# CATIA – Sketcher: Correlated dimensions (exercise)

This simple exercise shows how to create structures with correlated dimensions, e.g. width of the rectangle should be two times greater than his height.

Note: *Italic description means options in main menu of the CATIA user interface.*

1. **Create new part design document using *Start/Mechanical Design/Part Design* option.**

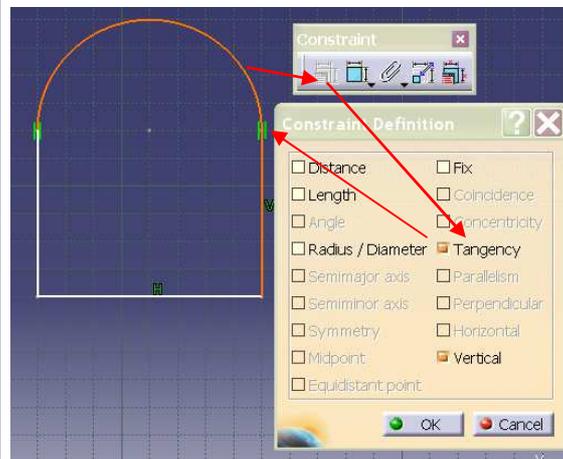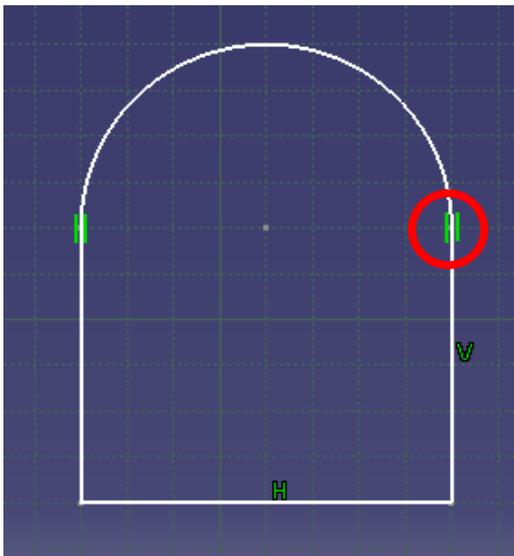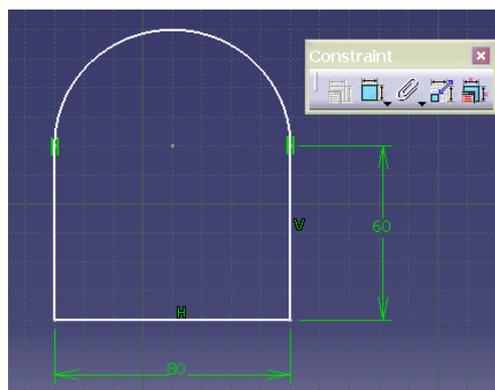2. **Select XY plane and open sketcher:**
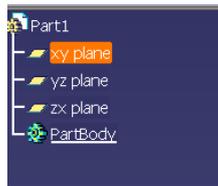
Insert/Sketcher/Sketch

