

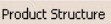
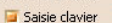
TP1

Support: Hydraulic flange

Objective: To be able to assemble a simple set

Procedure :

Setting options

- Click on **Tools->Options**
- click on **Infrastructure->Product Structure**
- click on the tab 
- tick the 

- click on Mechanical design

- check the box

- click OK


- open the assembly workshop (Assembly Design) by clicking on **Start->mechanical design-> Assembly Design**

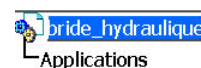
The following dialog box appears




- enter the new product reference: **hydraulic_flange**
- click on **okay** to validate

Inserting an existing component

- click on or on  **Insert->Existing Component**
- click in the construction tree on **Hydraulic_flange** the file selection dialog box appears
- choose from the directory.../CATIA/assembly/flange_tp/ the file **plinth.CATpart**.
- click on **Open**




- fixing a component

- click on the constraint or on  **Insert->Fixité**
- click on the base in the working window
- now a new constraint branch appears in the tree.

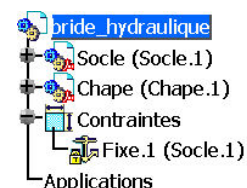




Inserting another existing component

- click on or on  **Insert->Existing Component**
- click in the construction tree on **Hydraulic_flange** the file selection dialog box appears
- choose from the directory.../CATIA/assembly/flange_tp/ the file **screeed.CATpart**.
- click on **Open**

Color change

- display the Graphic properties menu by clicking with the right button in the menu area.
- click on the screeed in the construction tree

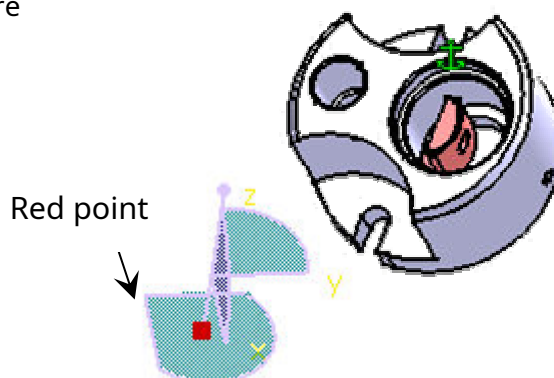
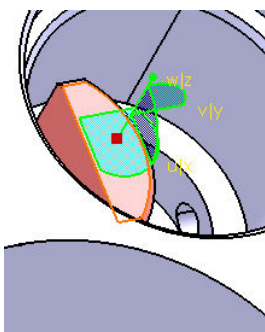


- click on the  colors area  Graphic Properties
- choose a color from the list
- the screen changes color

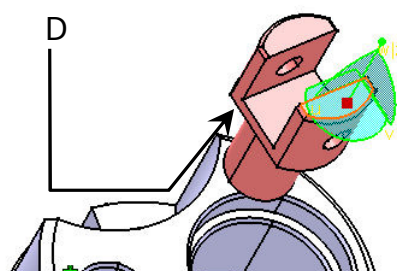
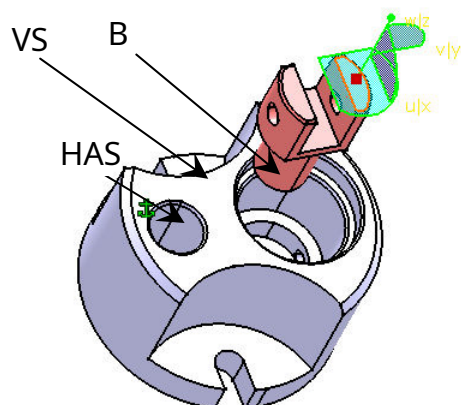
- orient your 3D image using the mouse as shown in the figure

Moving a component using the compass.

- Move the compass using the red dot on the top face screen




- select the W/Z axis and move the yoke vertically as in the left image
- perform a rotation using the arc U/X to obtain the configuration corresponding to the image on the right

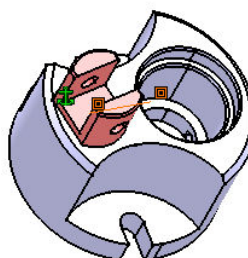


- click on **View->Reset Compass** to store the compass

Constraint creation between components

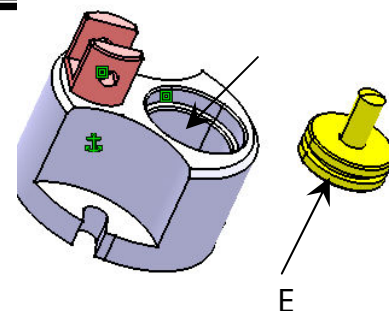
- click on 
- click on the cylinder marked A
- click on the cylinder marked B
- the two cylinders are aligned

