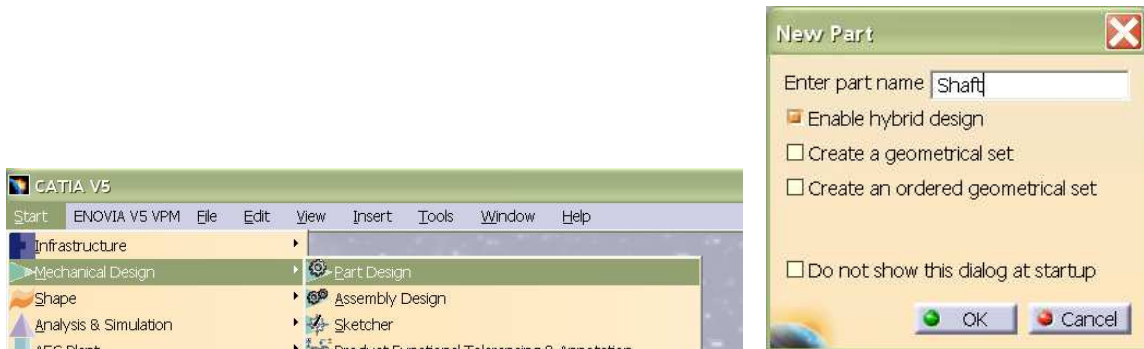
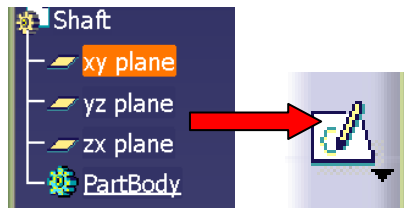


CREATING SHAFTS: Axis-rotational objects.

1. Create a new Part Design session and write “Shaft” as the name of the part.



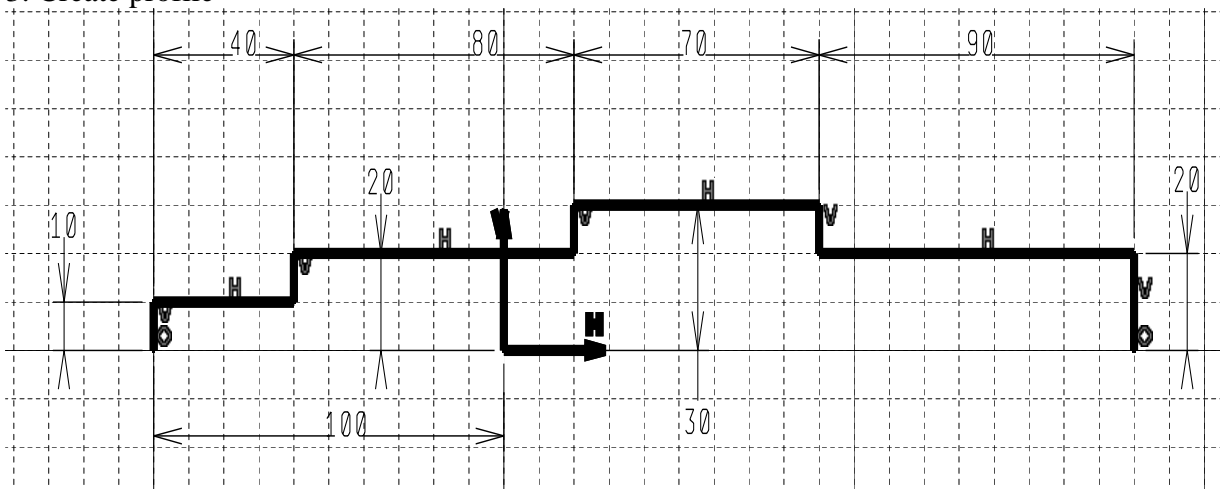
2. Select xz plane and open Sketch:



Check and activate *Snap to point* option !



3. Create profile



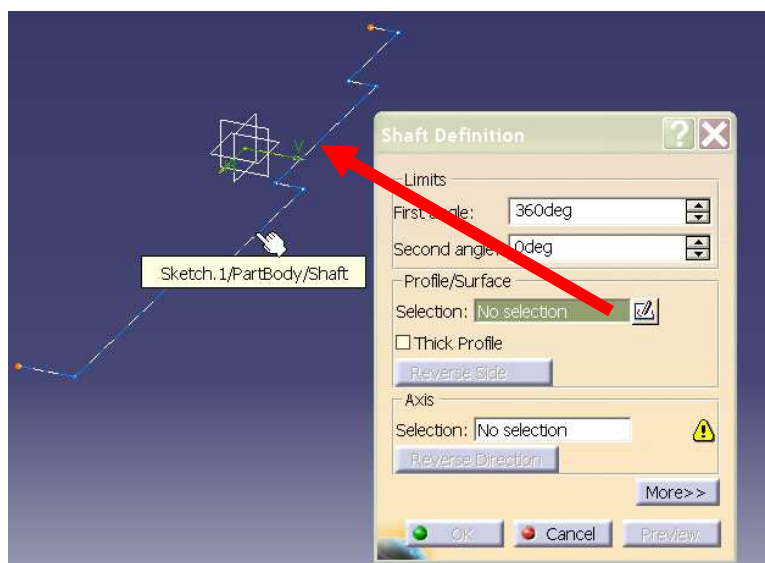
then create axis using which is horizontal and it is lying on H vector



