSION >** Type: Length > enter **9** > OK

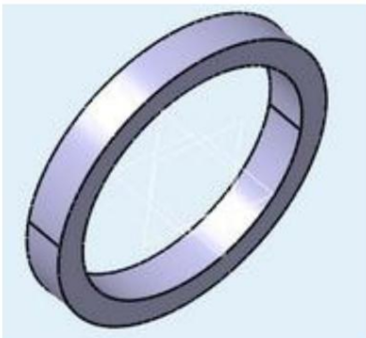


Select the two plane faces of the cylinder:

 on a face > keep the **Ctrl** key pressed > (the two selected elements turn red) > release the **Ctrl key.**

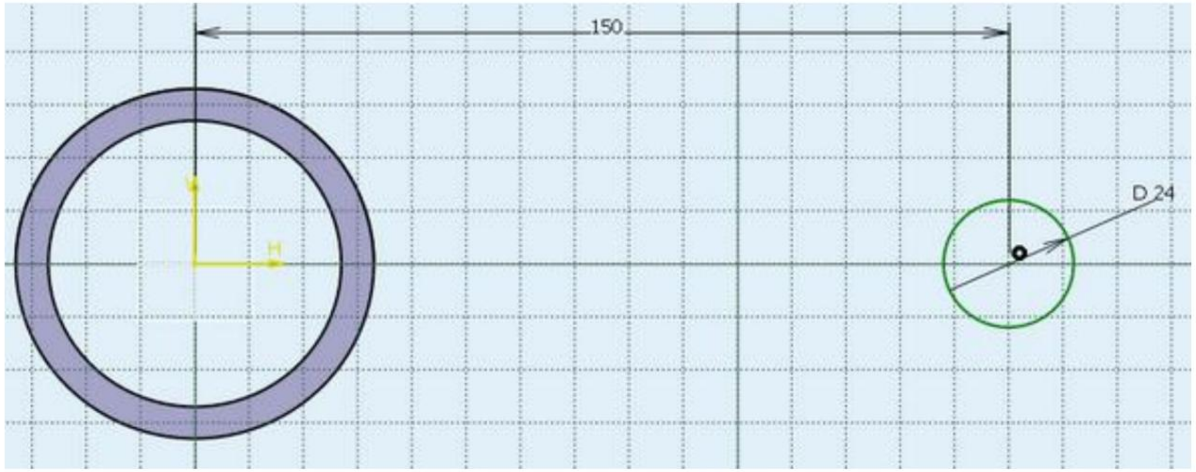
 on the other side

  **HULL** > Interior thickness: enter **0**
Outside thickness: enter **6**
>OK



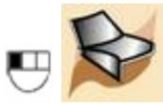
Insert menu > Body (a new Body appears in the tree view)

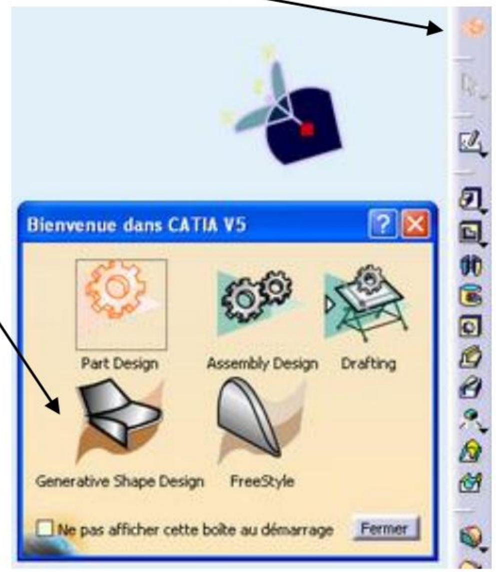
In the **XY plane,** draw a circle of diameter **24,** the center on the H axis, located **150** mm from the center of the first circle.



 **LEAVING THE WORKSHOP**

After making a 9mm extrusion from this sketch, As before, create a shell with interior thickness **0** and exterior thickness 4.

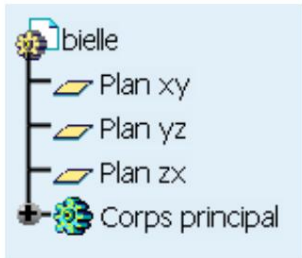
 (at the very top of the list of buttons on the right) >  **(Generative Shape Design)**



In the list of buttons on the right, find and click on the **ADD icon Add :**  select Part body.2 in the tree structure **A :** Main body (by default)