3. **Create profile similar to presented below:**

Be sure for tangency between arc and vertical lines. If they do not exist select arc and line (hold CTRL key to select two objects), then use **Constraints Defined in Dialog Box** to create <u>Tangency</u>. Repeat operation for second line.
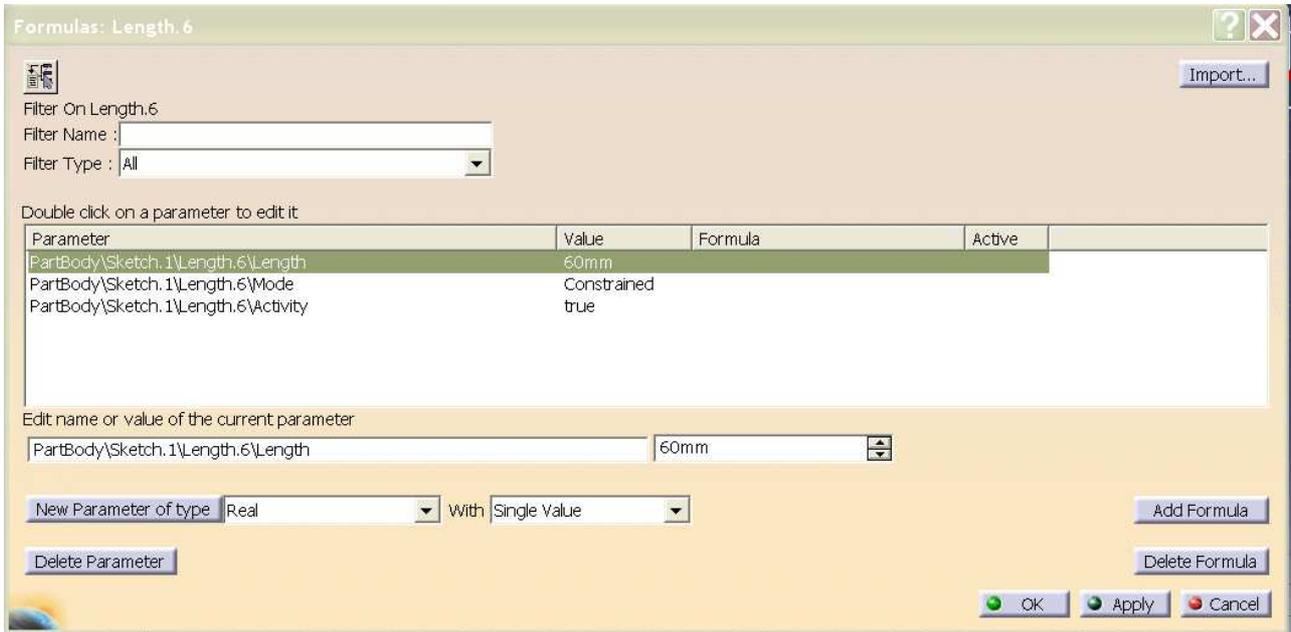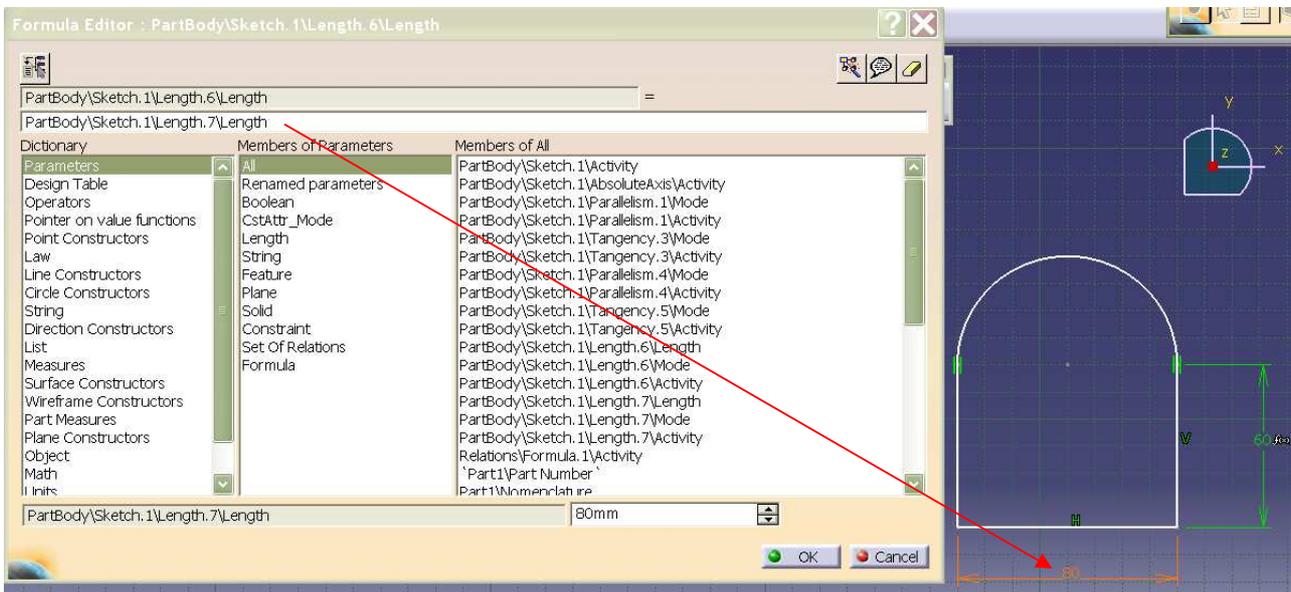

4. **Create dimensions:**

5. Create relation between dimensions:

Let assume, that height of the element should be equal 60% of its width. To create such correlation it is necessary a formula for selected dimensions. Select dimension 60 and start *Tools/Formula* option.

