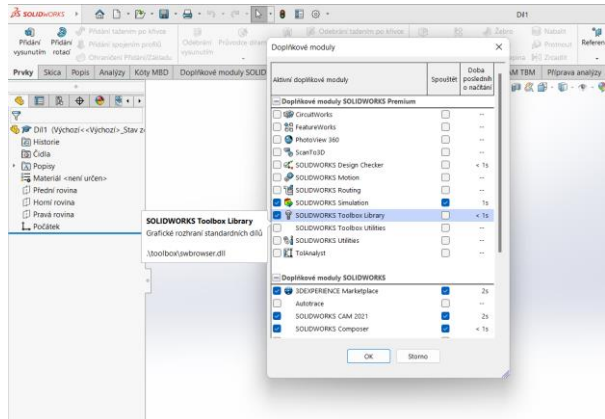


## Gear from the SW library

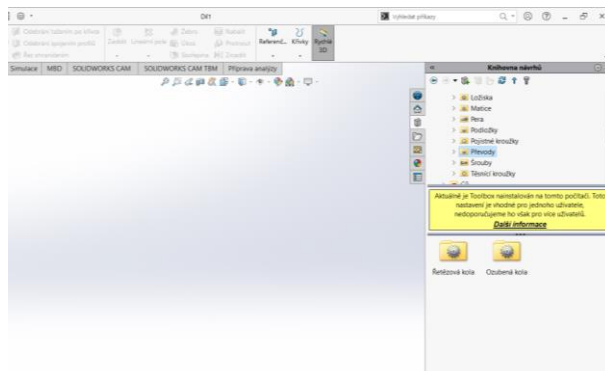
### 1. Install the tool library

- In the settings, choose the option "Add Ins"
- Here select "SolidWorks Toolbox Library" and confirm
- A tool library icon will appear in the upper right corner



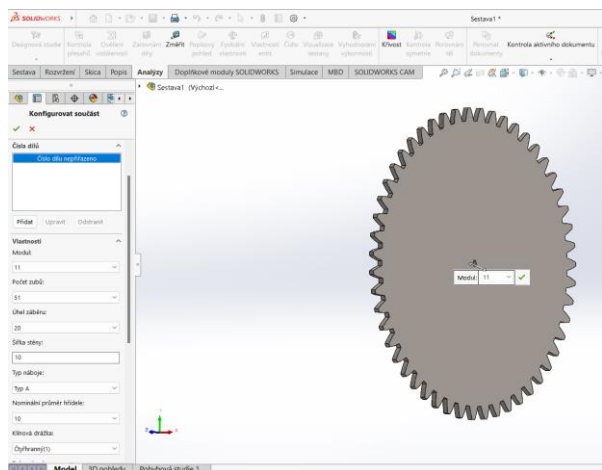
### 2. Open the tool library

- Work in "Assembly"
- Open the tool library
- Select the appropriate library (e.g. DIN)
- Select "Transfers"
- Select "Gears"



### 3. Gear setting

- Drag "Spur Gear" to the desktop
- Set 20 teeth
- pitch circle diameter 50
- angle of engagement 20
- charge type A
- nominal diam. shaft 7mm
- square wedge groove

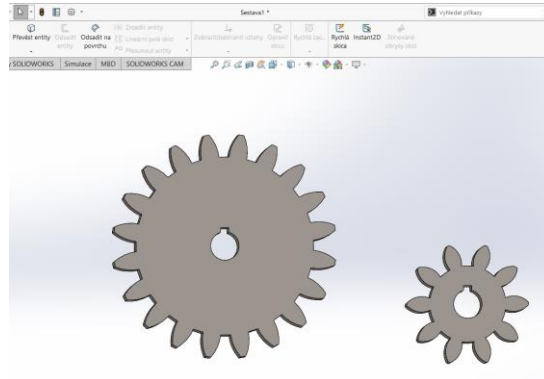


h. gear width 10 mm

#### 4. Setting the 2nd gear

- after confirming the 1st gear, you can insert the second gear
- adjust the parameters to be compatible with the 1st wheel, only with ten teeth
- What will be the nominal diameter of the shaft?

