

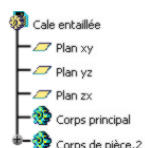
## Solid Exercise: Notched Wedge



**Purpose** :use the Boolean Intersect operation to create this part.

**Functions used** :Extrude, Intersect.


1. It is desirable to start by creating a new part body: **Insertion, Part body.**



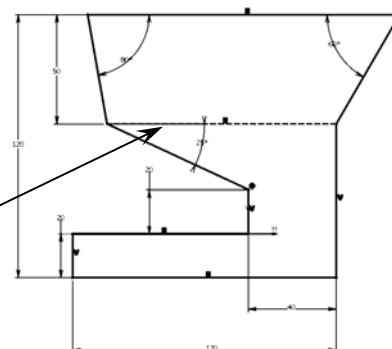
It is active (underlined).

2. In this active part body, create a sketch from

of sketch tools front view  quote in the boxes of the bar outline.

It may be interesting to draw the construction line  next.

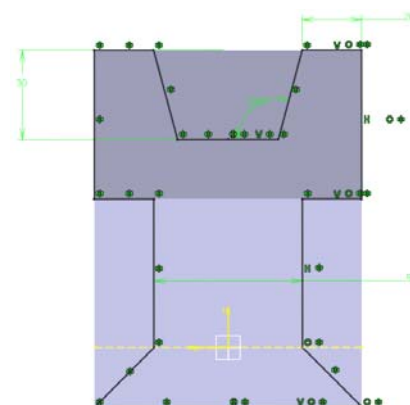
- Extrude this outline.



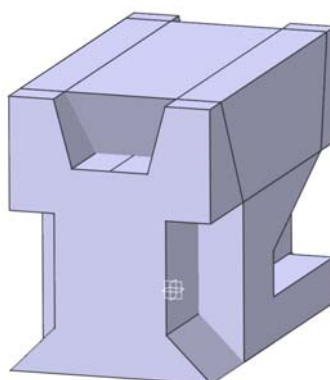
3. Insert a new part body

4. In this part body, create a sketch from the 1/2 contour of the left view,


- Place constraints to make certain entities coincide with surfaces or edges of the previous extrusion.
- Select a line then right click then on **Object...** and finally on **Automatic search** to select the 1/2 contour
- Make the symmetry with respect to the V axis
- Extrude this contour



5. We have 2 piece bodies and we will take the part common to both.



- Show menu **Boolean operations** (right-click on an icon then check Boolean operations)

- Click on the Intersection icon , expand the black triangle if the wrong icon appears.

- Select the 2 part bodies.

The piece is over.