Exit sketcher workbench



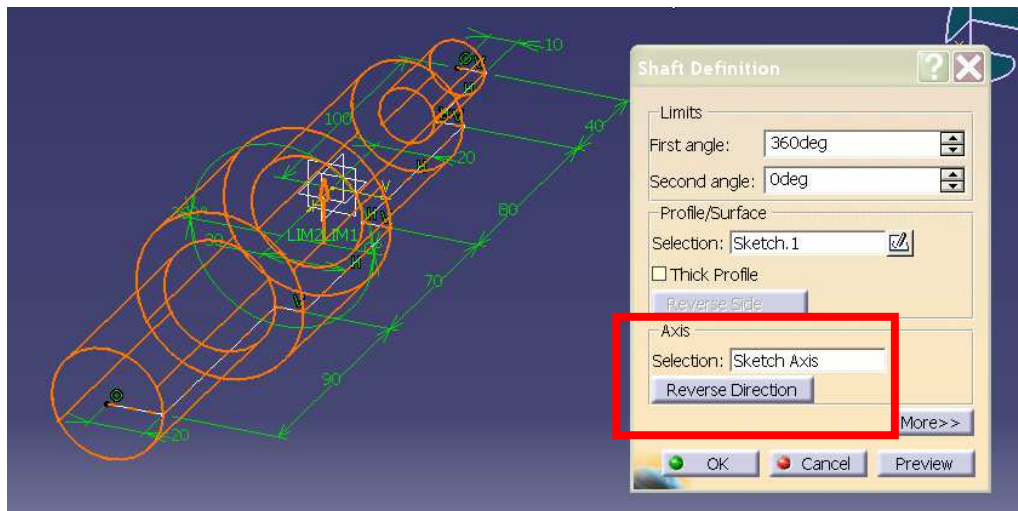
4. Create the shaft using **Shaft** option from **Sketch Based Features** toolbar



Select *Profile/Surface Selection* field and choose previously created sketch.

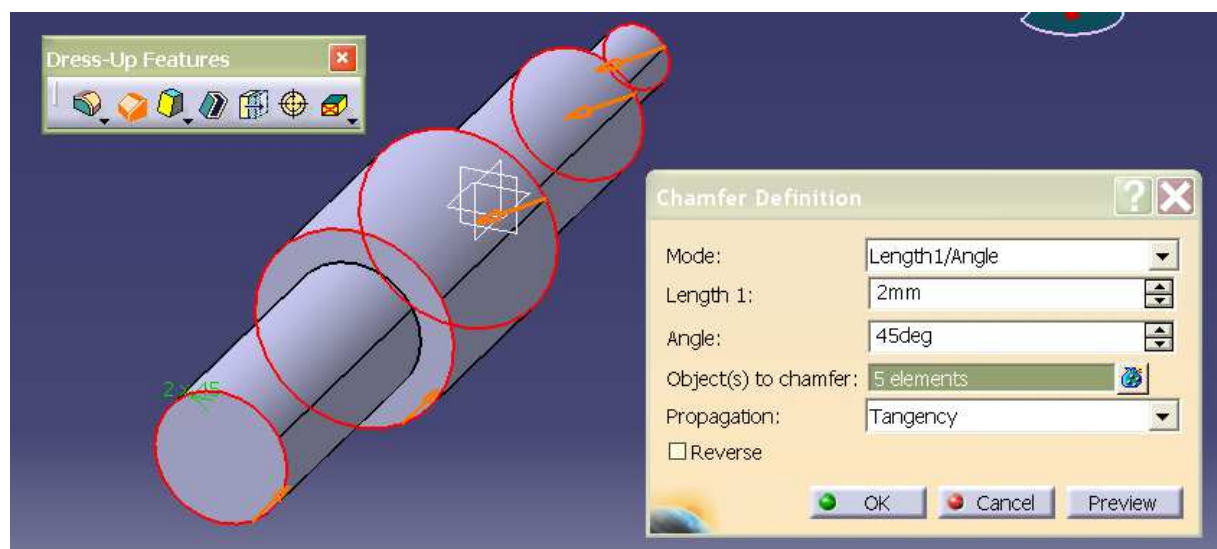
The shaft will be created, and the *Sketch axis* is automatically selected as the axis of the rotation for selected profile.

NOTE: in this case profile can be opened ! Axis is taken into account as the line, which is closing the profile.

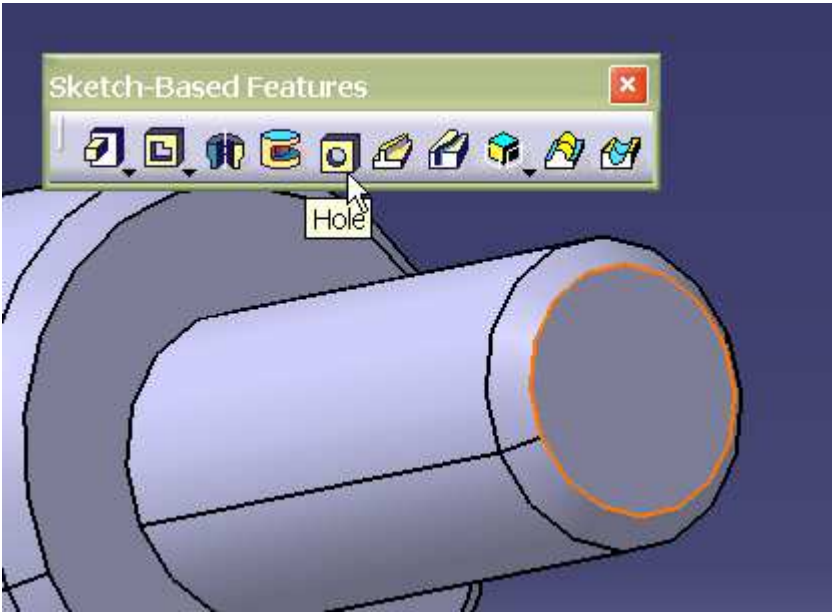


Press OK button to confirm operation.