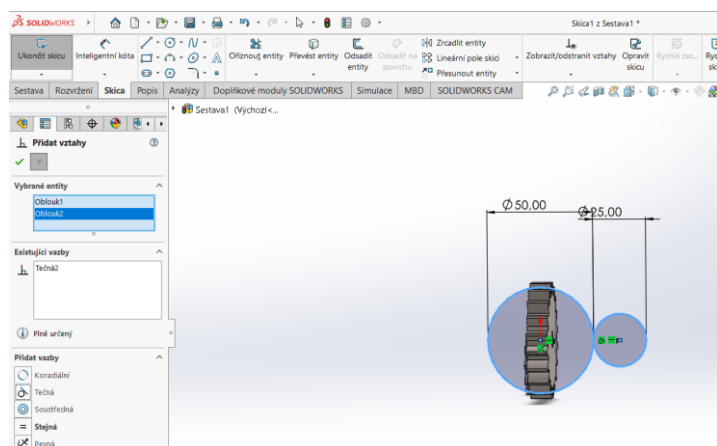
Advice: the module determines the ratio between the nominal diameter and the number of teeth



## Assembly with gears

#### 1. Sketch in assembly

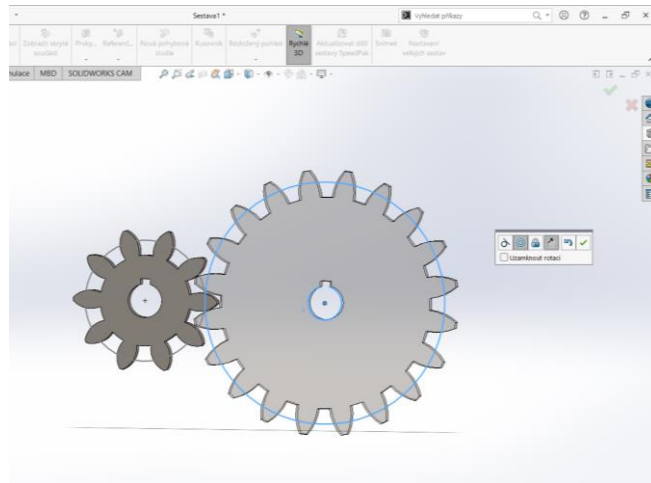
- Draw a new sketch
- Here, create two circles with tangent relations
- Circles must have centers with horizontal relations
- The diameter of one will be 50mm



- 
- e. Diameter of the second  
25 mm
- 

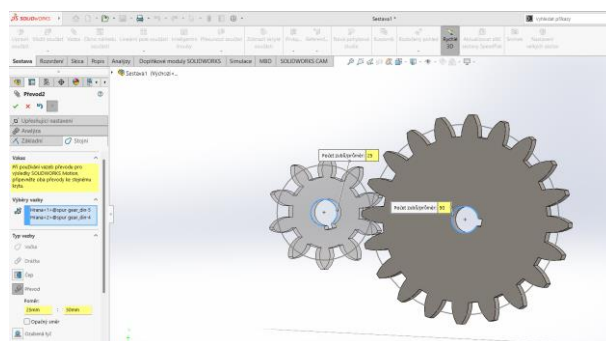
## 2. Basic mates

- a. Arrange the location of the front surface of both wheels in the front plane with the help of mates
- b. Make a concentric mate of gear one and the larger circle
- c. Repeat similarly for gear 2
- d. Align the teeth, so they don't intersect

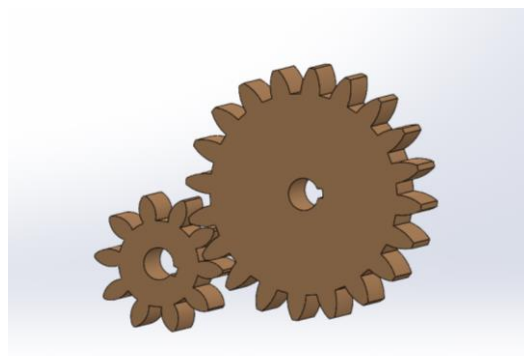


## 3. Machine mate

- a. In machine mates, select “Transfer”
- b. Select the circular parts of both gears
- c. The ratio will be 50:25