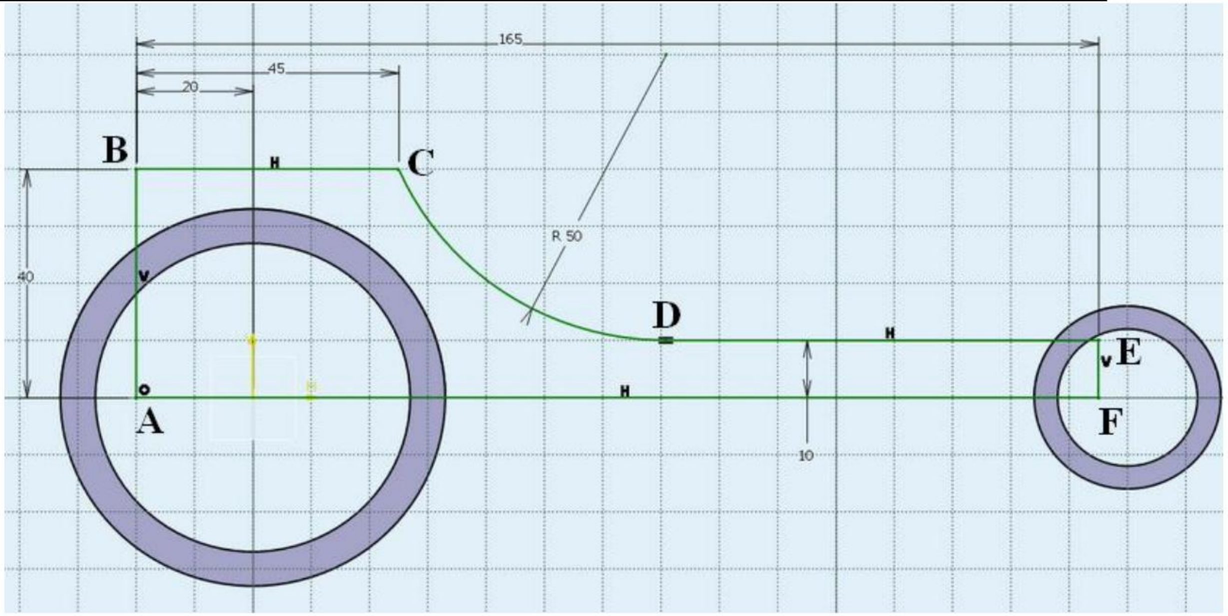
After : Shell.1 (default)
>OK

The tree then appears as shown in the figure (Part Body.2 has disappeared)



 (at the very top of the list of buttons on the right) >  **(Part Design)**

CREATION OF THE CONNECTING ROD BODY



Insert menu > Part Body (Part Body.3 appears in the tree view)
In the **XY plane,** draw the profile as it is in the figure
The **AF** segment is coincident with the H axis.

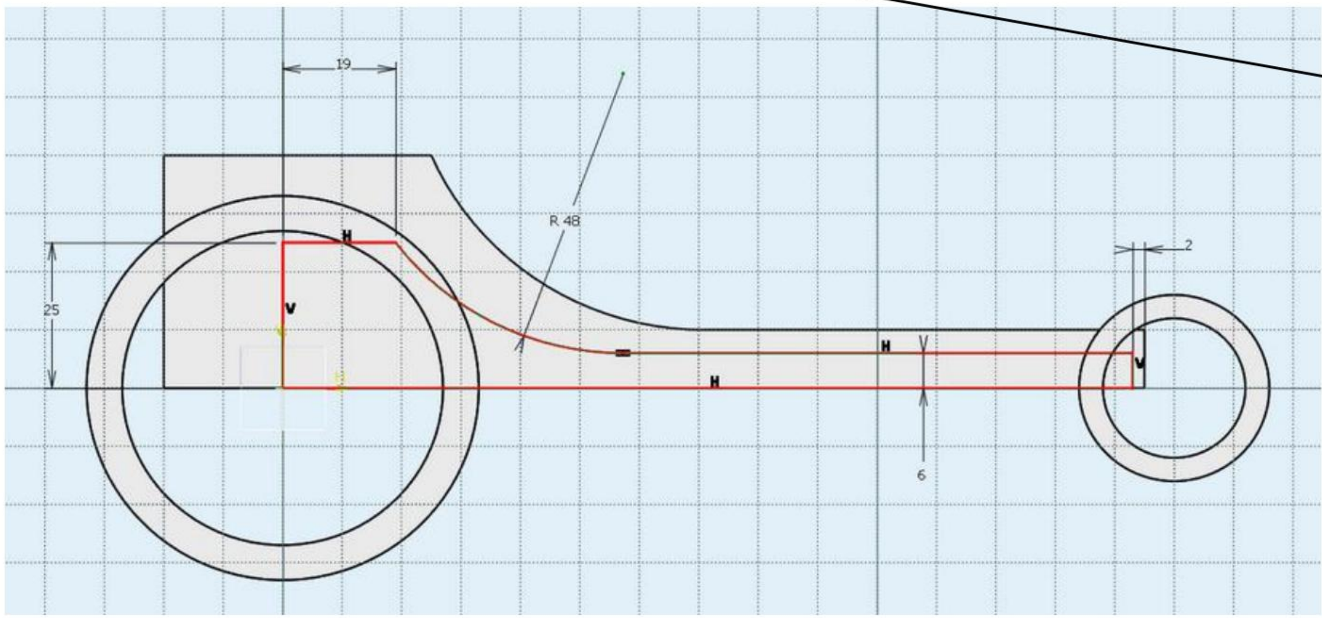
Arc **CD** is tangent to segment **DE.**

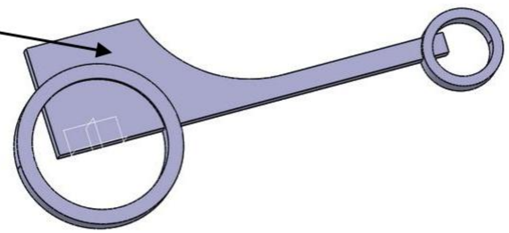
LEAVING THE WORKSHOP

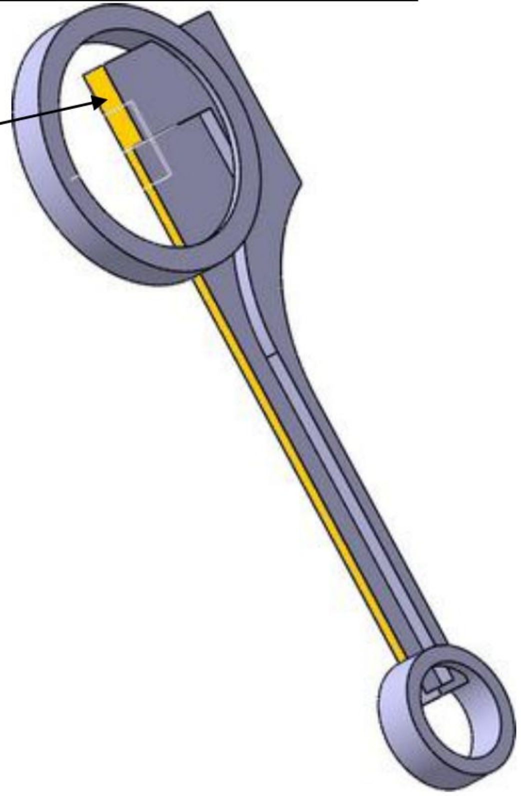
EXTRUSION > Type: Length > enter **7** > OK

upper face of the created part

SKETCH and Draw the profile below:





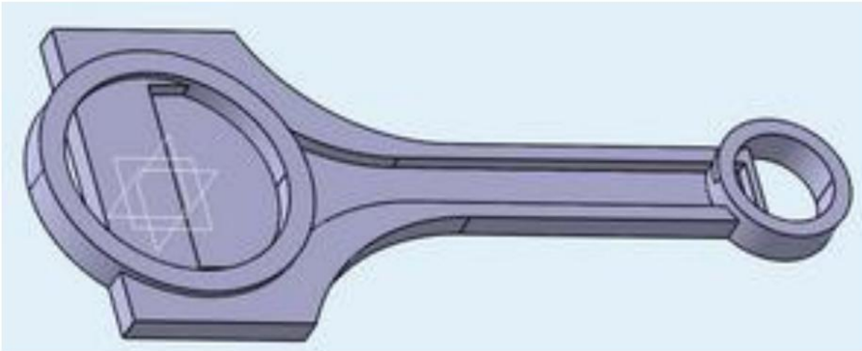


LEAVING THE WORKSHOP

POCKET > Length: 4, Selection: Sketch.4 (default) > OK

next to the room

SYMMETRY > plane of symmetry
>OK



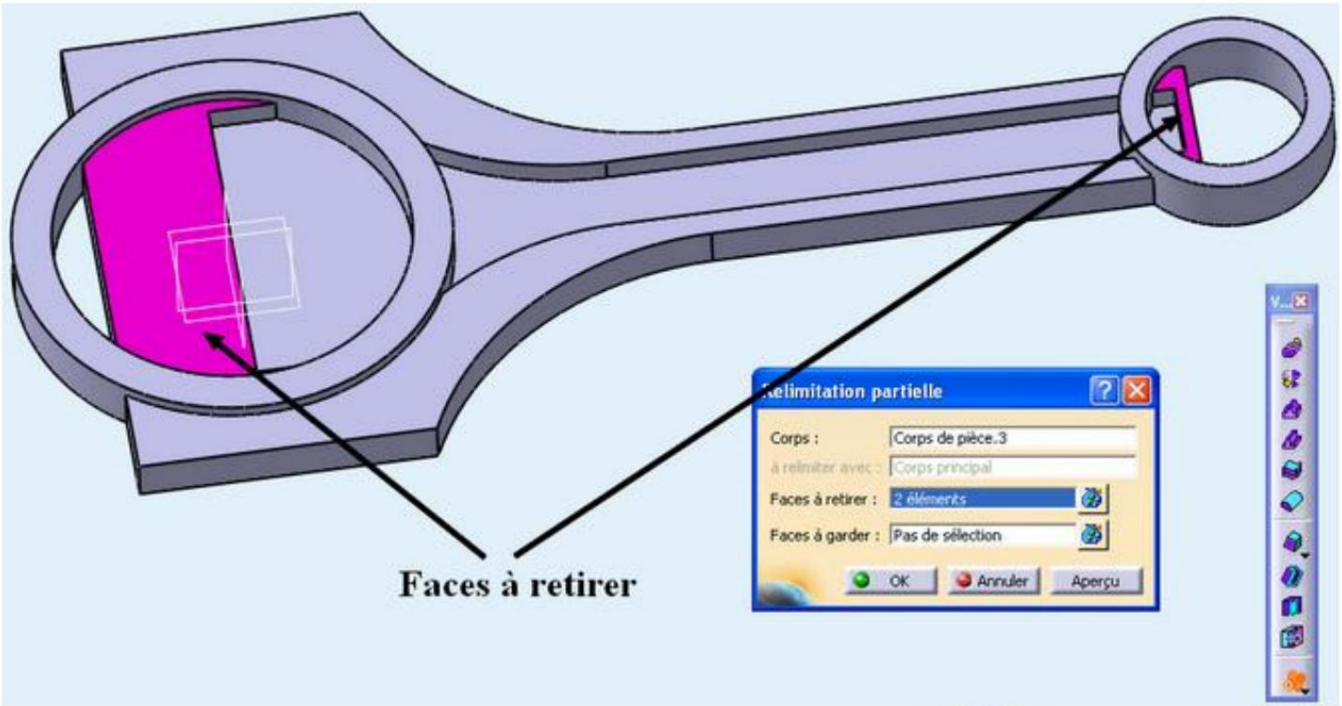
RELIMITATION OF THE ROOM

(at the very top of the list of buttons on the right) > **(Generative Shape Design)**

