Building a Worm Wheel Model by Simulation of Gear Cutting Process

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ABSTRACT

The article presents one of the ways to build a model of a worm wheel in solidworks by simulating the process of gear cutting using the rolling method.

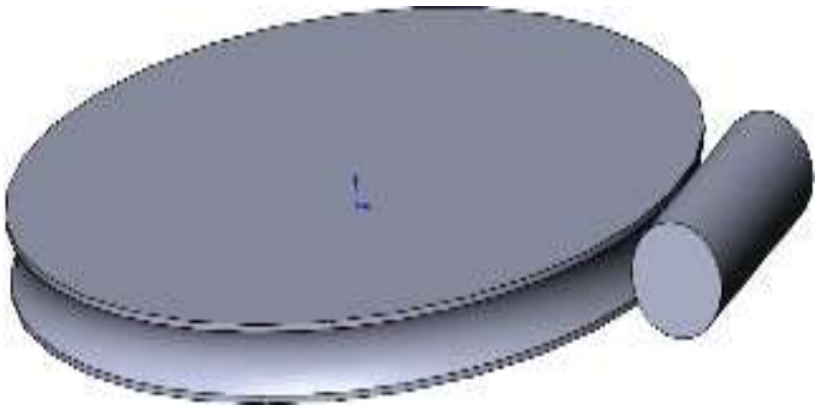
Keywords: Worm, workpiece, modeling, worm coil, simulation, tooth profile, wheel model.

A worm in worm gearcan simply be thought of as a milling cutter with an infinite number of teeth [1,3].

The sequence of construction by this method is described below.

First, it is necessary to build a worm wheel blank [2]. Then, on the "Front" plane, createa sketch in which we build a circle with a diameter equal to the diameter of the worm's cavities d_f, the center of the circle is at a distance of a_w. Then, using the command "Elongated bobbin / base" createm the body of the worm. Stretch by 66 mm using the middle plane. Be sure to remove the mark near the inscription "Combine results". The result is shown in the figure 1

Fig. 1. The intermediate result of the build.



The next step is to model the turns of the worm. To do this, you need to create a reference plane parallel to the plane "To the Right" and 105.5 mm away from thenon-ë. On this plane, create a sketch of the cross-section of the worm. When working with a sketch, it is convenient to use the command "Dynamic reflection of the sketch". For use, you need an axis, first of all, we draw eyo . Then we set aside the segments, two parallel, one at an angle. The constructions are mirrored relative to the axis. Arrange the calculated dimensions. When the sketch is fully defined, it becomes a figure(Figure 2).

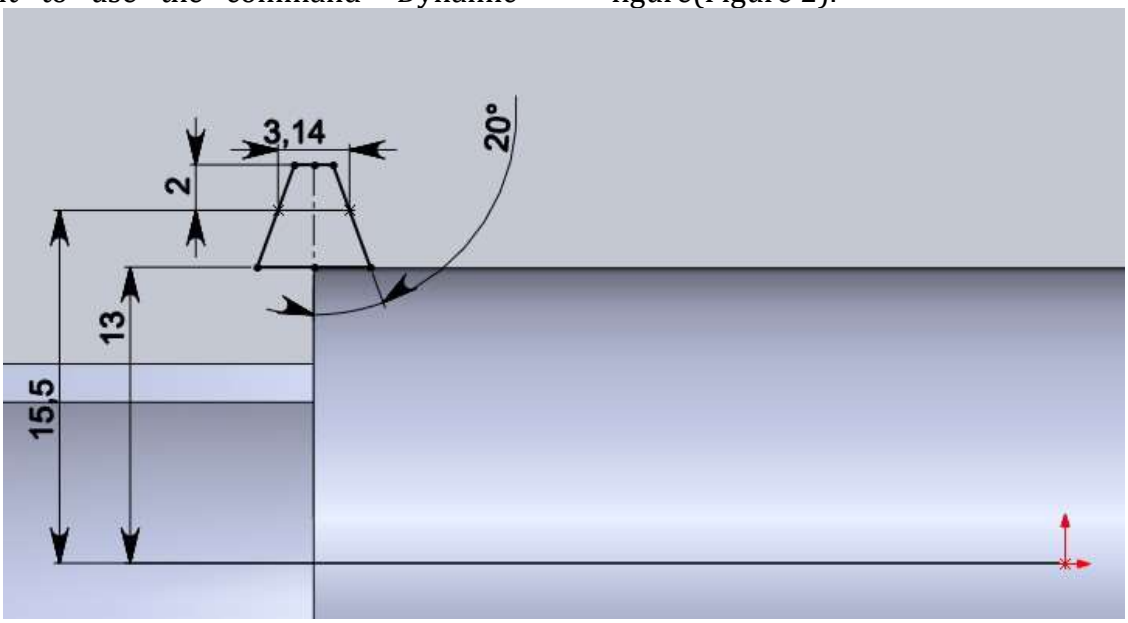


Figure 2 Sketch of worm turns.

