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AUTOMATED GENERATION OF 3D MODELS OF HYDRAULIC COMPONENTS

ZAUTOMATYZOWANE GENEROWANIE MODELI 3D
ELEMENTÓW HYDRAULICZNYCH

Abstract

This paper presents methods for automatic generation of 3D models of hydraulic components. For the purposes of generating models the authors' own software and PTC Creo Parametric CAD system was used.

Keywords: hydraulic component 3D model

Streszczenie

W artykule przedstawiono metody automatycznego generowania modeli 3D elementów hydraulicznych. Na potrzeby generowania modeli wykorzystano autorskie oprogramowanie i PTC Creo Parametric CAD system.

Słowa kluczowe: element hydrauliczny model 3d

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1. Introduction

The paper presents the automated generation of 3D models of hydraulic components. The purpose of the generating models is to visualize and test the correctness of the previously prepared geometric parameters of hydraulic elements. The automation of the model generation significantly speeds up the process of evaluating the correctness of these models.

For the purposes of generating models the authors' own software was used which works with the PTC Creo Parametric CAD system. The PTC Creo Parametric CAD system allows for the creation of complex 3D models that can be saved in various formats, which allows them to be opened and tested in various programs. Also the PTC Creo Parametric CAD system allows for testing models for such attributes as lack of collision between the individual elements or their movement. The authors' own software was created in C++ language on the Visual Studio platform, and it works in Windows systems. It uses a graphical interface with predefined types of hydraulic components.

The paper presents the process of generating only one of the predefined hydraulic components that the created software is capable of. The selected element is a gearwheel pump.

2. Parametric description of a sample hydraulic component – gearwheel model of the pump

The modelling of involute gearing is necessary to use a curve called the involute of the circle. In the PTC Creo Parametric CAD system also at the geometric modelling gearwheels parametric curves can be used conveniently. To draw an involute tooth's profile equations described in the literature [1, 2, 4–6] are used (more important variables are shown in Fig. 2).

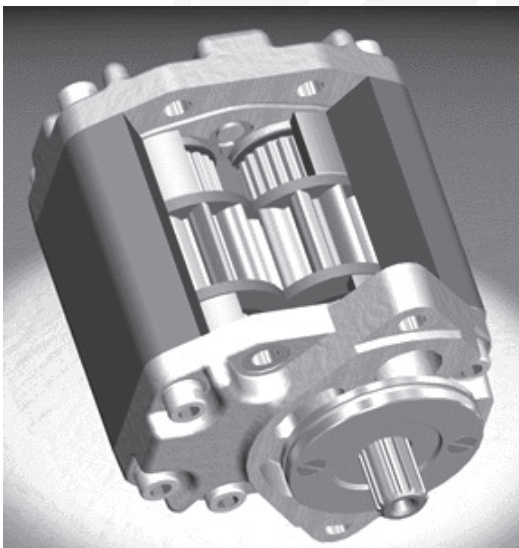


Fig. 1. Parametric 3D model of a gear pump

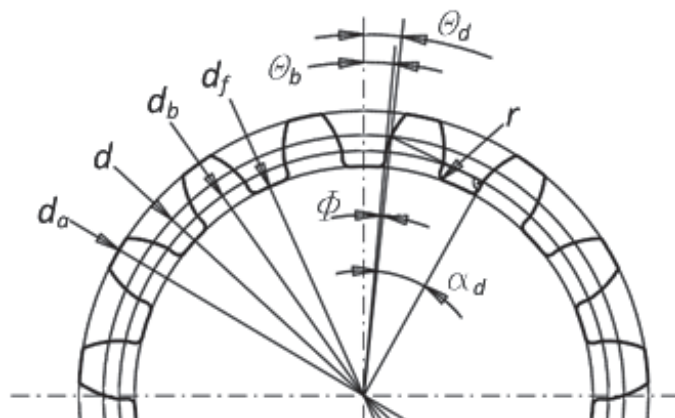


Fig. 2. Parametric 3D model of a gear pump

The main parameters of the gear are as follows:

reference diameter:

$$d = m \cdot l_z \quad (1)$$

basic circle diameter:

$$d_b = d \cdot \cos(\alpha) \quad (2)$$

tip clearance:

$$c = 0.2 \cdot m \quad (3)$$

height of a tooth head:

$$h_a = (y + x) \cdot m \quad (4)$$

height of a tooth root:

$$h_f = (y - x) \cdot m + c \quad (5)$$

where:

- m – module pitch of a gear,
- l_z – number of teeth,
- α – pressure angle,
- x – addendum modification coefficient,
- y – tooth height coefficient.