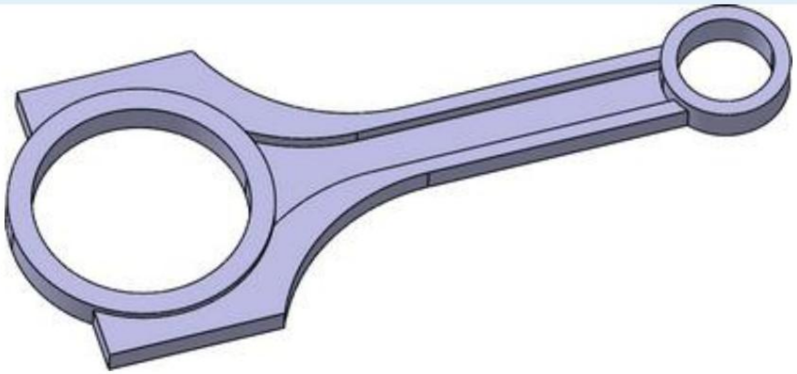
Part body.3 (in tree view)

PARTIAL RESTRICTION

Sides to remove :  sides shown below



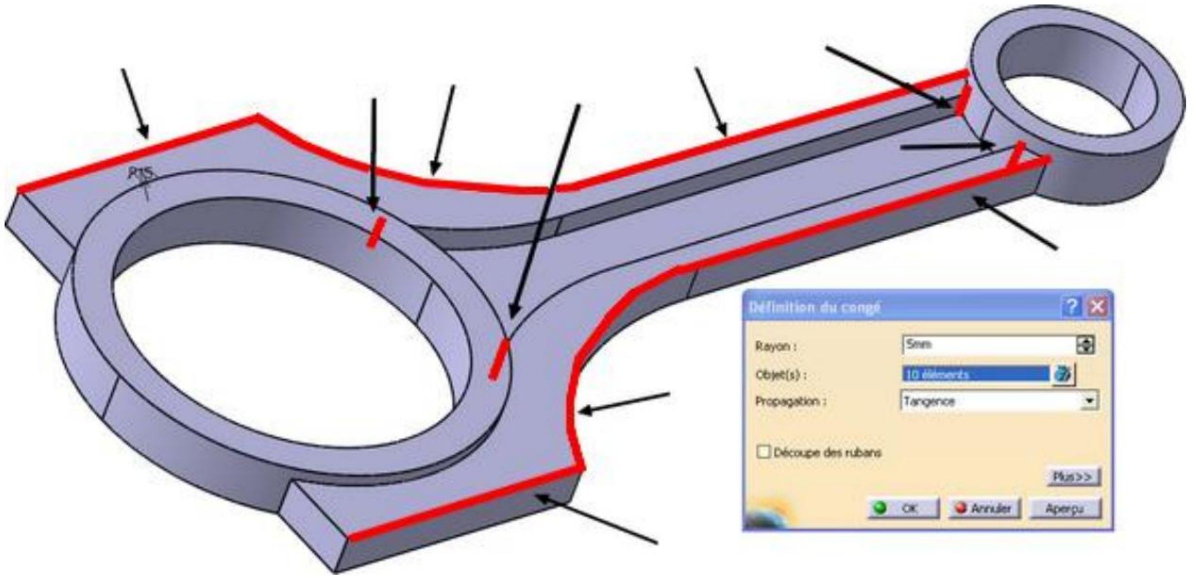
>OK

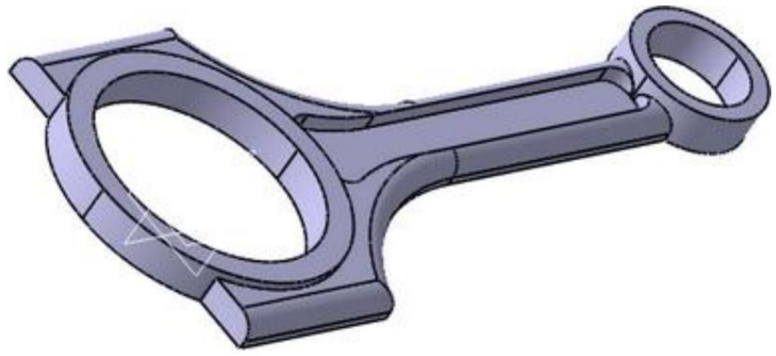


REALIZATION OF EDGE FILLES

(at the very top of the list of buttons on the right) > **(Part Design)**

> the 10 edges shown in the figure below (while holding down the Ctrl key) > Radius: **5** > OK

