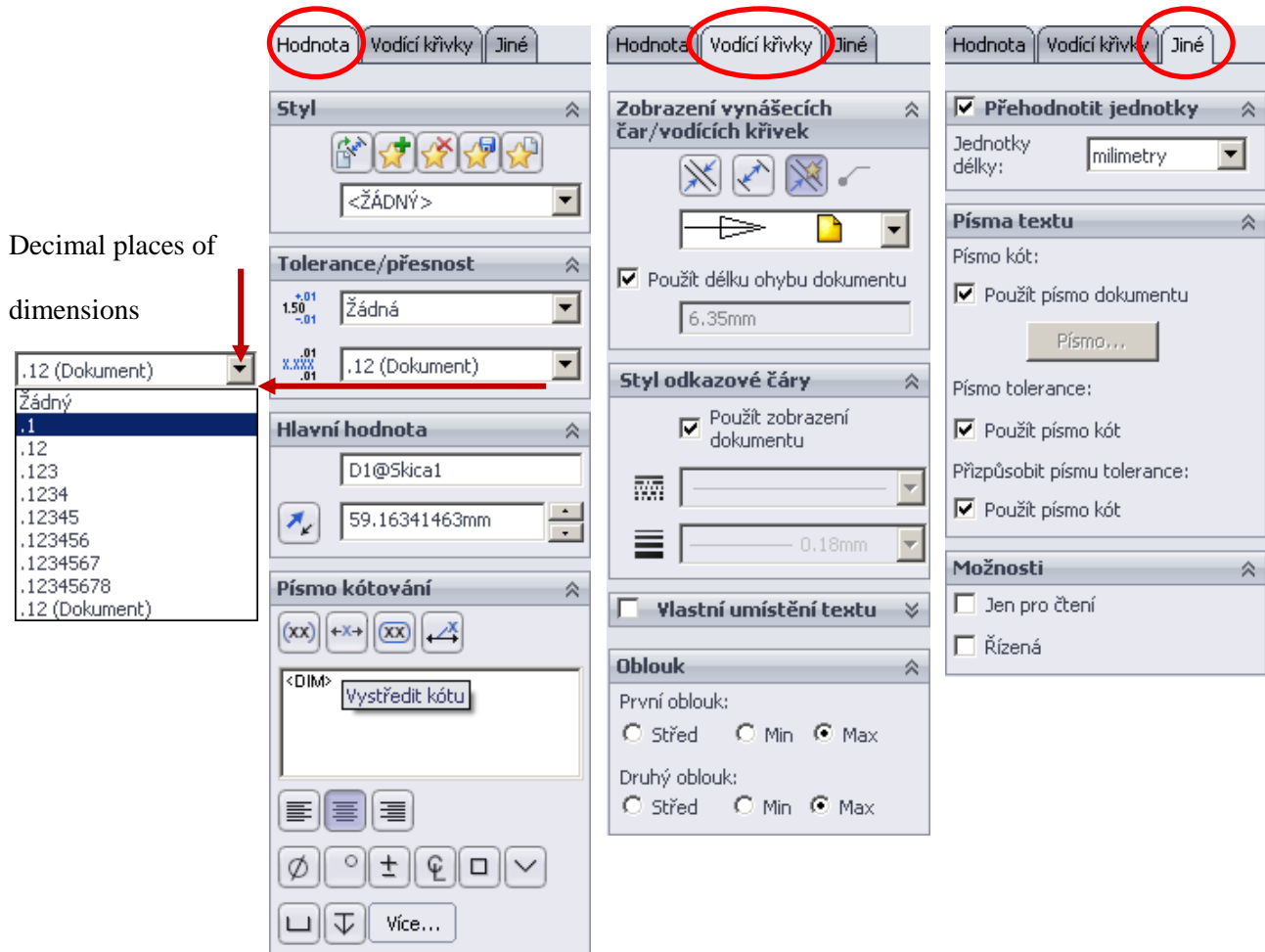


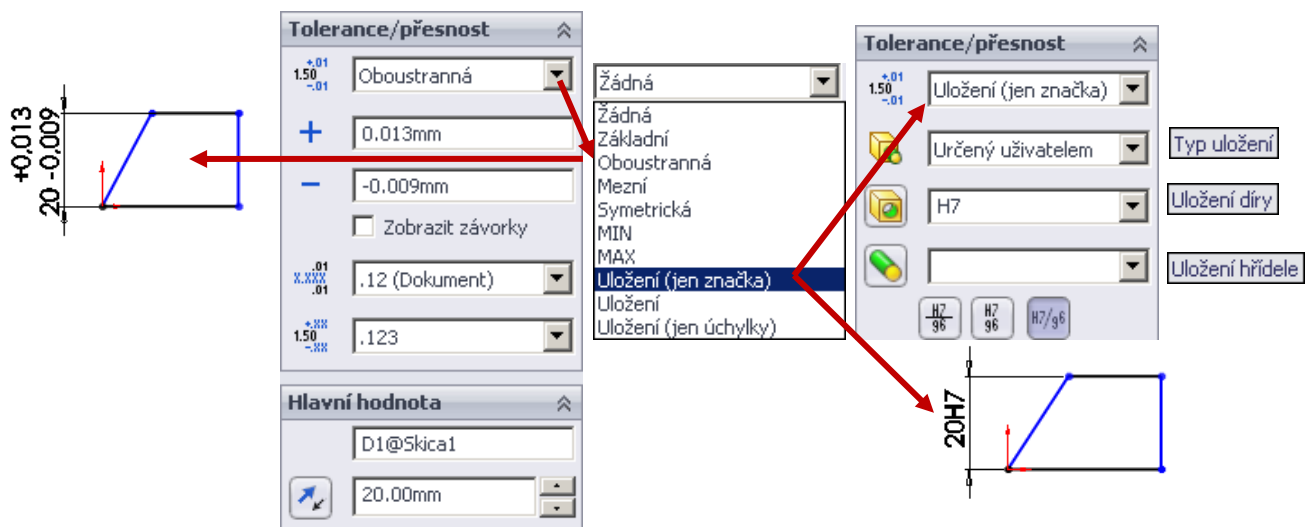
PARAMETRICAL DIMENSIONING

Remember, entered dimension are reflected into the model and the drawing. They are parametrical – if the size of entity in model is changed, it will be changed in drawing too.

The way of dimensioning and the style if dimension you can enter by these three bookmarks:



Define tolerance in **Tolerance** bookmark

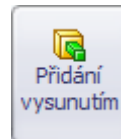


If you define tolerance or dimension saving by this way, they are parametrical. Systems for machining (CAM) are able to work with them. .