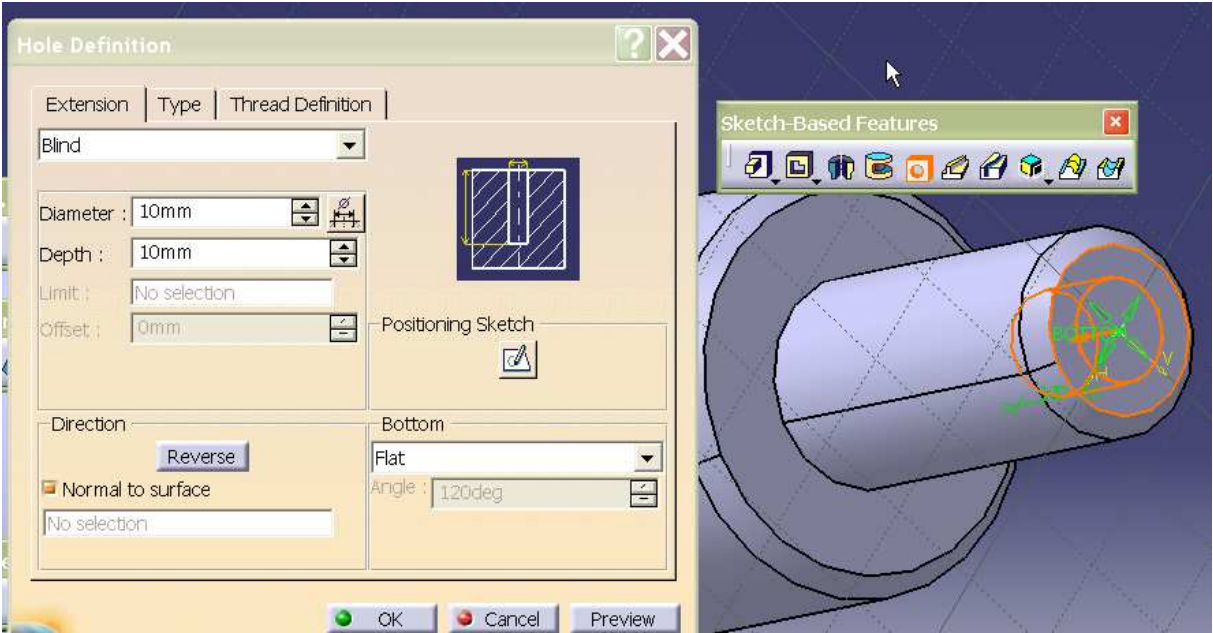
Remove sharp edges using **Chamfer** option from **Dress-Up Features**. For *Length1/Angle* mode set *Length1=2mm* and *Angle=45deg*. Then select *Object to chamfer* field and choose edges as on the figure below. Use OK button to confirm.



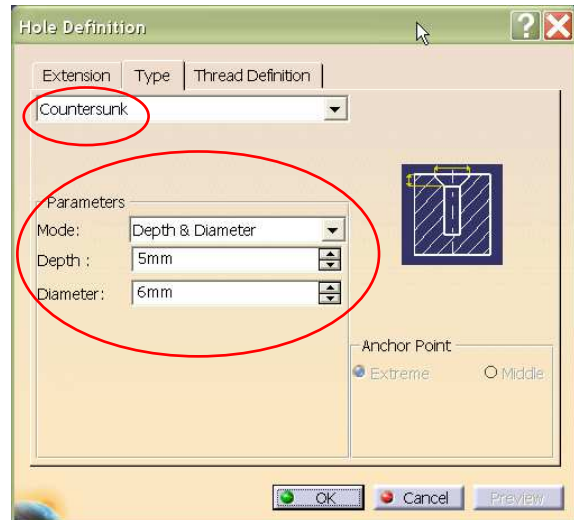
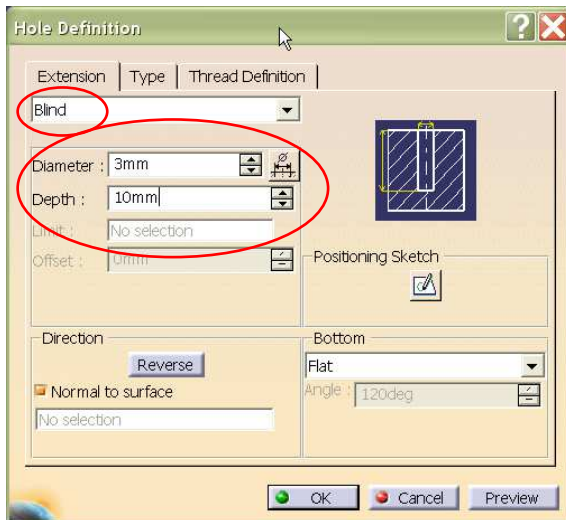
Select **Hole** option and choose plane in front of the shaft.



Using this option it is possible to create predefined holes with special shapes including threads.