Next, you need to create a spiral along which the sketch will be stretched. To do this, create a sketch on the plane of the end of the bobbin. The spiral is defined by a circle. To reduce the time to determine the position of the circle on the plane, use the command

Object Conversion, Thumbnail toolbar. Select the edge on the bobble stretched in the previous operation, creating acompletely definedcircle with the relationship "On the edge".

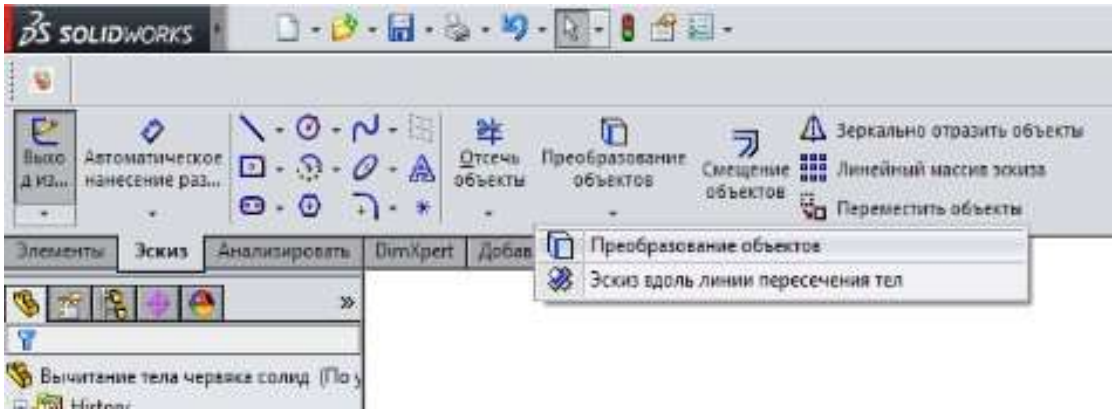


Figure 3. Convert Objects command.

In the Items panel, choose Curves from the Helicoid and Spiral drop-down menu.

The spiral can be determined by several methods, select "Step and Rotation". The pitch of the spiral is equal tothetensile stroke of the worm, the number of revolutions should be such that it completely covers the bobbin (Figure 4) Use the command "Bobby/ base according to the trajectory figure" in the "Elements" panel to simulate the turns of the

worm. As a profile, select the sketch (Figure 2) as the direction - the spiral (Figure 4).

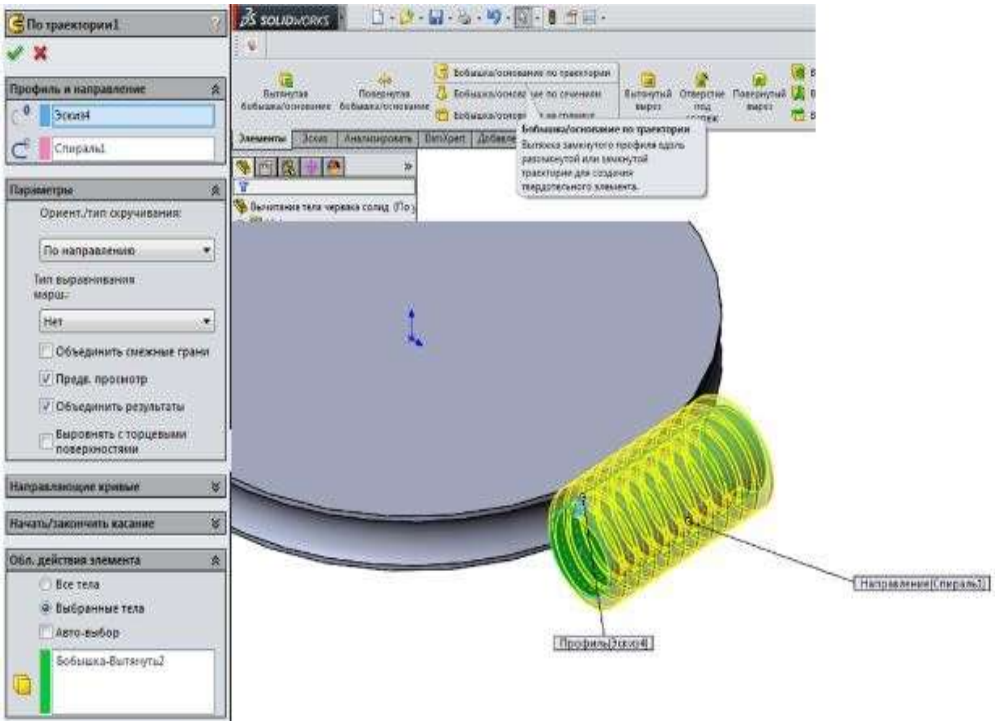


Figure 4. Creation of turns of the worm.