- click on 
- click on the map marked C
- click on the map marked D
- the yoke comes into contact with the base



- insert the plunger

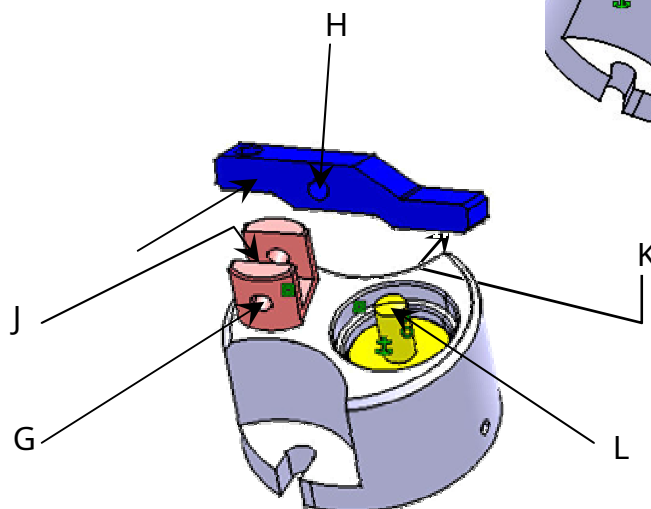
constrain the axis of the cylinder E of the piston in coincidence with the axis of the cylinder F



insert the pendulum

constrain the pendulum with:

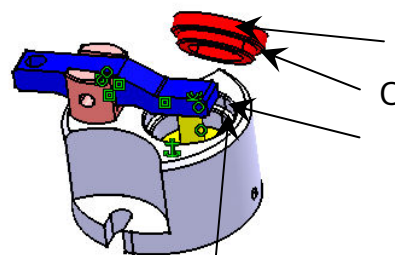
- contact of plane J and plane I
- coincidence of the G axis and the H axis
- contact of plane K and surface L



insert the lid.

Constrain the lid with:

- coincidence of the M axis and the N axis
- contact of plane O and plane P



Hide the base

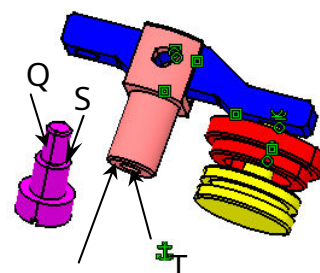
- right-click on base in the construction tree the contextual menu is displayed

- to select  Cacher/montrer

insert the pivot screw constrain

the pivot screw with:

- coincidence of the Q axis and the T axis
- contact of plane R and plane S



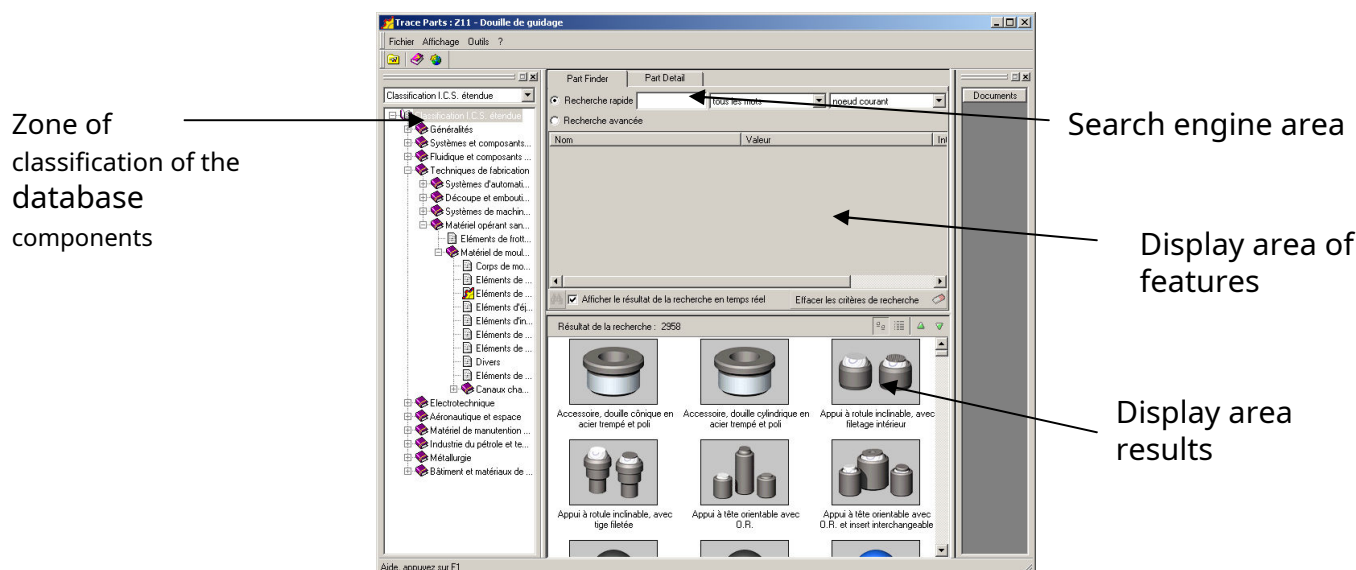
Insertion of components from a "Trace Parts" library