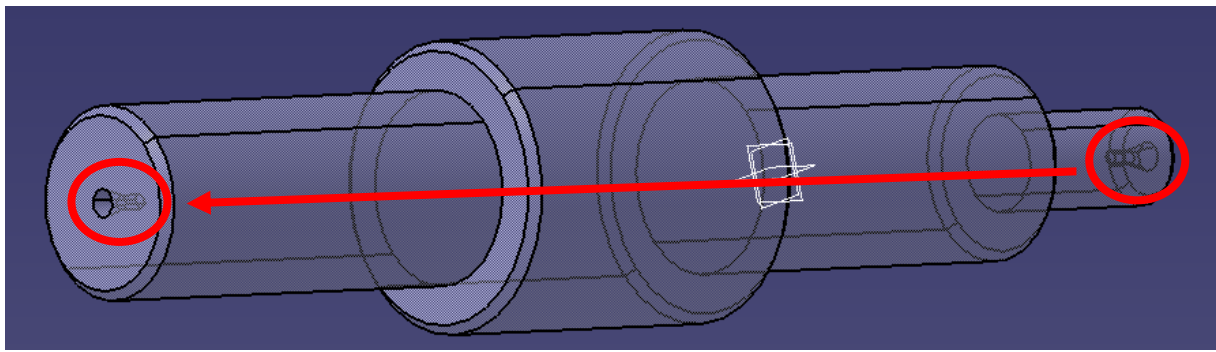


For *Hole definition* window fill the tabs **Extension** and **Type** as follow:

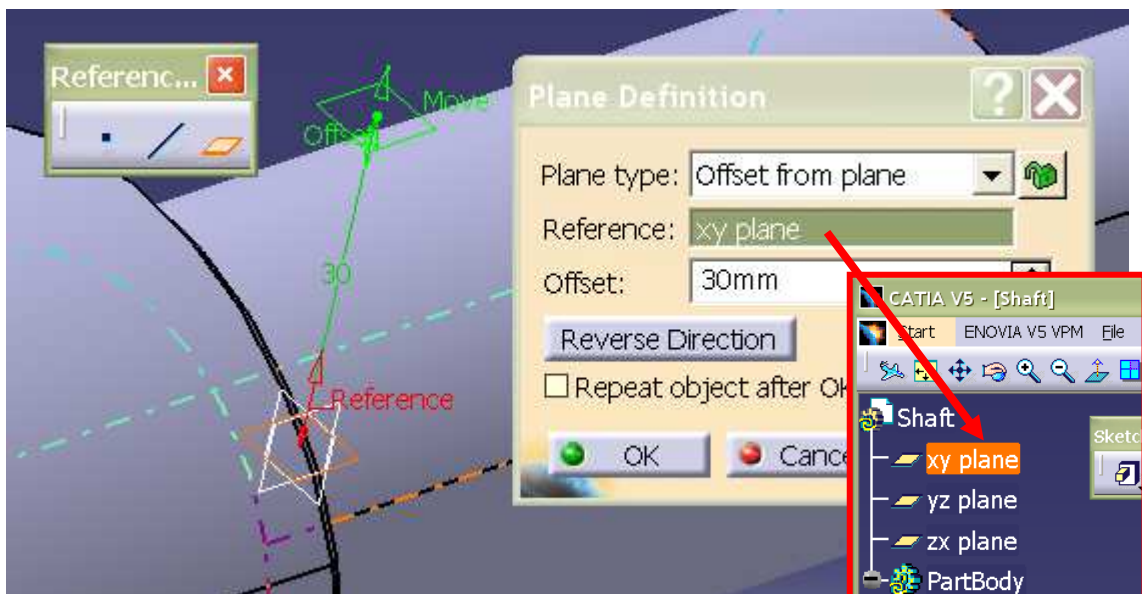


Use OK button to confirm operation.

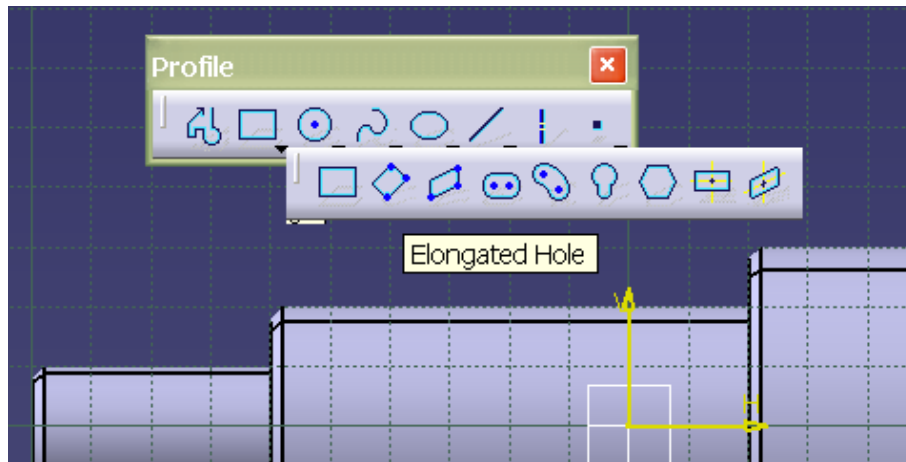
NOTE: create the same hole on front-plane at opposite part of the shaft.