Press **Add Formula** button and select dimension 80 (Length.7 in this case). Then complete formula by multiplying selected dimension by 0.6.
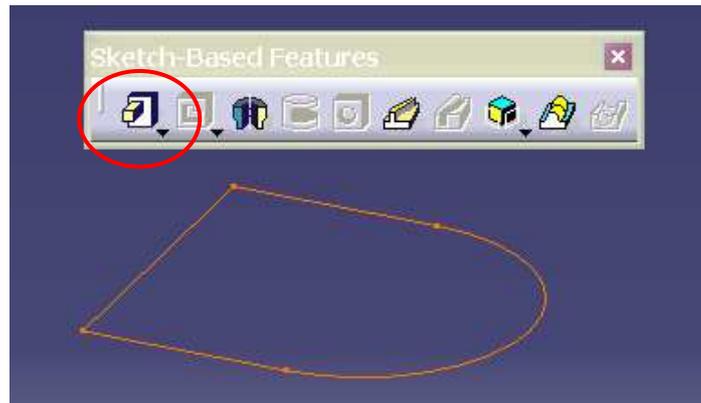

Confirm operation. Dimension 60 should be recalculated to 48. Since now every changes of the dimension 80 will involve changes of the correlated dimension.
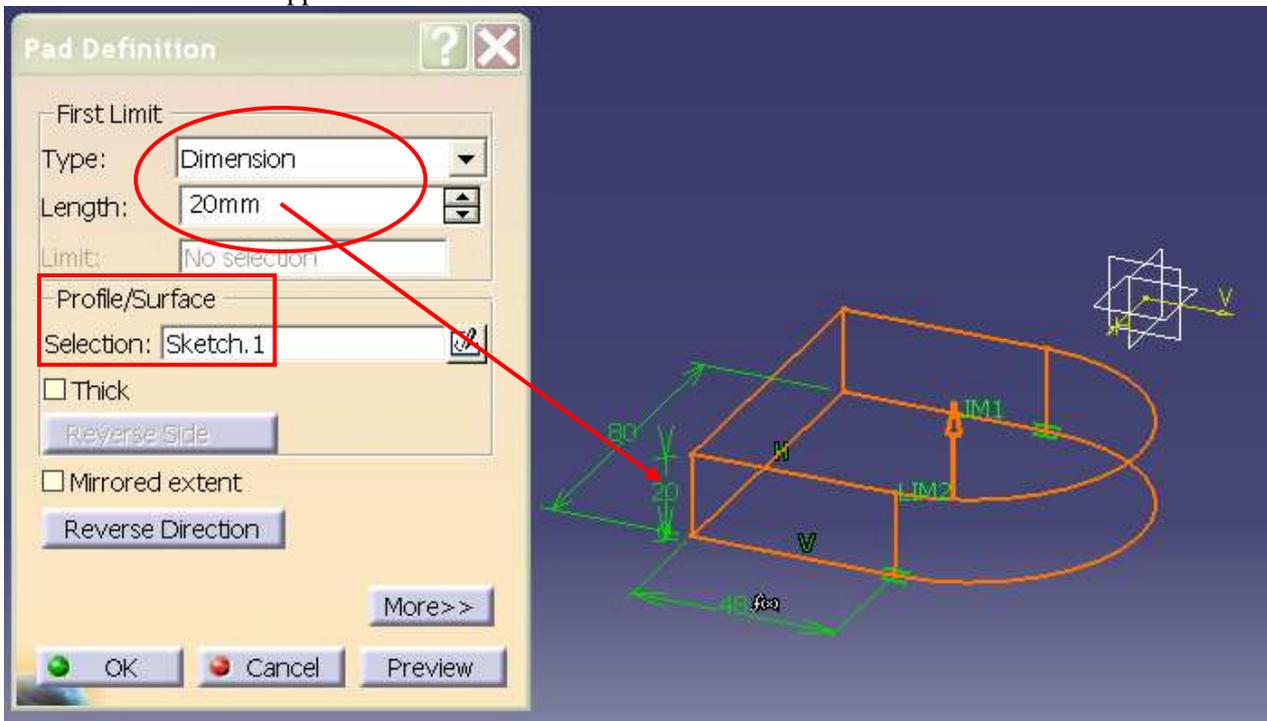
Exit sketcher and save the project.