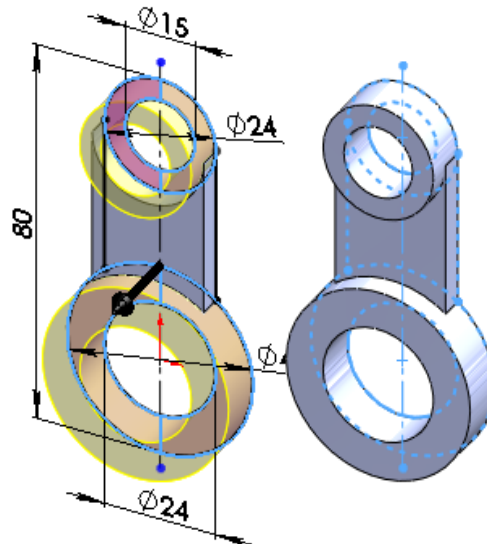
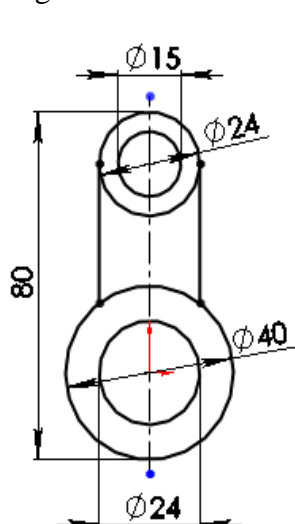
Draw the sketch in the defined planes. Create solid figures by command **Adding by putting out with chosen outlines**.

Vybrané obrysy




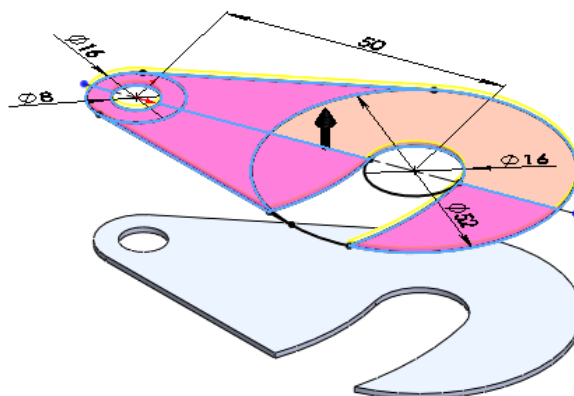
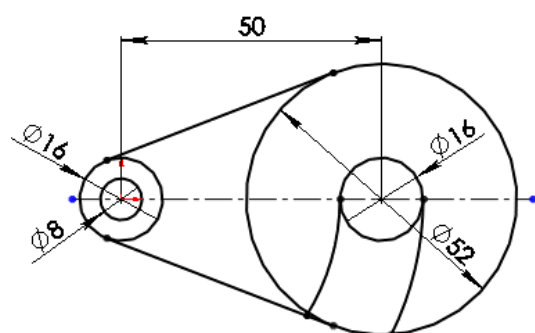
FRONT SKTECH PLANE

Define the first sliding 5 mm from sketch plane, the second 10 mm.



UPPER PLANE

Use  **Středový oblouk** and indent it. Slide 2 mm from the sketch plane.



RIGHT PLANE

Draw 1/8 of sketch. Copy it by reflection. Slide the sketch 8 mm from the sketch plane.