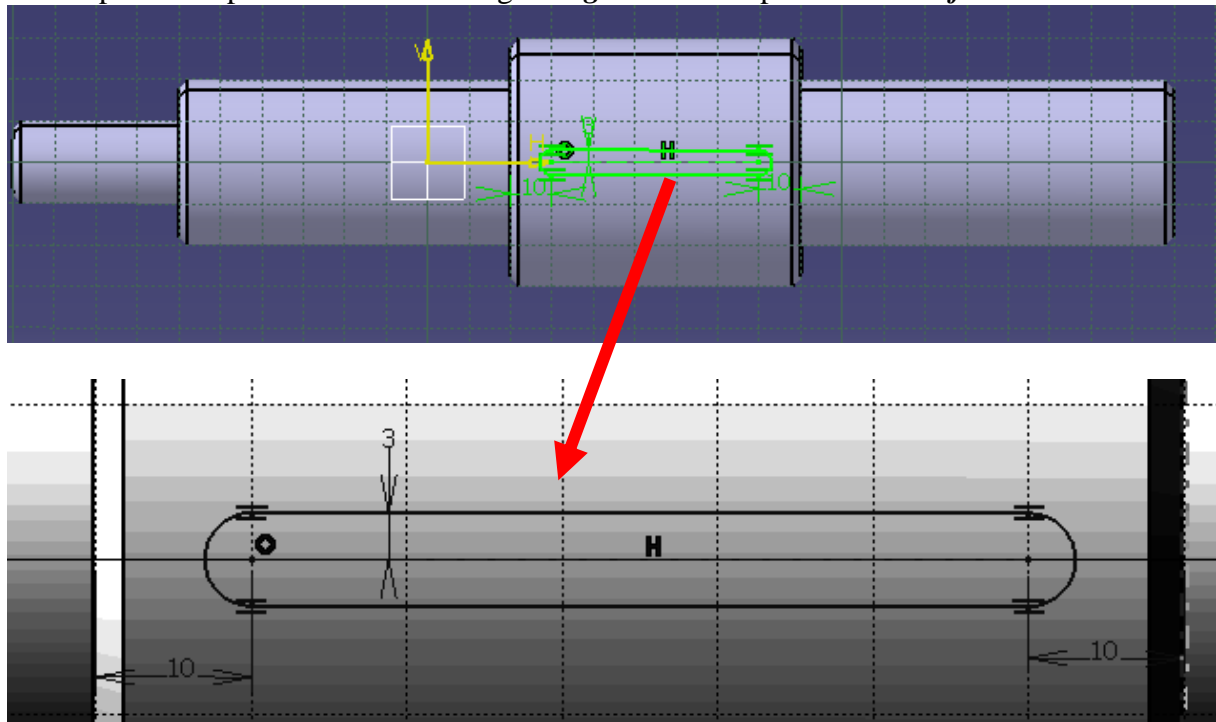
Create additional **Reference Element** as plane parallel to the XY plane with *offset* 30mm. Select **Plane** option from **Reference Element** toolbar. Set parameters in Plane Definition Window as on picture below. Choose **Reference Element** field and select xy plane on tree (left upper corner of the screen).




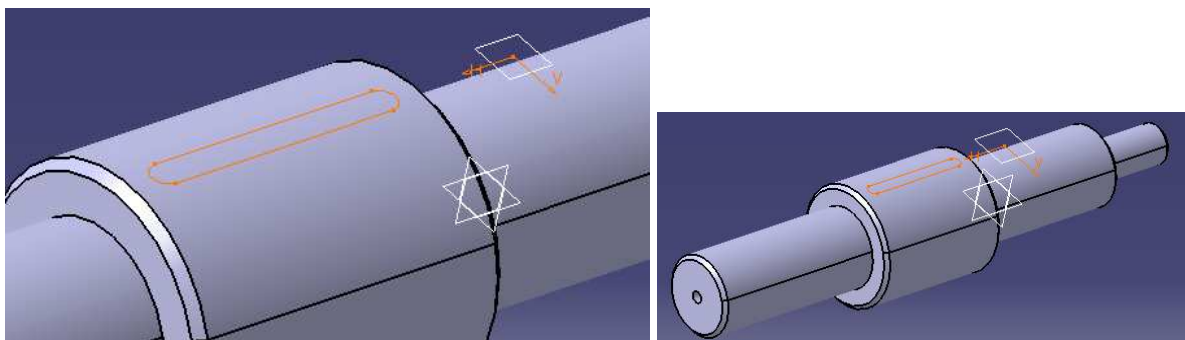
Select new created plane and open *Sketcher*. Place cursor on small, black triangle near the *Rectangle* option to see additional options:



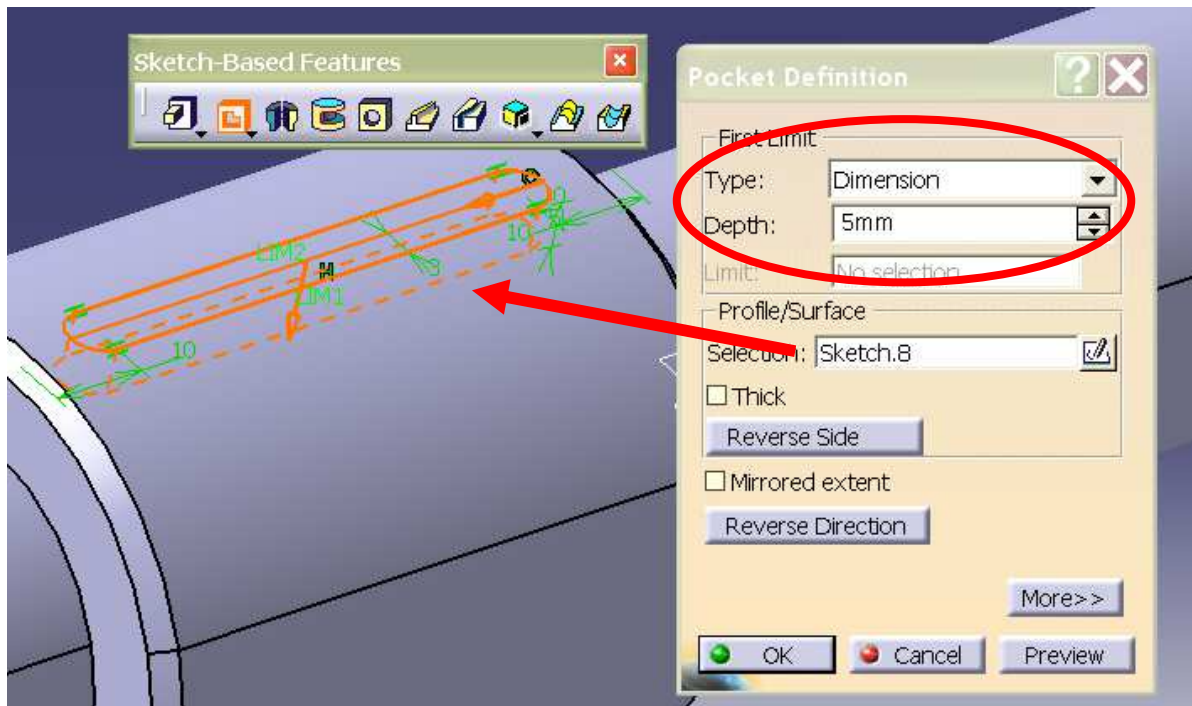
Create profile as presented below using *Elongated Hole* option from *Profile* toolbar.



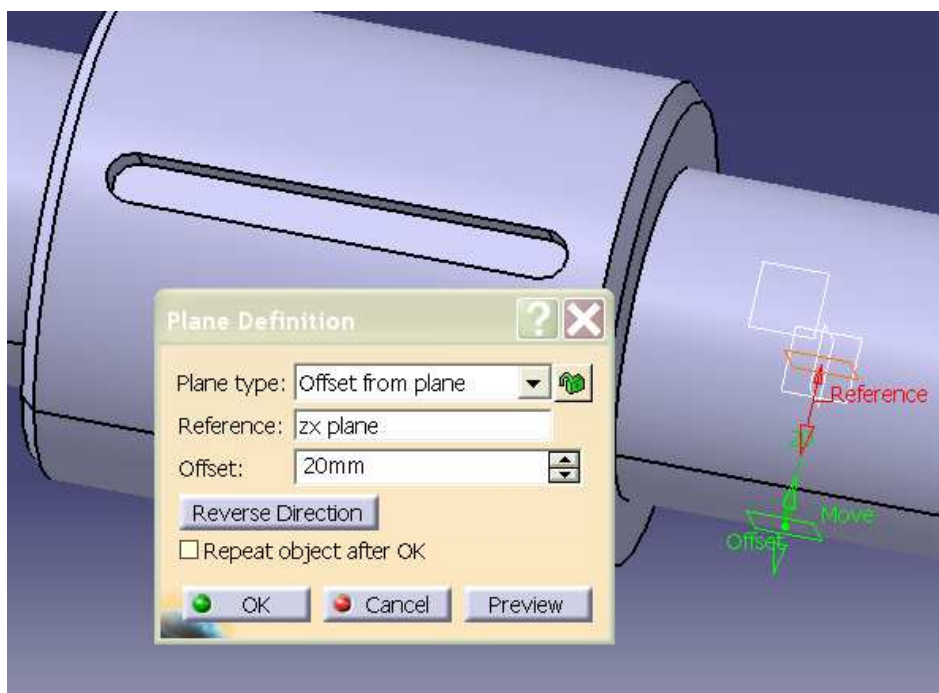
Exit *Sketcher* , it should be visible a sketch on new plane:



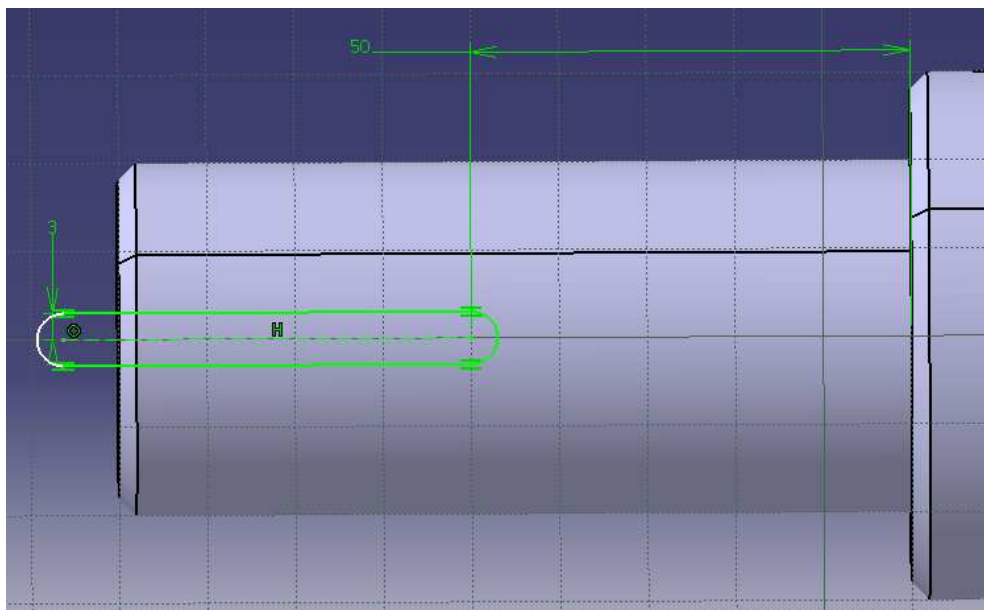
Select **Pocket** option from **Sketch Based Features** toolbar, choose previously created profile (sketch) to fill **Profile/Surface Selection** field. The depth of the inlet is **Length: 5mm**.
NOTE: Use Reverse Direction button if necessary