At generation of an involute curve in the PTC Creo Parametric it is convenient to use parametric notation in polar coordinates [4]:

$$r = -\left(\frac{d_b}{2} + t \frac{d_a - d_b}{2}\right) \quad (6)$$

$$\Theta = \tan\left(\arccos\left(\frac{d_b}{2r}\right)\right) \frac{180}{\pi} - \arccos\left(\frac{d_b}{2r}\right) - \Theta_b \quad (7)$$

$$z = 0 \quad (8)$$

where: θ_b – start of uncoiling the involute from the base circle, which can be calculated using geometrical relations:

$$\theta_b = \theta_d - \Phi \quad (9)$$

$$\Theta_d = \frac{90}{l_z} \quad (10)$$

$$\Phi = \tan(\alpha_d) - \alpha_d \quad (11)$$

$$\alpha_d = \arccos\left(\frac{d_b}{d}\right) \quad (12)$$

where: θ_d – position of the angle of the involute point on the dividing wheel.

In the described example the parameters defining the geometry of the gearwheel are as follows: the number of teeth, module, pressure angle, displacement outline coefficient and height tooth coefficient. As additional parameters can be used: r_f – fillet radius at the base of the tooth and b – gearwheel width.

Due to the record of equations required by the PTC Creo Parametric appropriate substitutions were made, for example, DB for d_b , ALFA for α , etc. Before the introduction of equations for the PTC Creo Parametric it is necessary to put this data into the program. These parameters can be inserted by selecting *Tool/Parameters* or directly as relationships by the command *Tool/Relations* (Fig. 3). Equations and data can be inserted directly or copied from the previously prepared ASCII file. After introducing the parameters and the relations auxiliary curves were prepared. By using the *SKETCH* command, two circles were drawn on the *FRONT* plane to which corresponding parameters DF i DA were assigned. At the plane *FRONT*, when you chose *Datum/Equation*, and after selecting the type of the coordinate system as *Cylindrical*, involute equation was inserted as shown in Fig. 4.

The next step was to generate a cylinder by the command *Extrude*. In the sketchbook an auxiliary circle with a diameter of DA was used as an outline of the base of the cylinder. For this purpose, it was projected by the command *Edge/Use/Single*. The projection of the cylinder was made by entering parameter B as its height (Fig. 4). After generating a cylinder one notch was made, for that purpose *Extrude* command was selected with option *CUT*.

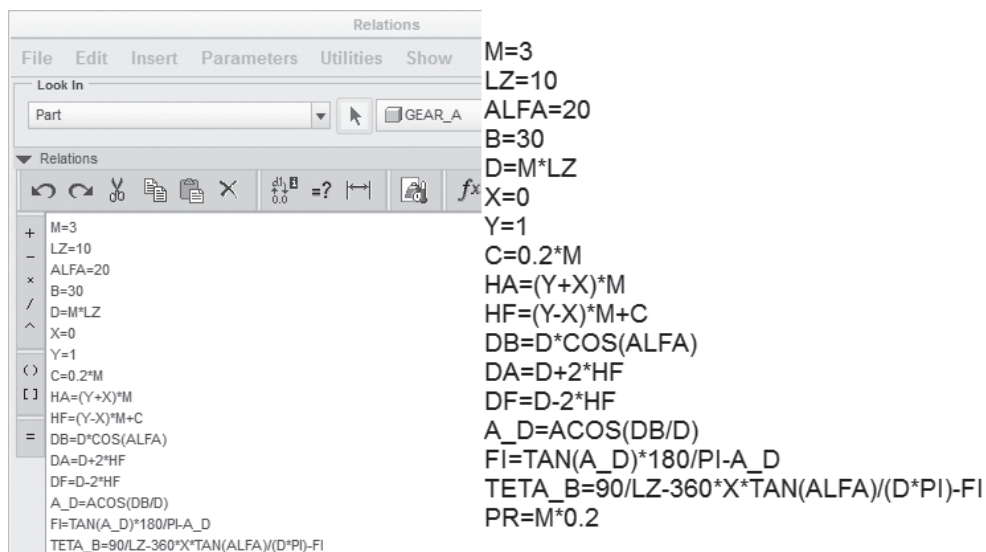


Fig. 3. Window Relations with the inserted equations and relations which describe the modeled gear wheel