Preparatory operations have been carried out, you can begin the process of forming a depression. You must reproduce the worm and worm wheel roll movements in the SolidWorks environment. It should be remembered that the gear ratio u = 90, that is, when you turn the wheel by 1 °, the worm turns 90 °.

The main commands when simulating the process of tooth cutting of the worm wheel will be "Combine", "Move-Copy".

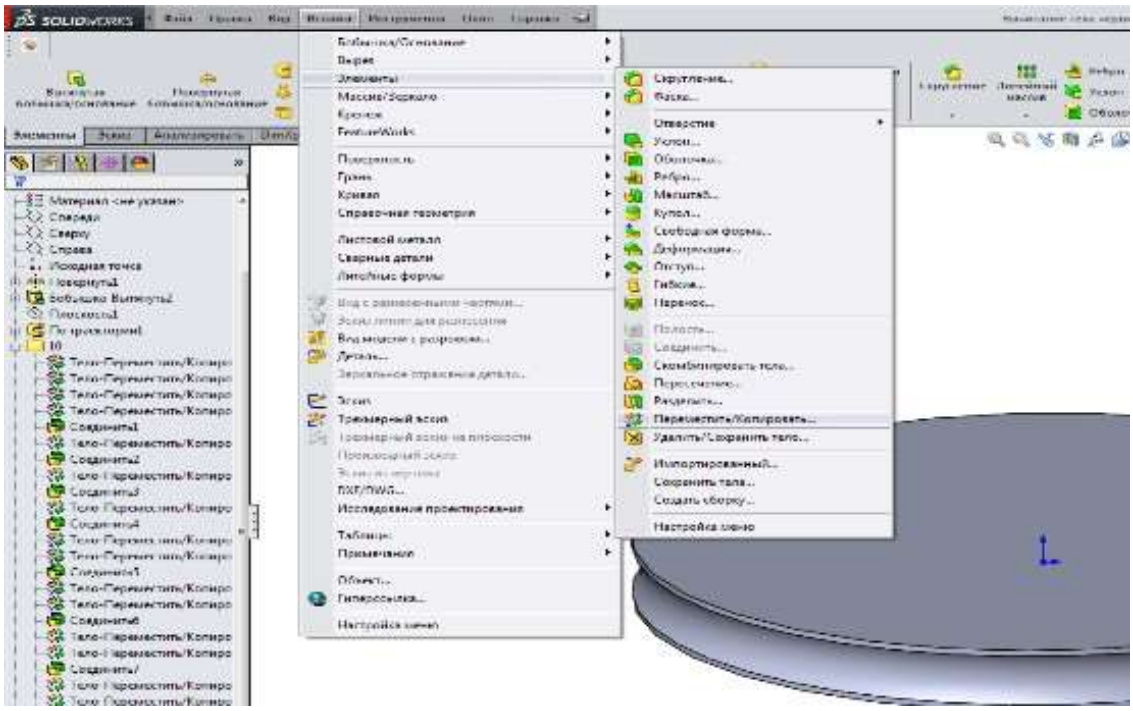


Figure 5. Move/Copy command.

The construction is a cycle that repeats until the hollow is fully processed. First you need to rotate the worm 90° around the axis

Z. Use the Move/Copy command. Select the body along the trajectory, put a mark next

Now it is necessary to subtract the body of the worm from the body of the wheel. To do this, use the Combine Bodies command. As the main body, choose the model of the wheel, the type of construction "delete" Fig. 6.

subtracting the body of the worm, it

to the copy inscription. It is necessary to click on the "Convert" button, then select the "Rotate" tab. The center of rotationis shiftedalong the x-axis by the center of the axle.

disappears, so you had to first rotate and copy it.

The workpiece of the wheel after subtracting the body of the worm must be rotated. Rotate using the Move/Copy command around the Y-axis by 1°.