6. Create 3D model using PAD:

Select previously created profile and press **Pad** icon (*Insert/Sketch-Based Features/Pad…*).

Pad Definition window appears.

Using **First Limit Type** option as <u>Dimension</u> user can define the height of the created 3D model. Use **Preview** button to see the result. **OK** button allow to accept results.

Note: It is important, that the sketch used for 3D creation (here *Sketch.1*) must be displayed in **Profile/Surface Selection** field. User can select Pad icon without sketch selection, but in this case it is necessary to set the sketcher in this field by selection from exists sketches.
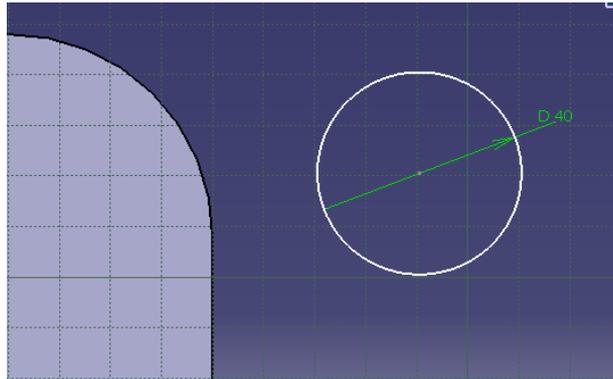
7. Create new sketch on plane, which is a part of previously created 3D model:

Note: Each plane (flat surface) from any of 3D model can be used as a plane for sketching.

Select the top plane of the previously created 3D model and open the sketcher.
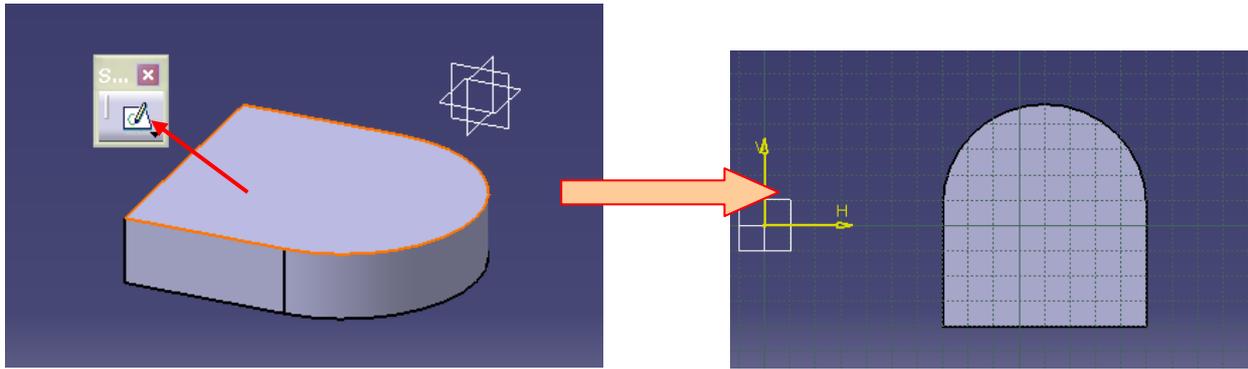
Draw circle on sketch and use **Constraints** to define its *Diameter=40mm*.


Next, locate the center of the circle in the same point as the center of the arc. The *Concentricity Constraint* must be created.