- 4. Adjust the resulting assembly to its final form



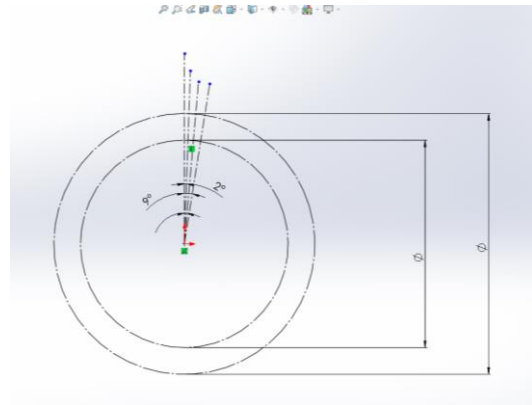
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## Creating your own gear

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### 5. Sketch of support axes

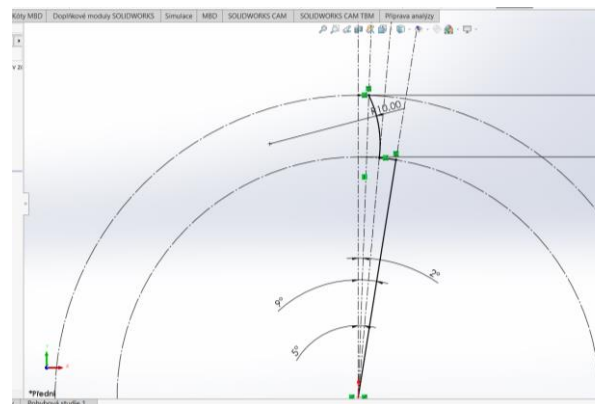
- Create two circles centered at the origin of the coordinate system
- The diameter of the outer circle will be 55 mm
- Inner circle: 43.75 mm
- Convert both to axes
- Create four axes that will have angles from the center:  $2^\circ$ ,  $5^\circ$  and  $9^\circ$

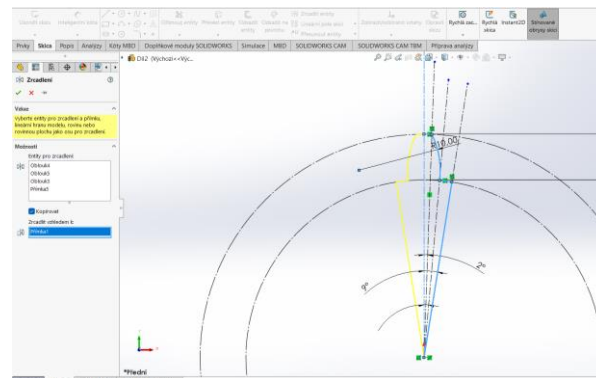


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### 6. Completing the sketch

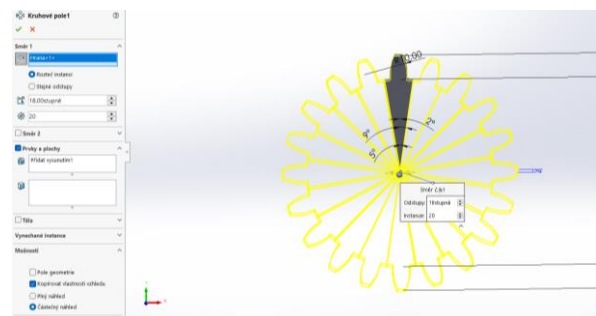
- Draw a straight line along the external axis to the inner circle
- Draw an arc along the inner circle from this external axis to the minor axis
- Draw an arc along the outer circle from the central axis to the minor axis
- Connect the resulting arcs with an arc with a radius of 10 mm
- Mirror the entities using the middle axis