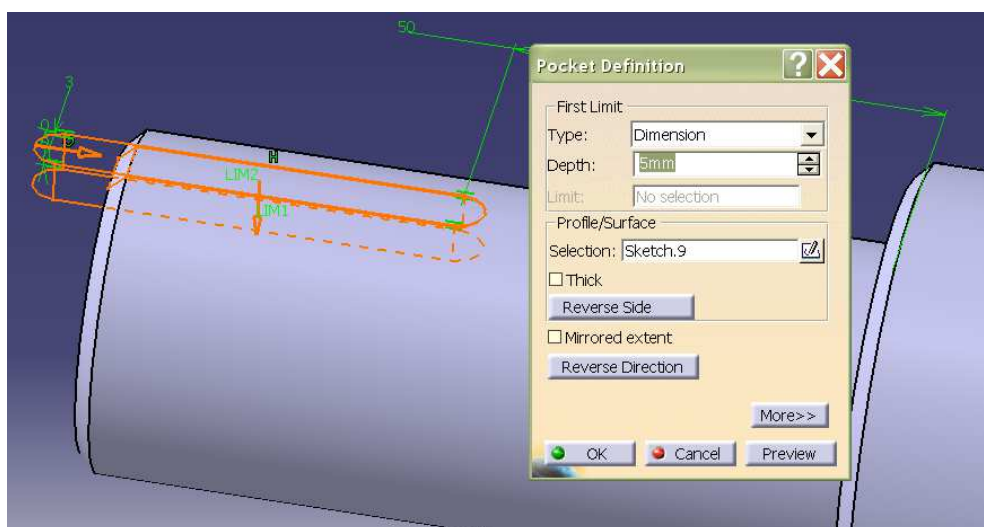
Create additional **Reference Element** as plane parallel to the ZX plane with **offset 20mm**.
Select **Plane** option from **Reference Element** toolbar. Set parameters in Plane Definition Window as on picture below. Choose **Reference Element** field and select zx plane on tree. Press OK button to confirm.



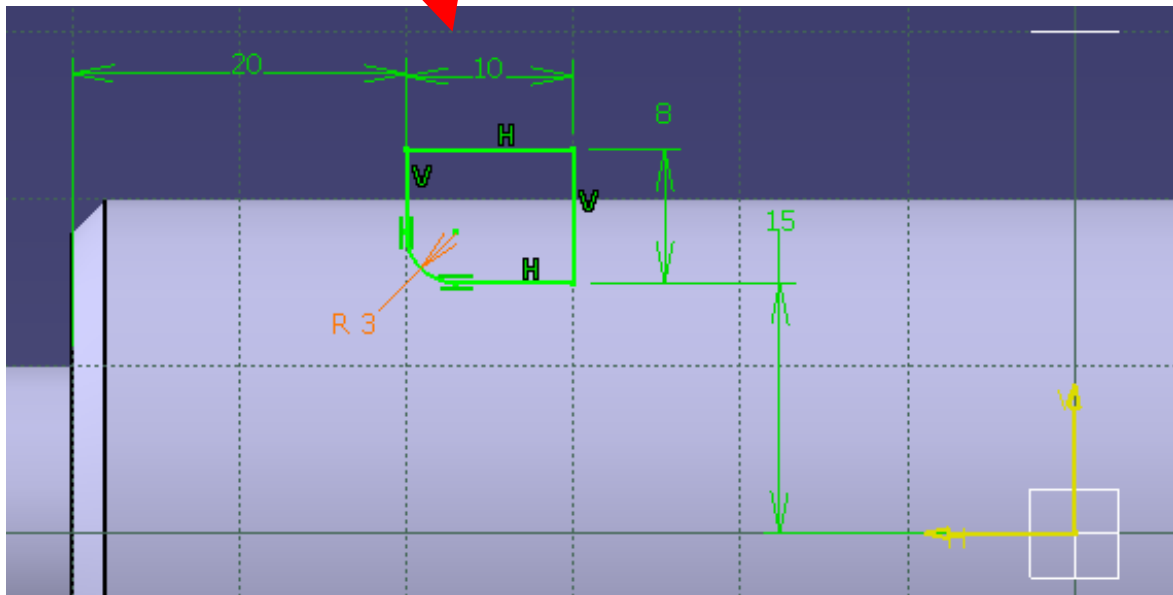
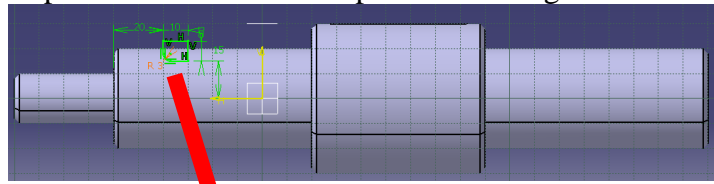
Select new created plane and open *Sketcher*. Create profile as presented below using *Elongated Hole* option from *Profile* toolbar.