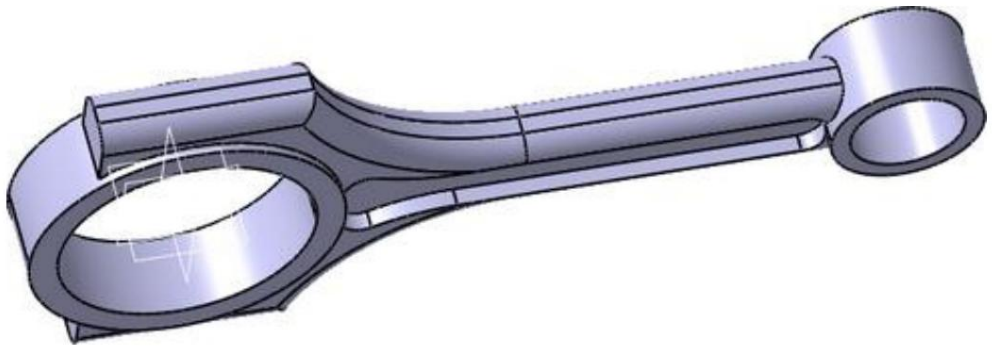




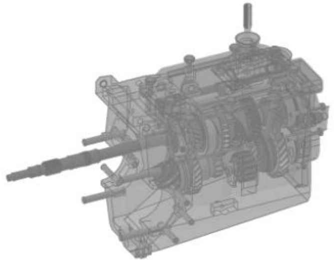
END OF CONNECTING ROD

SYMMETRY > symmetry plane (XY **plane)** > OK

File menu > Save



Guidelines for managing Catia assemblies

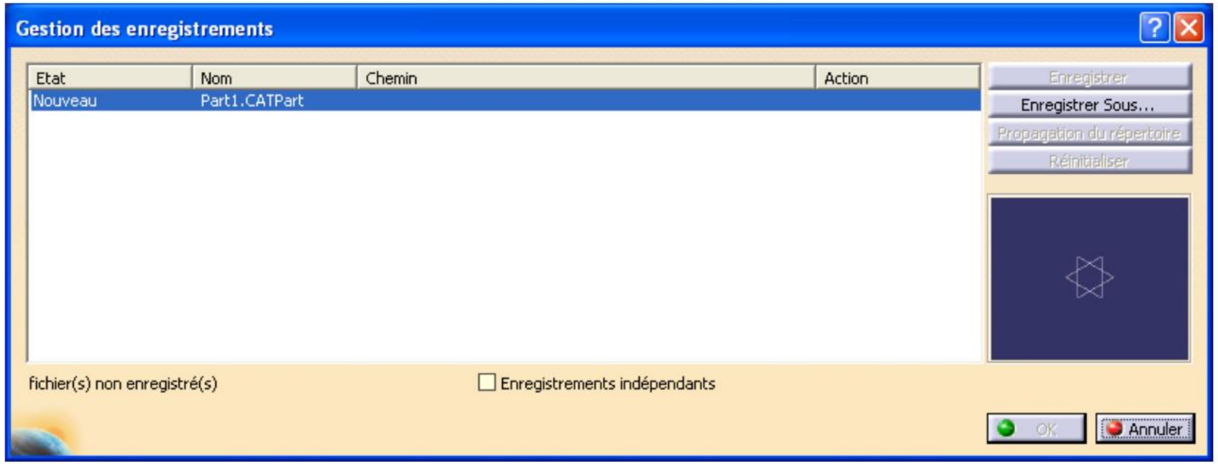


24. File naming

Two files cannot have the same name. We advise you to follow the name of the part with your initials and a number. Example: axis_MB_009

Do not rename files.
Do not use accents and special characters in file names.

25. Backing up files Backing up
files is done with the recording manager : *File / Recording management*



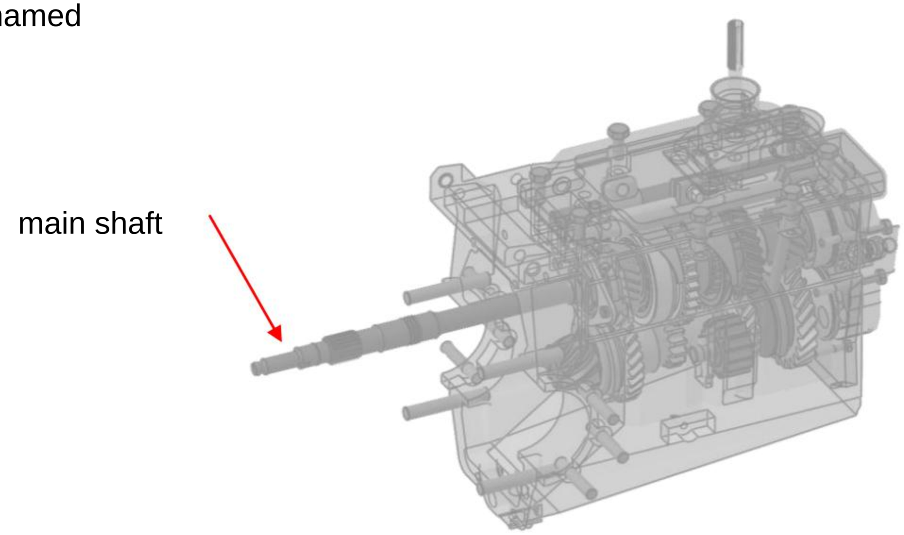
It is strongly recommended to save all the files in your "Permanent Data" (also with parts imported from TracePart or other).