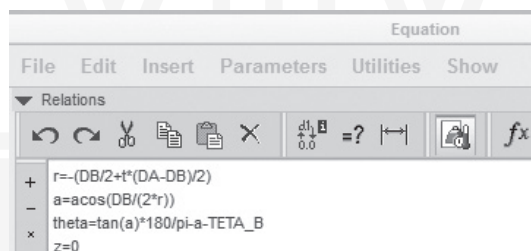


Fig. 4. Window with inserted involute equations in cylindrical coordinates

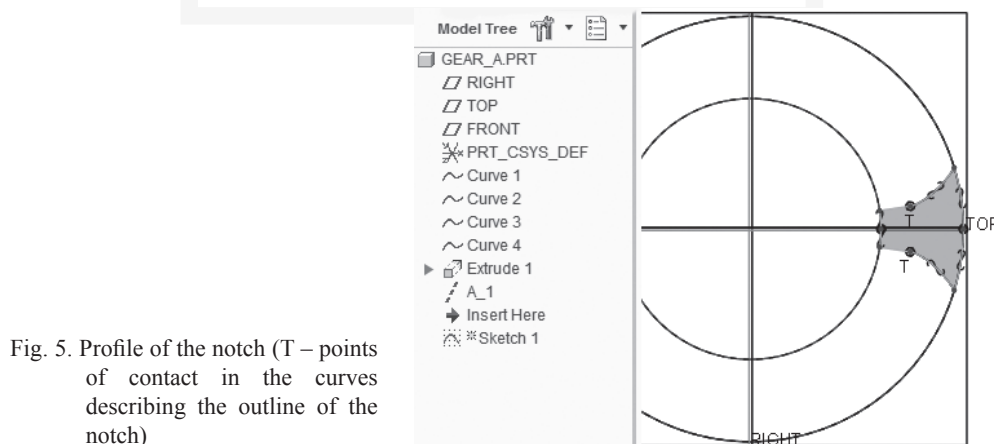


Fig. 5. Profile of the notch (T – points of contact in the curves describing the outline of the notch)

For sketchbook this operation selected the frontal area of the roller. The profile of the tooth notch is made, using mostly previously prepared curves. Using the options *Edge/Use/Single*, circles with diameters DA and DF and involute are projected on the sketchbook. Subsequently, the involute was extended tangentially by a section to the circle DF and rounded by radius PR to the circle DF (Fig. 5). After sketching half of the outline of the tooth its mirror image was made and then the sketch was used to remove the material and receive a one cut notch (Fig. 6).

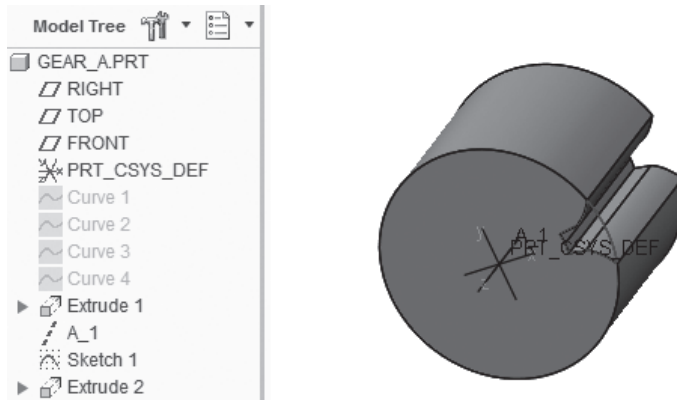


Fig. 6. The model after performing one notch

Pattern command and option *Axis* (wheel pattern) were used for the arrangement on the circumference of a wheel other notches. In the relation window you need to add a variable corresponding to the number of the copies and assign it a value, for example $P16 = LZ$. After refreshing (*Edit/Regenerate*) notches will be replicated all over the wheel (Fig. 7).

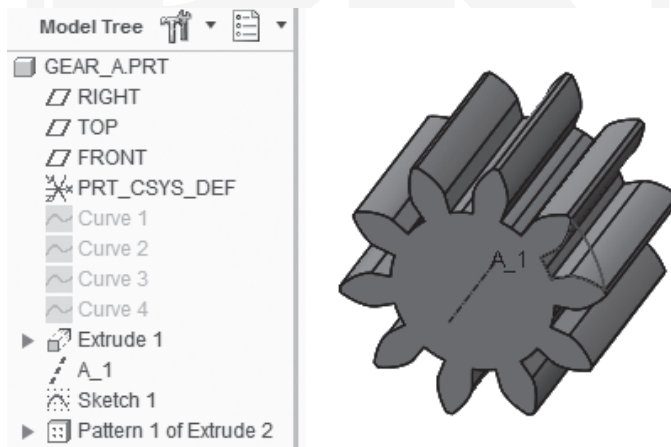


Fig. 7. A parametric model of the gearwheel: 1 – involute, 2 – outline of the tooth, 3 – material in the form of cylinder, 4 – a set of notches

3. Models generator

A program for the automated generation of 3D models was created using the C++ language on the Visual Studio 2013 platform. This software can automatically generate selected models in the PTC Creo Parametric CAD system. For connection and generation of models a Creo Parametric TOOLKIT 3.0 [3] was used. Creo Parametric TOOLKIT 3.0 is a set of software customization tools that, inter alia, allows to automate processes of generating models. To use this software several programming languages can be used (e.g Java or C++). The PTC Creo Parametric CAD system can only work in the Windows systems, therefore the NET platform was used to create this program.

4. Conclusions

This paper presents the automated generation of 3D models of hydraulic components. To generate 3D models was used the authors' own software using C++ and Creo Parametric TOOLKIT 3.0 for working with the PTC Creo Parametric CAD system. The generation process works very well and fast. The generated models are always complete and consistent with the objectives of geometry. One hydraulic element generation was presented in this paper, the gearwheel pump. Further possible improvements of the software are: adding more hydraulic elements options to generate, adding automated tests for created models, and add new types of models.

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