It is recommended to do not use the edge of the 3D model as arc selection. Much better solution is to use cylindrical surface, which appears after Pad definition. Rotate the object and select circle and surface as presented below:
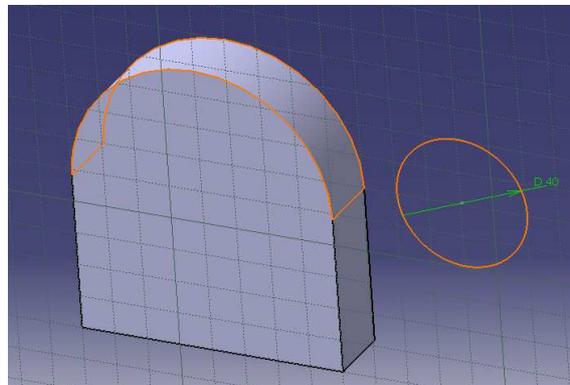

Use **Constraints Defined in Dialog Box** to create *Concentricity* constraint. The circle should be moved to the center of the arc automatically.
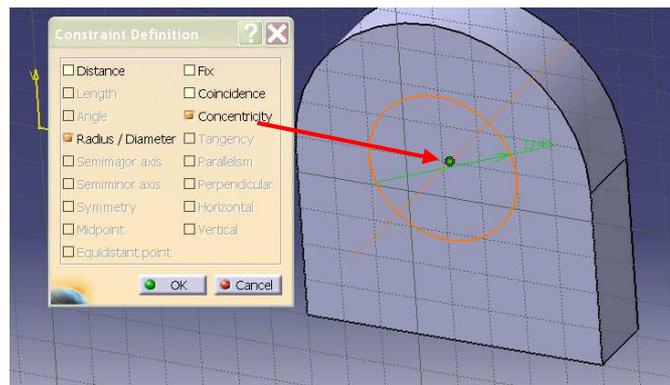
8. Define relation for dimension between different sketches:

It is possible to create some relations between dimensions defining in different sketches. Let assume, that difference between radius of the arc and radius of the circle should be equal to 10. Of course radius of the arc is equal to half of the length=80. To create this relation it is necessary to define a formula.
Select diameter of the circle and start new formula: *Tools/Formula* option (sketcher should be steel active !).
Note, that <u>Diameter=40</u> is represented as radius 20mm.
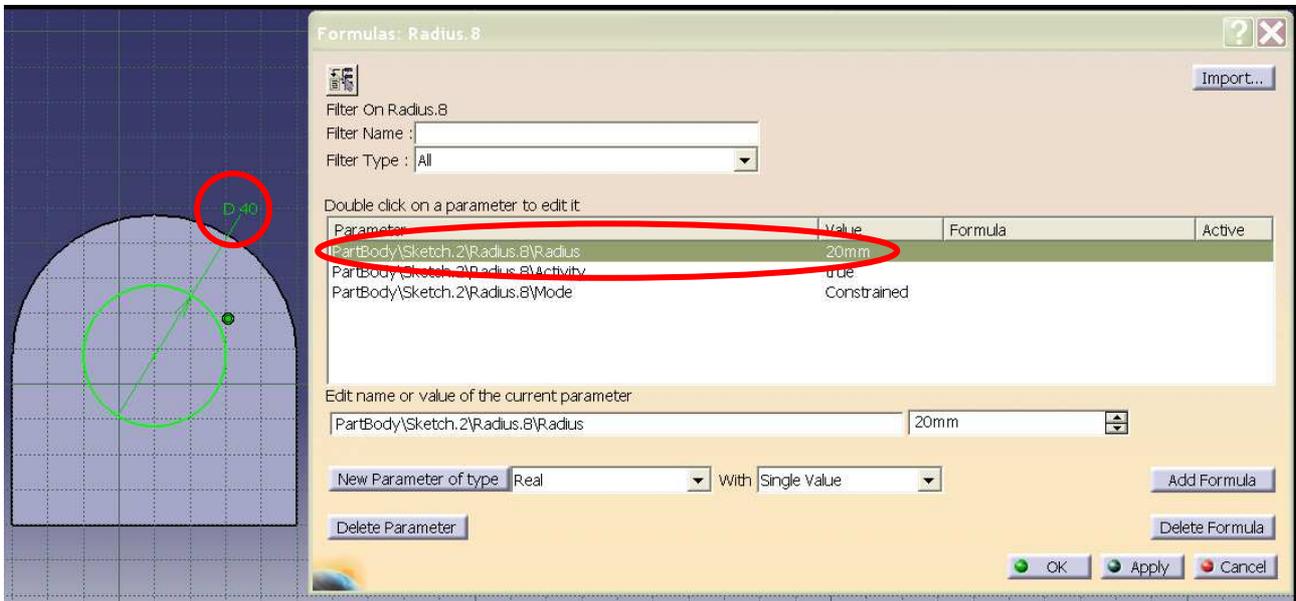
Press **Add Formula** button. To select length=80 from Sketcher.1 just find and activate **Sketcher.1** on the **History Tree**. Then select dimension 80.
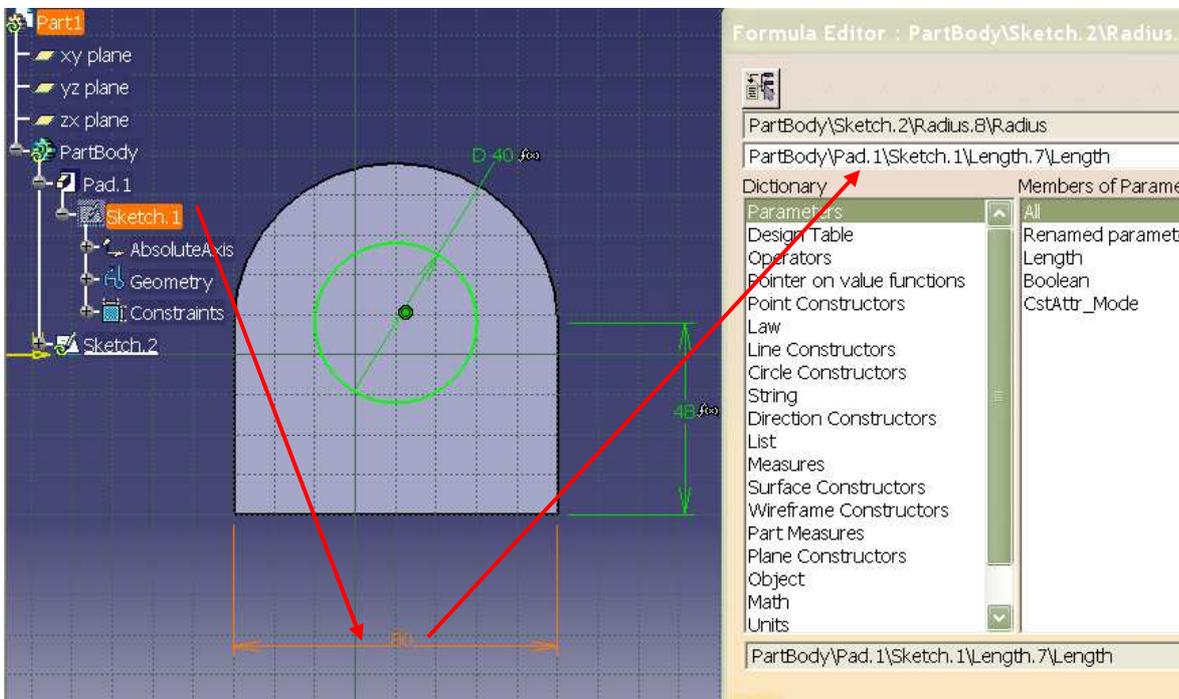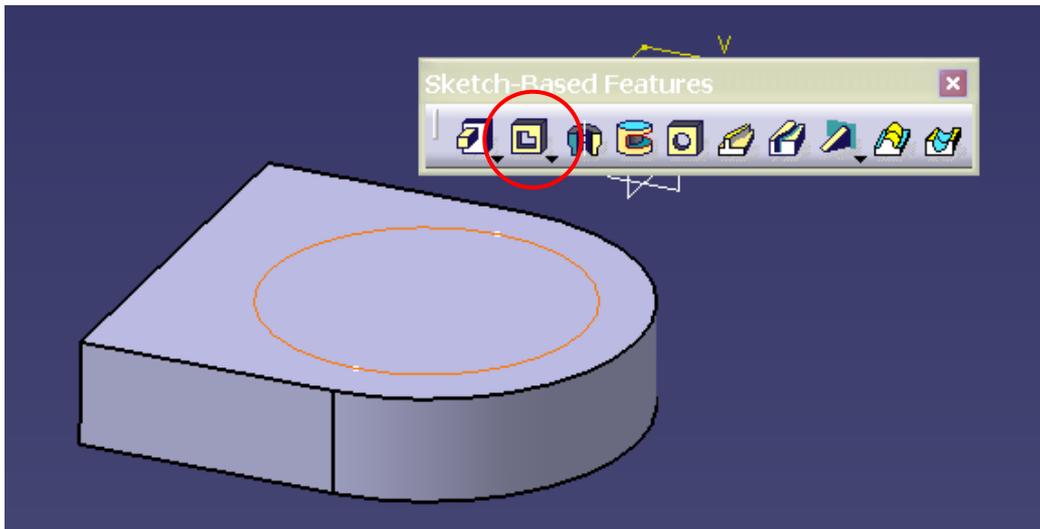

