

Tp1

Support: mobile jaw plate of the quick-tightening vice

Objective: Draw a simple part.

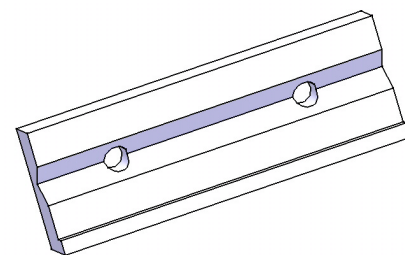
Procedure :

For any drawing, you must open the file that contains the 3D part.

- open the file **fixed jaw plate.CATpart** stored in the directory **Q:/...../layout**


- click on  and choose **drawing** or select **File->New....->Drawing** The drawing workshop is launched.

- to select **A4ISO**, check the box **Portrait** and click on **okay** The drawing sheet appears



Creation of the frame and title block

- Click on **Edit->Background layer**

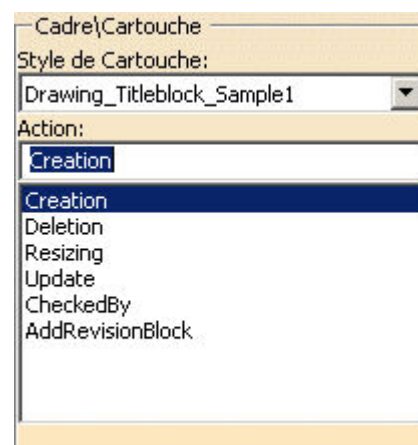
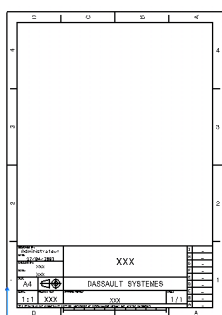
- click on  or on **Insert->Drawing->Border and Title Block** the following menu appears:

- choose the cartridge style: **Drawing_titleblock_Sample1**

- choose the action: **Creation**


- click on **okay**

title block and frame are created



Title block modification



- ZOOM on the title block

Tolérances générales ISO 2768 mK			
DESIGNED BY Votre nom	Plaque de mors fixe		
DATE 17/04/2003			
CHECKED BY XXX	Lycée Louis ARMAND		
DATE XXX			
SIZE A4			
SCALE 1:1	WEIGHT (g) XXX	DRAWING NUMBER Etau à serrage rapide	SHEET 1/1
This drawing is our property; it can't be reproduced or communicated without our written agreement.			

- complete the cartridge as on the previous model

- to modify the text zones, double-click on the text to be modified then complete the text field and click on OK to validate the modification.


Adding text

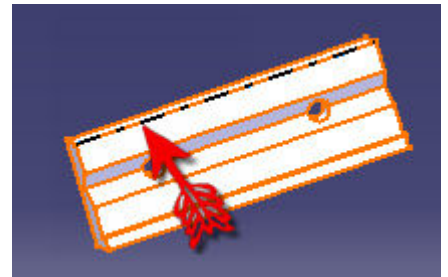
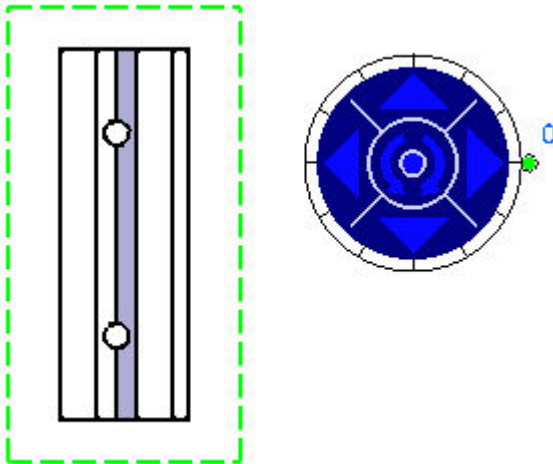
- click on  to add the text general tolerances ISO 2768 mK
- click where the text begins
- zoom **center all** by clicking on the icon 


Creation of views

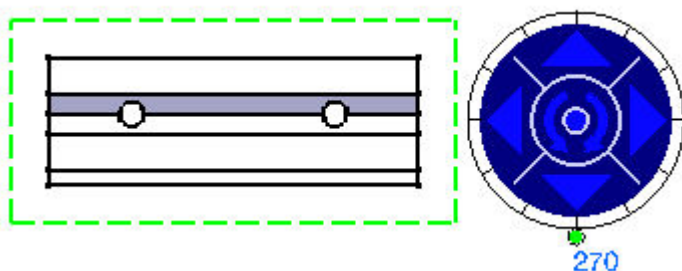
- Click on **Edit->View Layer**
- click on **Window -> Vertical Tile** to display the 3D part and the drawing sheet
- click in the drawing window to make it active


Inserting a front view

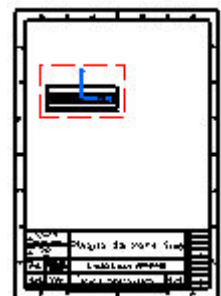
- click on  Or on **Insert->views->projections->front view** on the 3D part that is open
- select the surface marked by the arrow of the blue arrows and a green frame is displayed:



- straighten the view as follows using the arrow 



- validate by clicking on 
- place the view, as on the drawing, using the mouse (by clicking on the red frame and keeping the button pressed) .