Open the Product Structure workbench by clicking on **Start->Infrastructure->Product Structure**

- click on 

displaying the library may take several minutes

the following window is displayed



Setting the "TraceParts" options

- click on **Tools->Configuration** The following dialog box appears
- check the box **insert the component into an alternate working directory.**
- click on **okay** to validate the configuration
- choose in the classification field: **extended ICS classification**
- check the box **Rapid search**
- type **O-ring** in the search field
- bang on **Hall**

the results display area is displayed


- click on **the O-ring (ref: 992)**

The characteristics area is displayed

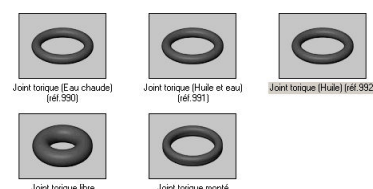
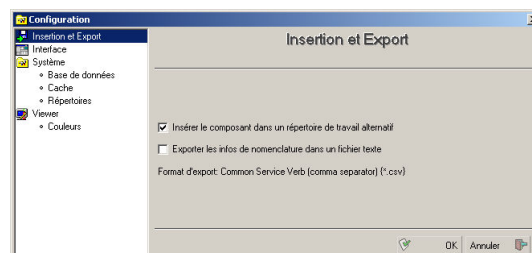
- select the line **D1: 32.5 D2: 3.6 Reference: 992-24**

- click on 

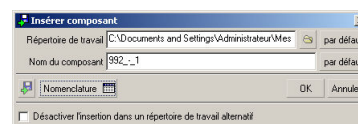
The following dialog box appears

- choose the backup directory by clicking on 
- click on **okay** to save the joint in your directory
- if a part design window opens in catia with the joint close it.

- insert the seal like the previous parts.

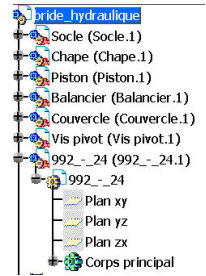
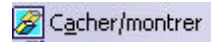


D1 (mm)	D2 (mm)	Référence
24.6	3.6	992 - 19
26.2	3.6	992 - 20
27.8	3.6	992 - 21
29.3	3.6	992 - 22
30.8	3.6	992 - 23
32.5	3.6	992 - 24
34.1	3.6	992 - 25
35.6	3.6	992 - 26
37.3	3.6	992 - 27
37.47	5.33	992 - 28
40.64	5.33	992 - 29



- make visible the ZX plane of references of the joint (it is necessary to develop the tree structure of construction)

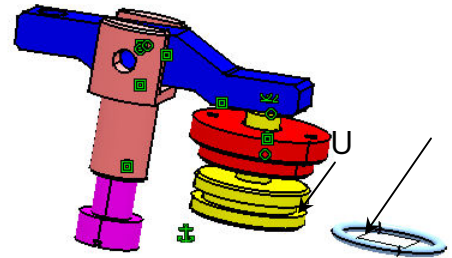
- right click on the ZX plane and choose from the context menu



constrain the joint with:

- a distance constraint set to **4.8mm** between the U plane and the ZX plane

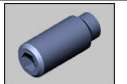
- coincidence of axis between the axis of the torus and the axis of the piston



- Insert from the "TraceParts" library an adjustment screw

- click on the tab Part Finder

Search criteria :**set screw**



Vis à métaux - Vis de réglage

- click on

- select a **M12 screw length 60**



- click on

- click on **ok** to save the screw in your directory

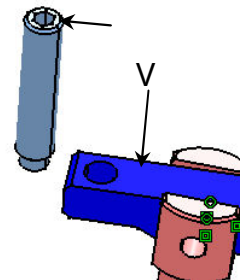
- if a part design window opens in catia with the screw close it.