Insert the following formula (take care of the bracket !!!), **also unit after 10 (here mm) is very important**:
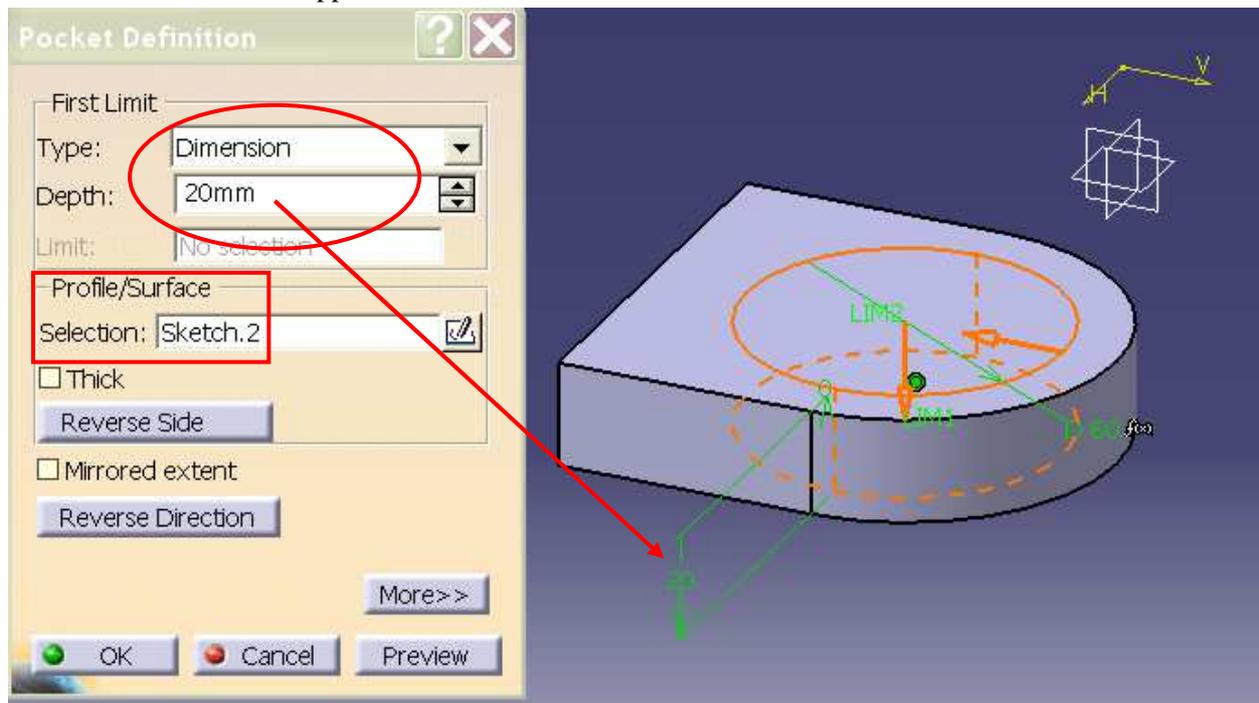

Close sketcher and save your project.

9. Create hole in 3D model:

Select previously created circle and press **Pocket** icon (*Insert/Sketch-Based Features/Pocket…*).

Pocket Definition window appears.

Using **First Limit Type** option as <u>Dimension</u> user can define the height of the created 3D model. Use **Preview** button to see the result. **OK** button allow to accept results.

Note: It is important, that the sketch used for 3D creation (here *Sketch.2*) must be displayed in **Profile/Surface Selection** field. User can select Pad icon without sketch selection, but in this case it is necessary to set the sketcher in this field by selection from exists sketches.
This option has the same structure as Pad definition – however a hole is created.