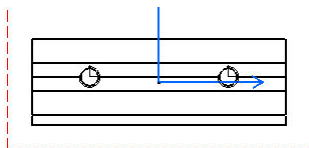


Inserting a projected view

- Click on the icon **Insert->Views** the following menu appears:

>Projections->Projection

- place the top view



Inserting a Section View

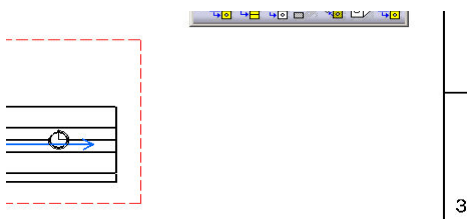
to draw the cutting plane.


- you can select points and holes.
- you can draw lines

To complete the cutting plane, double click on the last point.

- you can select a plane on the 3D part.

- zoom as in the figure



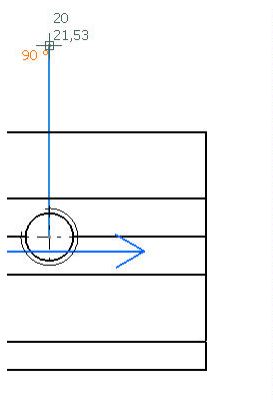
- click on  **Insert->Views->Sections->Broken Section**



- select the point corresponding to the center of the hole the symbol

- draw a vertical line

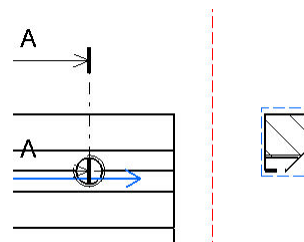
should appear next to the cursor.



- double click on the last point

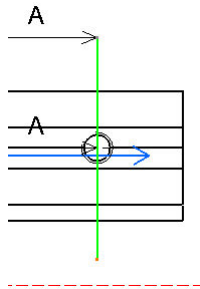
- place the view on the right

the view is incomplete



modification of the section plane

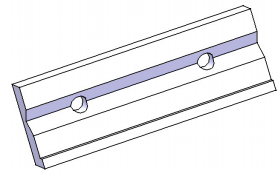
- double click on the center line of the cutting plane
- extend its end as shown in the figure



- to click  on to rebuild the section view

Inserting an isometric view

Position the 3D part as in the figure



Click in the drawing window

- Click on the icon  **Insert->views** the following menu appears:
>**projections->Isometric view**



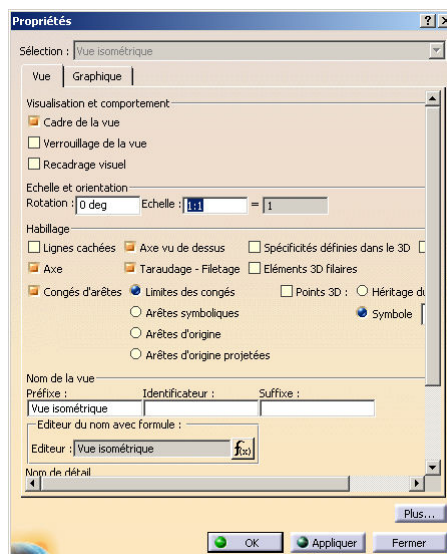
Choose  Where

- click on the 3D part
- orient and position the view using the manipulator then click on .

Dressing up views

To add the hidden edges of the top view

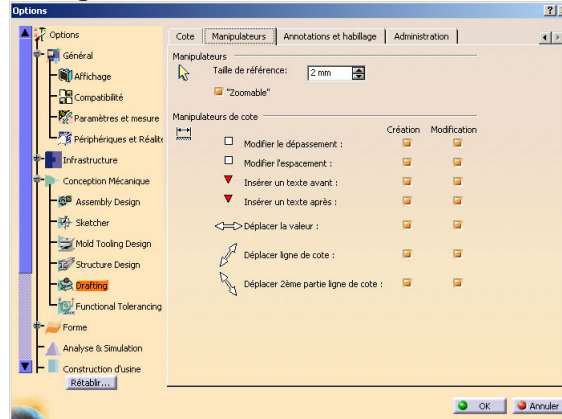
- click on the top view with the right button of the mouse the following menu appears:
- click on **property** the window is displayed
- check the box  **Lignes cachées**
- click on OK to confirm.



Part quotation :

Setting Dimension Manipulation Options

- click on **Tools->Options**
- click on **drawing** located in shaft mechanical design
- choose the tab **manipulator**
- check the options as in the following window:






- click on **ok**

DIMENSION GENERATION

Automatic generation

The dimensions will be generated from the 3D part. Only the following constraints can be generated: dimensions of distances, length, angle, radius and diameter.