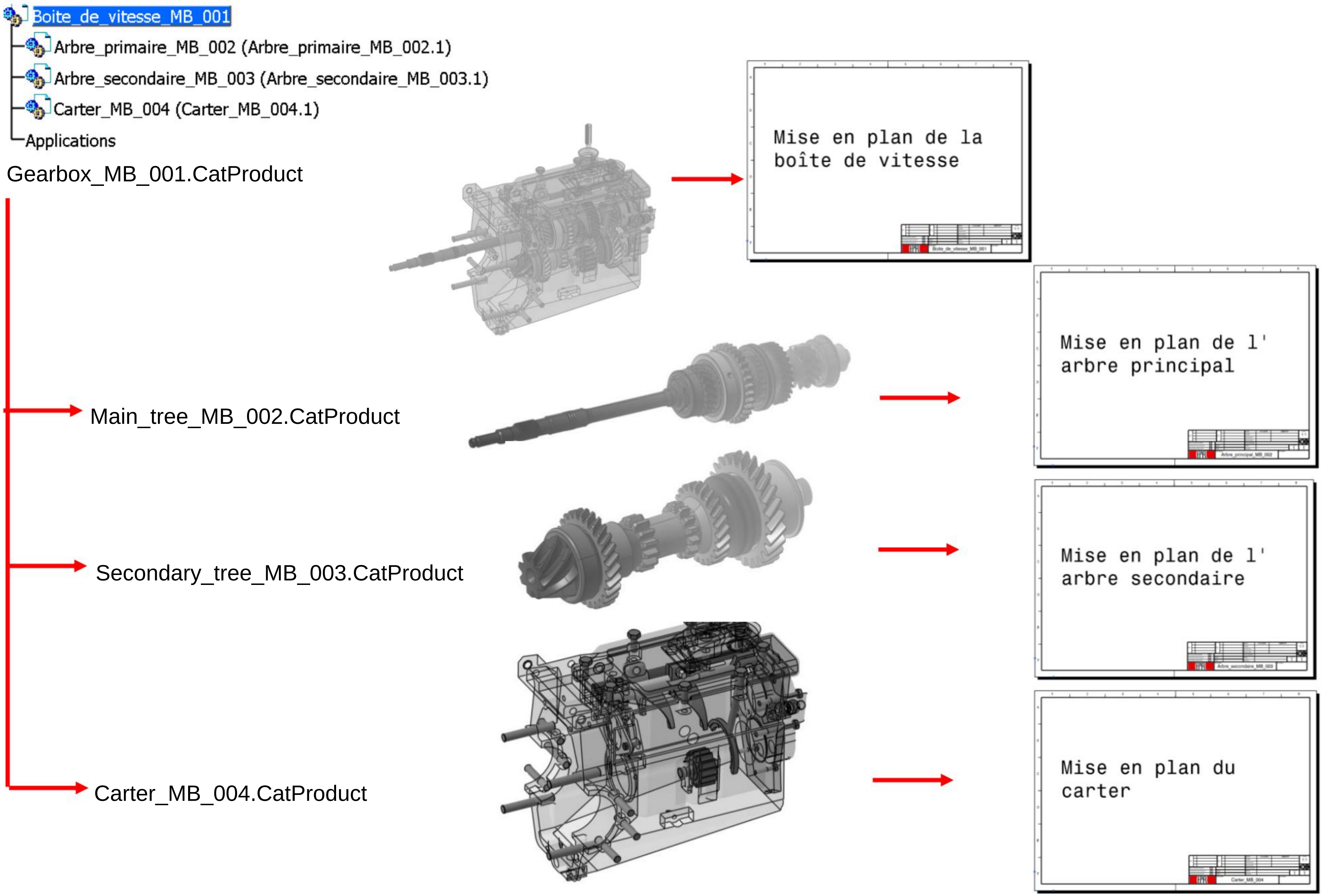
26. Structure of assemblies A
sub-assembly must be created if it can be assembled independently by an assembler on the shop floor.
For each 3D assembly, a 2D drawing must be made with a BOM and parts list.

27. Example of an assembly containing several sub-assemblies A
gearbox is made up of a primary shaft, a secondary shaft, a casing and accessories (screws, lever, oil cap, seals, etc.) .

The assembly structure will be as follows: 1 main
assembly named Boite_de_vitesses_MB_001.CatProduct containing the following sub-assemblies:
ÿ 1 sub-assembly named Main_shaft_MB_002.CatProduct containing all the main shaft parts ÿ 1 sub-assembly named Secondary_Shaft_MB_003.CatProduct containing all the secondary shaft parts ÿ 1 sub-assembly named Carter_MB_004.CatProduct containing the crankcase and all the accessories

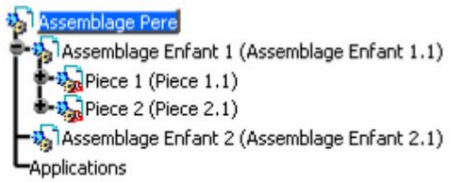
A drawing will be made for each assembly and sub-assembly, namely: ÿ 1
drawing named Boite_de_vitesses_MB_001.CatDrawing ÿ 1 drawing named Arbre_principal_MB_002.CatDrawing ÿ 1 drawing named Arbre_secaire_MB_003.CatDrawing ÿ 1 drawing named Carter_MB_004 .CatDrawing



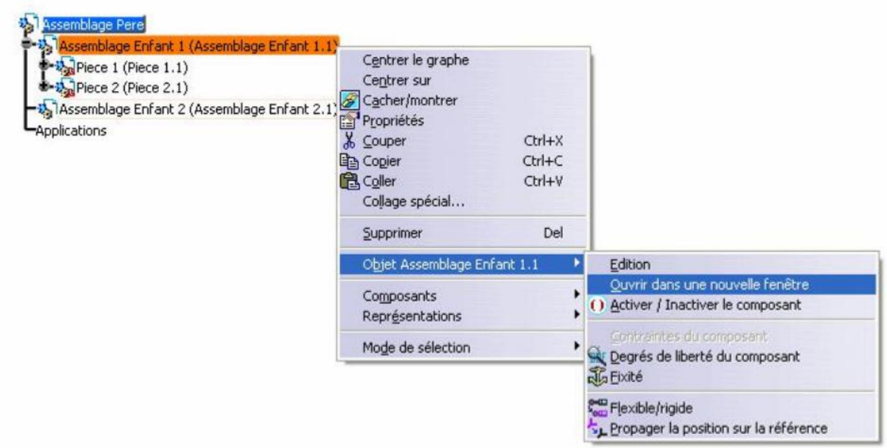

Boite_de_vitesse_MB_001
Arbre_primaire_MB_002 (Arbre_primaire_MB_002.1)
Arbre_secondaire_MB_003 (Arbre_secondaire_MB_003.1)
Carter_MB_004 (Carter_MB_004.1)
Applications
Gearbox_MB_001.CatProduct
Main_tree_MB_002.CatProduct
Secondary_tree_MB_003.CatProduct
Carter_MB_004.CatProduct
Mise en plan de la boîte de vitesse
Mise en plan de l' arbre principal
Mise en plan de l' arbre secondaire
Mise en plan du carter

28. Example of drawing of an assembly and its sub-assemblies For the links between the documents to be coherent, the drawing of a sub-assembly must be done when this assembly is opened in an independent window.