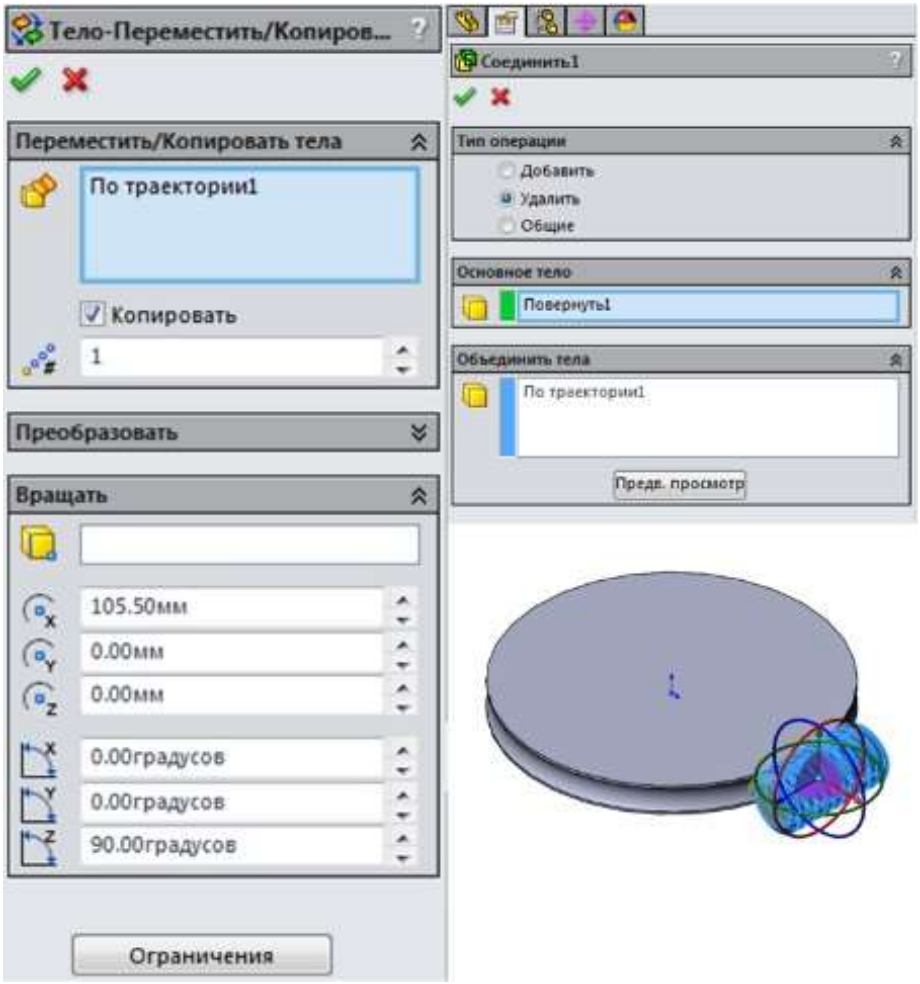


Figure 6: Move/Copy and Combine Bodies commands.

The described three actions are repeated until the tooth profile is obtained. It took 50 cycles for this wheel. Since with so many operations the build tree becomes very large, it is advisable to combine the cycles into folders. Manually building is very monotonous and time-consuming, takes a lot of time. Due to the large number of faces, it is difficult to display the model, the speed of processing the model by the computer decreases.

To obtain a complete model of the wheel, it is necessary to cut a sector with a sliced cavity. In this case, the angle of the sector should be $^{360}{}^{\circ} = 4\ ^{\circ}$. Then 90 the resulting section of the wheel is copied by an array in a circle. This creates90 bodies. They must be combined using the command "Combine bodies".

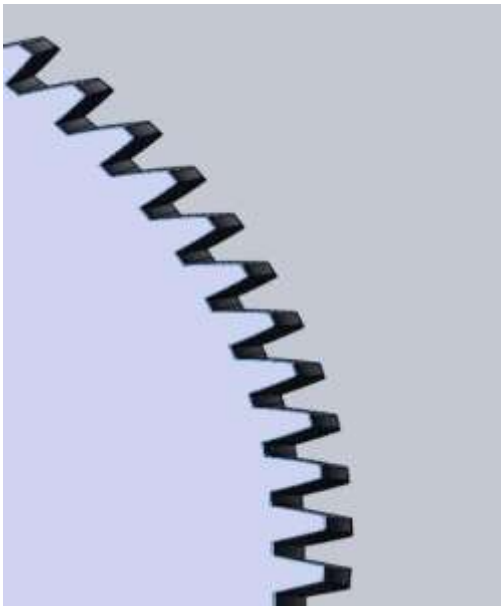


Figure 7. Profile of the teeth of the simulated wheel.

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To obtain the correct surface of the cavity of the worm wheel, it is necessary to simulate the method of running-in. In models of worm stakes designed by this method, the average deviation from the real wheel is 0.051 mm for the wheel obtained by subtraction and 0.089 mm for the worm body obtained by subtraction. Based on the possibility of reducing the labor of the yo smallness for further work, we choose a model of the wheel constructed by subtracting the body of the worm. Based on the possibility of reducing the labor of the yeo.

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