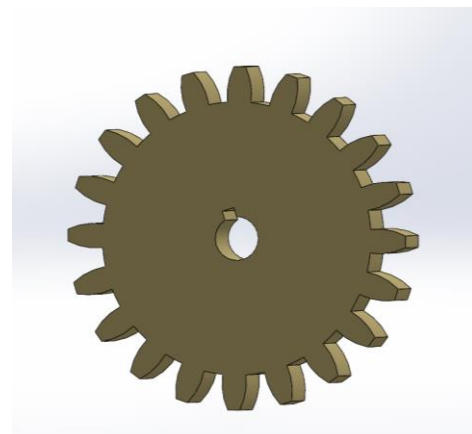
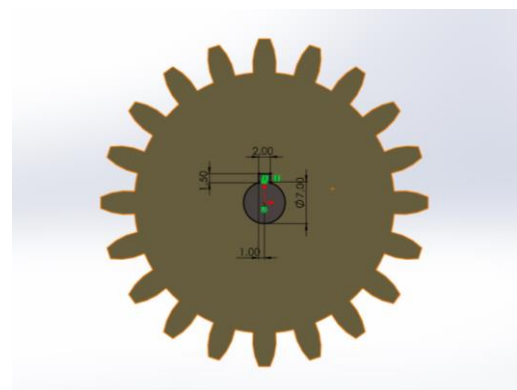
## 7. Creation of the part

- Use the 10mm extension to create one tooth
- Create a gear with the circular field of the tooth
  - angle  $18^\circ$
  - repeating 20x



## 4. Final version

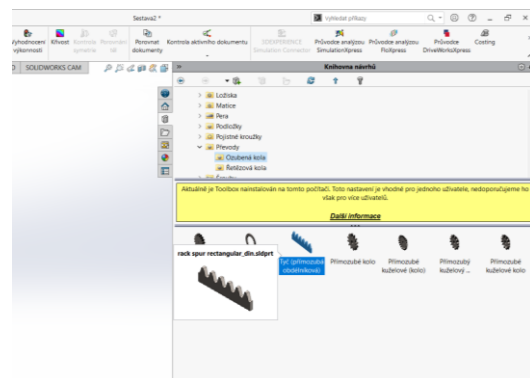
- Create a sketch in the middle of the gear (the center of the coordinate system)
- In the sketch, create a circle with a diameter of 7 mm
- Create a wedge groove with dimensions: 1.5x2x1.5 mm
- Finish the final version using "Extruded Cut"



## Rack and pinion

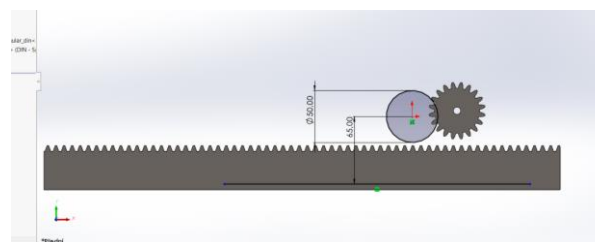
### 1. Setting the rack in the assembly

- From the tool library, select the gear rack from the “Gears” folder
- Drag it to the desktop and set the parameters
- Keep the parameters for the larger gear
- Choose a rise height of 40mm
- Choose a length of 5 meters



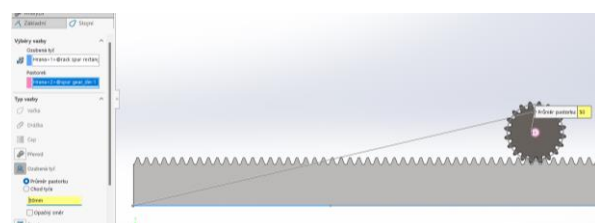
### 2. Creating a sketch in an assembly

- Make a circle with a diameter of 50 mm
- Create a straight line 65mm away from the center of the circle



### 3. Assembly

- Arrange the placement of the front surface of both parts in the front plane. Make a concentric mate of the gear and the circle



- 
- b. Make a coincident mate of the line and the bottom of the rack
  - c. Align the teeth, so they don't intersect
  - d. In machine mates, select "Rack Pinion" (diameter = 50)
- 