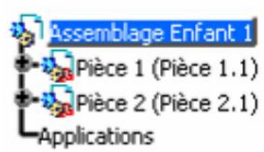
In the following case, the drawing of *Child Assembly 1* requires it to be opened in a new window.



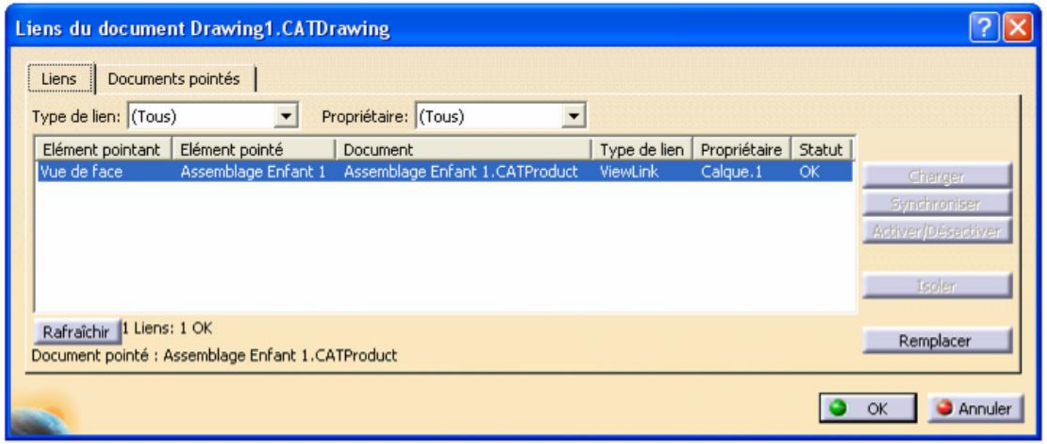
When *Parent Assembly* is enabled, right click on *Child Assembly* 1, then *Child Assembly Object 1.1,* and *Open in New Window.*



Sub-assembly drawing can be done when the top-level item in the specification tree is the sub-assembly.

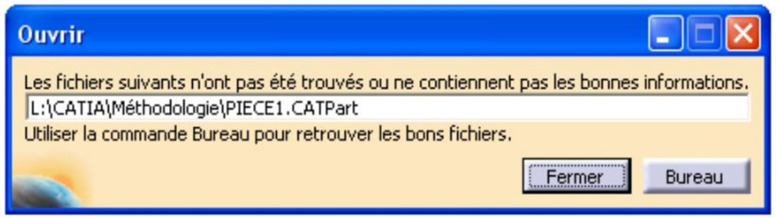


In order to check that the links are correct, in the drawing, go to *Edit / Links.* The pointed element must be the sub-assembly and not the parent assembly.



29. Example of replacing a part by another What does not work

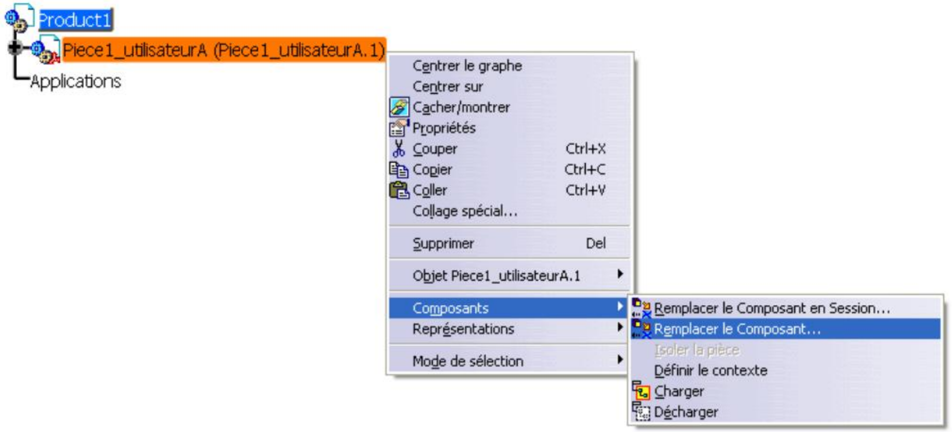
<u>(except in special cases): A user A creates an as</u>sembly in which is located a part that he has named Piece1 (Piece1.CATPart).
A user B creates a part which he names Piece1 (Piece1.CATPart).
User A would like to recover Piece1 from user B… He therefore copies the file and overwrites his Piece1.CATPart file.
When opening his assembly, he gets the following message !!! Indeed, the name is identical but the UID is different. (For more information, refer to paragraph 8: Managing file names)



What works: A

user A creates an assembly containing a part that he named Piece1_userA (Piece1_userA.CATPart).

User B creates a part that he names Piece1_userB (Piece1_userB.CATPart).
User A would like to retrieve User B's Piece1… He therefore copies the Piece1_userB.CATPart file into his directory, opens his assembly and by right-clicking on Piece1_userA (see illustration)



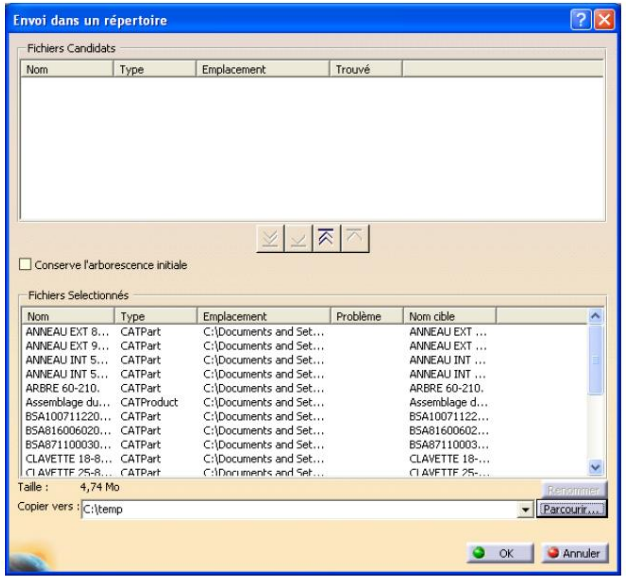
He selects the Room1_userB.CATPart file and clicks OK in the next window.