Generation of dimensions in step-by-step mode.

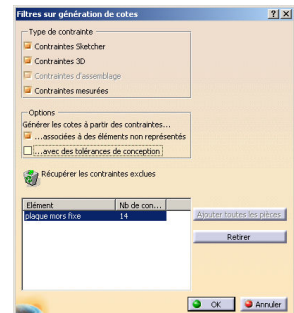
- Click on the  icon the following menu appears:  Choose  Where **Insertion-**

> **Generation->Step-by-step dimension generation** the following menu is displayed:

- check the generation options as in the menu.

- click on **ok**

the step-by-step generation dialog box is displayed:



- uncheck the pause time box

- click on the button to generate the first dimension several cases arise:

the dimension is on drawing and in the correct view (see definition drawing in file)

- click on the button to generate the next dimension

the dimension is on the drawing but in another view (see the definition drawing in the file)

- click on then on the new view.

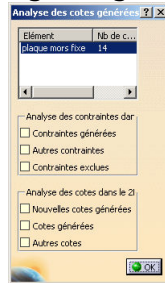
- click on the button to generate the next dimension

the dimension is not on the drawing (see the definition drawing in the file)

- click on  to delete the dimension.

- click on the button  to generate the next dimension

When all the dimensions are generated the following dialog box is displayed:

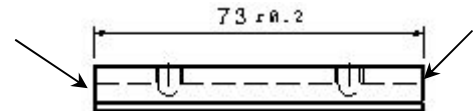


- click OK

placing dimensions manually.

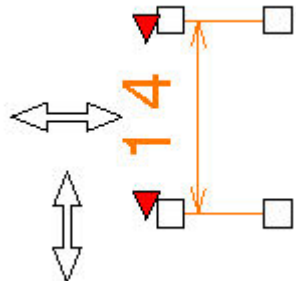
- click on 



- set the rating to 73 by clicking on the two lines marked with an arrow.




positioning of dimensions

- click on the dimension to position the following cursors are displayed



- click on  or  to move the position of the dimension to

- click on  move the position of the dimension line

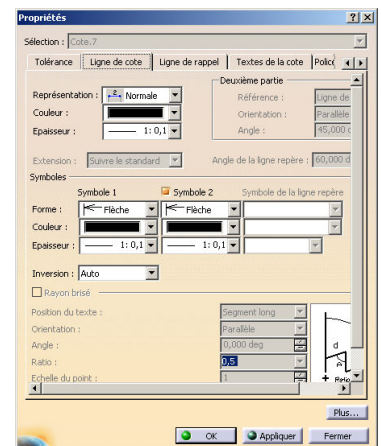
- position all the dimensions as on the definition drawing (see file).

Changing Dimension Properties

- click with the right button on the dimension of the M6 thread

- select properties and the following dialog box is displayed: This box is made up of several tabs to change the different parameters of a dimension.

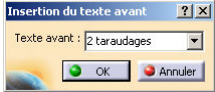
Note: you can click the apply button to view the changes.



- click on the tab 
- in the field  Choose 
- position the dimension as on the definition drawing (see file)

to add text before the dimension


- click on the front of the dimension of the M6 tapping the following dialog box is displayed



- type the text **2 threads** then click on **ok**

- do the same manipulation to add the parentheses before and after the odds of 25 and 10

to add a tolerance


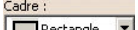

- right-click on the odds of 8
- select properties and the properties dialog box is displayed:
- click on the tab  The following dialog box appears :



- Choose **10 ± 0.2 TOL_NUM2** in the main value field


- complete the upper (0.2) and lower (-0.2) values.
- click on OK to confirm
- carry out the same manipulations for the other toleranced dimensions.

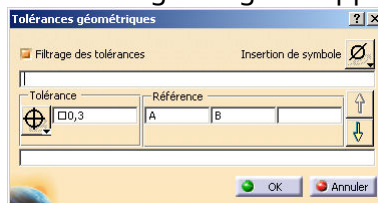
To Frame Dimensions

- right-click on the odds of 40
- select properties and the properties dialog box is displayed:
- click on the tab  the following dialog box is displayed:
- in the field  choose 
- do the same manipulations for the other framed dimensions

To Add a Geometric Tolerance

- activate the view in which you want to work (double click on the frame of the view)


- click on  Where **Insert->Dimensioning->Tolerancing->Geometric Tolerance**
- click on the dimension **2 M6. threads**, The following dialog box appears :



- enter the type of tolerance, the value then the references.
- click on OK to confirm

To add a reference

- activate the view in which you want to work (double click on the frame of the view)

- click on  Where **Insert->Dimensioning->Tolerancing->Reference**
- select the edge to which the reference surface should be attached
- place the reference as on the definition drawing (see file)

- do the same manipulations for the other reference
- modification of the attachment triangle
- click on the reference
- right click on the yellow dot
- select **Symbol shape** then