Select *Pocket* option from *Sketch Based Features* toolbar, choose previously created profile (sketch) to fill *Profile/Surface Selection* field. The depth of the inlet is *Length: 5mm*.
NOTE: Use Reverse Direction button if necessary

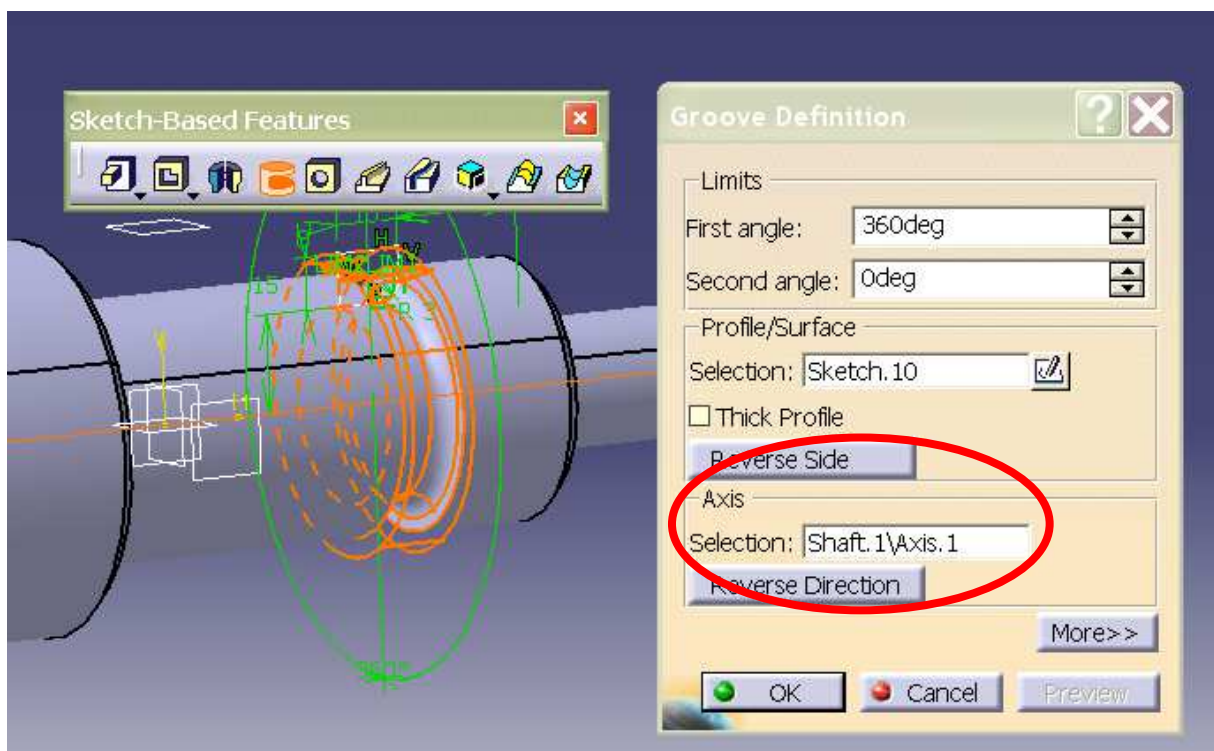


Select plane ZX and open *Sketcher*. Draw the profile as on figure below:

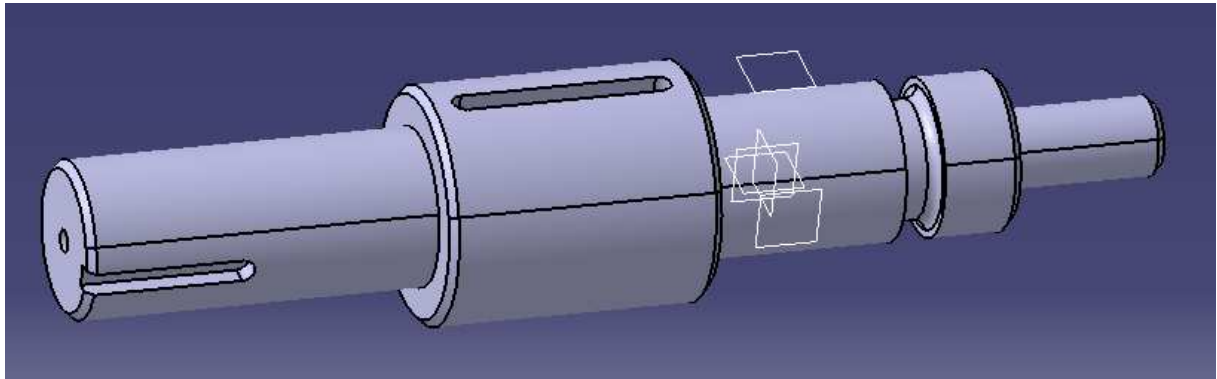


Select *Groove* option from *Sketch Based Features* toolbar, choose previously created profile (sketch) to fill *Profile/Surface Selection* field.

To set the axis of rotation (*Axis Selection* field) select the field and place cursor just on the shaft. Konfirm operation using OK button



Final part should look like on picture below.



End of the task.