The instance name has not changed and is no longer consistent. (cf. Managing file names)

It must be changed manually by editing the properties of each instance of Piece1_userB

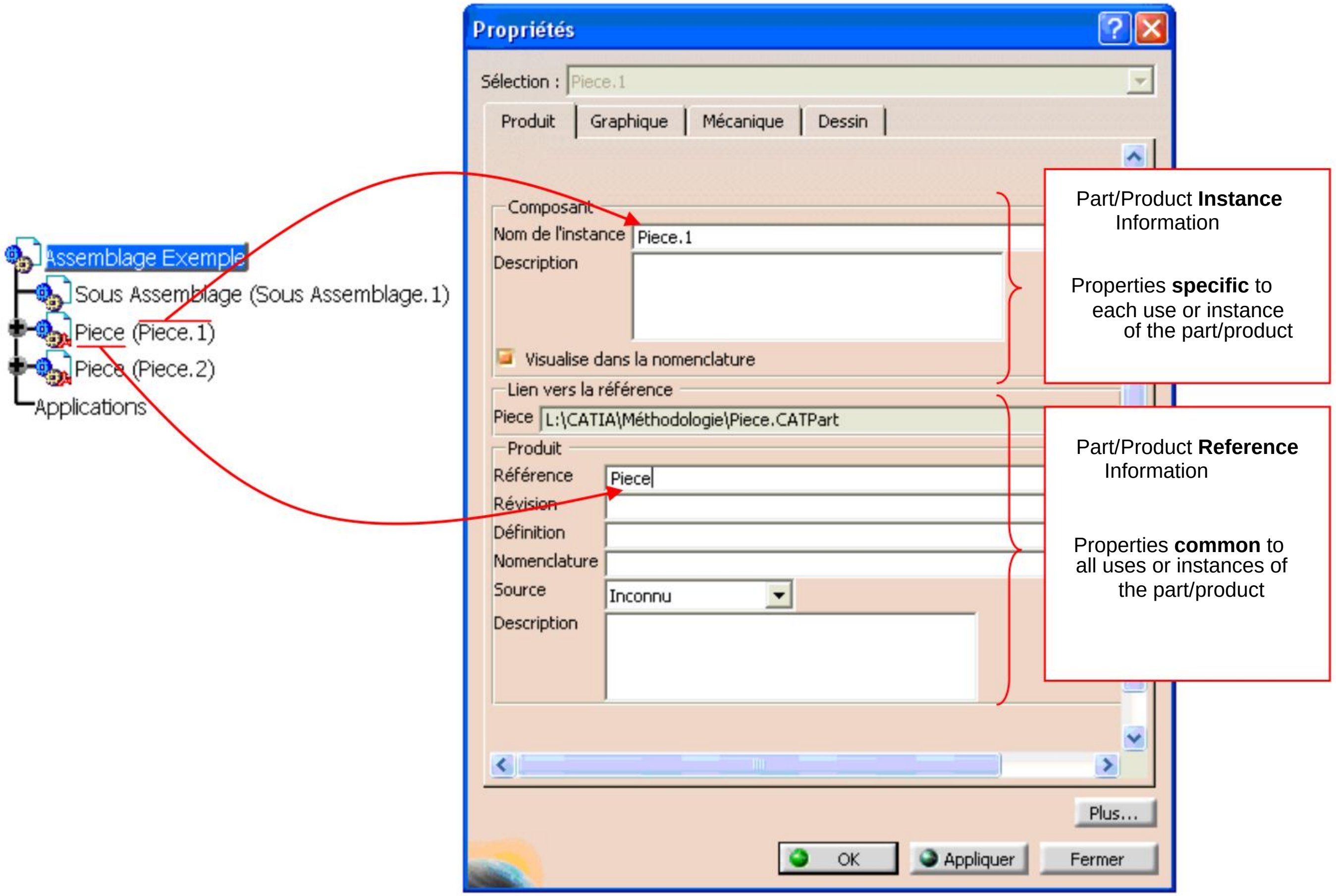
30. Transmission of files to another user

In order not to be dependent on file paths, proceed as follows:
File / send to / directory
Select the files you want to copy and the destination folder:



Note: All the documents of the CATIA session must be saved beforehand, the easiest way being to do this operation when the session is empty (without open document).

31. File name management



A part (or product) used in an assembly has three names:
- the name of the reference
- the name of the file - the
name of its instance (use)

For consistency, it is recommended that these three names be identical.
The instance name is constructed when inserting the part into the assembly from the reference name plus a point and increment. There is no automatic mechanism that changes the name of the instances when the name of the reference changes, this operation must be performed manually (in the case of the replacement of one part by another for example).

A few rules to follow: The file
name (.CATPart, .CATProduct, .CATDrawing) must be unique.
In a CATIA session, two different parts cannot have the same reference name (automatic verification).
The instance name must be unique within a single assembly level.

Each CATIA file is also identified by a unique, non-editable UID. Inter-document links are based on both document names and UIDs. Any replacement of a part by another outside of CATIA is therefore likely to fail! (cf. Example of replacing one part with another)