- insert the screw like the previous parts

constrain the screw with:

- a distance constraint set to **20mm** between the V plane and the W plane

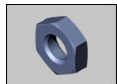
- axis coincidence between the axis of the screw and the axis of the balance hole



- Insert from the "TraceParts" library a nut

- click on the tab Part Finder

Search criteria :**low nut**



Ecrou bas hexagonal

- click on

- select a nut **M12**



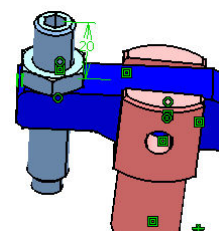
- click on

- click on **OK** to save the nut in your directory

- if a part design window opens in catia with the screw close it.


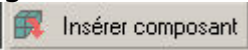
- insert the nut like the previous parts

constrain the nut as in the figure




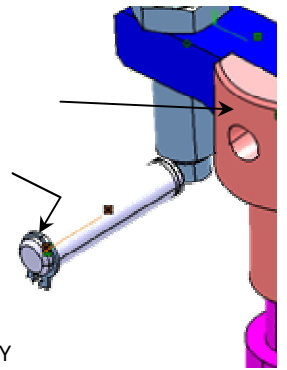
- insert the axis from your working directory
- Insert from the "TraceParts" library a stop segment
- click on the tab 
- search term: stop segment



- click on 
- select a segment for a tree of $\varnothing 8$
- click on 
- click on OK to save the segment in your directory
- if a part design window opens in catia with the segment close it.

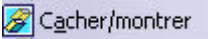
- insert the segment like the previous pieces

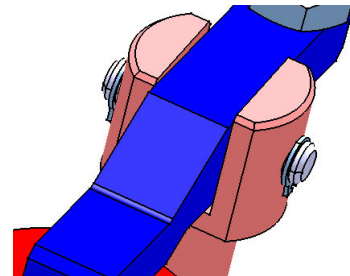
- Constrain the segment on the axis as in the figure
- constrain the axis in coincidence with the hole of the yoke
- constrain in external contact  the cylinder X and the plane of the segment Y



- insert another segment and constrain it as in the figure

Show the base

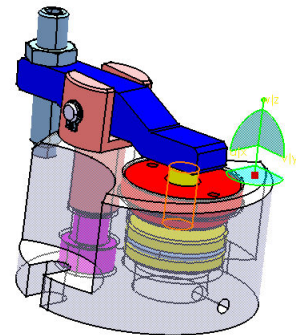
- right-click on base in the construction tree the contextual menu is displayed
- to select 



- click on the base in the construction tree
- click on the opacities area  Graphic Properties
- choose 50% from the list, the base becomes transparent

Moving Constrained Components Using the Compass

- put the compass on the base as in the figure
- select the piston in the construction tree
- move the W/Z axis while holding down the SHIFT key.



Piston, rocker arm, adjusting screw and nut move under stress

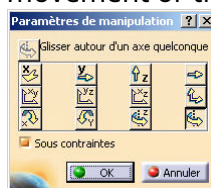
- click on to  rebuild the assembly.

Click the Manipulation icon .

The Manipulation Settings dialog box appears. You can translate or rotate components using the following options:

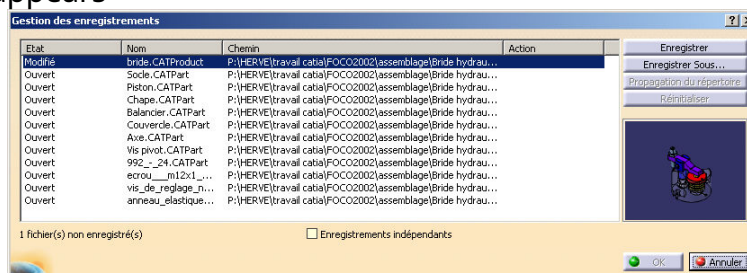
- The first and second horizontal lines are reserved for translations. You can move a component along the x, y, or z axes as well as in the xy, yz, and xz planes.
- The third line is reserved for rotations. You can rotate the component around the x, y, or z axes.

The fourth column allows you to specify the direction of your choice by selecting a geometric element. This element defines the direction of movement or the axis of rotation.



Records management

- click on **File->Records Management** The following dialog box appears



- select the file **flange.CATProduct**

- click on **Enregistrer Sous...**

- choose the backup directory and validate the proposed name

- click on **Propagation du répertoire**

- click on **ok** to validate the registration