PROCEDURE OF EDITING EXISTING POLYGONAL MODELS BY SOLIDWORKS SYSTEM

Rohozhyna N., Samoilenko I., Meleshko K.
*National Aviation University, Kyiv*
Scientific adviser – Pavlenko P., PhD, Associate Professor

The ability to use 3D printing technology directly depends on the quality of the input geometric model. It is usually difficult to see small defects, missing triangles, convex and other deformations of the grid in slicers [1]. There are online services stlfixers that fix network models. They quickly fill in defects, but are limited by the size and number of triangles of the model. In this case, slicers "do not see" the smallest holes in the grid or one / two triangles that protrude beyond the shape of the part, such defects are simply not printed, so in most cases stlfixers are quite enough.

To eliminate significant defects or to make changes in the geometry of the model, the following sequence of actions is proposed using solid-state design tools.

The first step is to check the geometric model. The model is opened in the slicer program and the layer-by-layer printing process is checked. This procedure allows you to understand the defects in the model and what points will need to be addressed. Next, the polygonal model opens in the SolidWorks system, while immediately converting it into a solid (or, if necessary, a surface body). The sequence of actions in the system is shown in the figure below.
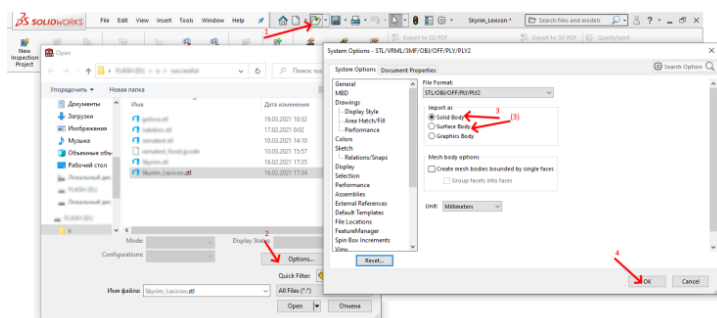

The next step is to recognize the basic geometric components of the model. The process of recognizing geometric components can take a long time, so it is desirable to use a powerful processor. Instead, operations such as extrusion, rounding, and chamfering are now available, and you can edit the dimensions of the parameterized model.

It should be noted that in some cases the system cannot recognize the elements of the model and the automatic recognition operation may end with an error message. In this case, you should choose the faces for recognition yourself. Then editing the model becomes a bit more difficult.

Then, since the model is now solid, it is possible to use existing vertices of triangles to construct planes and sketches. In some cases, planes parallel to the screen are created to simplify constructions (so as not to move the model to the origin and not to create planes depending on it).
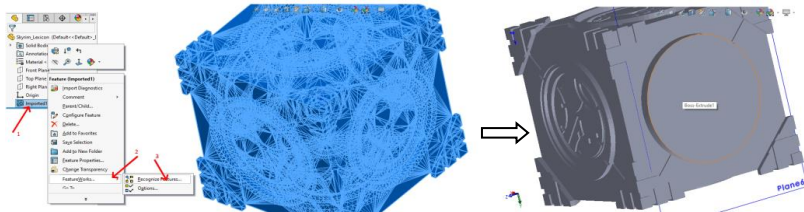
Then, with the help of basic operations, solid elements are created that allow fixing the shortcomings of the initial model. The image above shows an example in which the relief is removed from the face, which is used as the basis of the model during printing. In addition, the thickness of the protrusion is aligned with the side inserts, to eliminate the need for unnecessary support in the printing process. Very often there is a need to scale the model. Slicers also have this function, but the scaling in them is controlled only by the external dimensions of the model. The use of a solid model makes it possible to scale individual elements and use an uneven scale. The next step is to save the model in STL format, controlling the quality of the result.
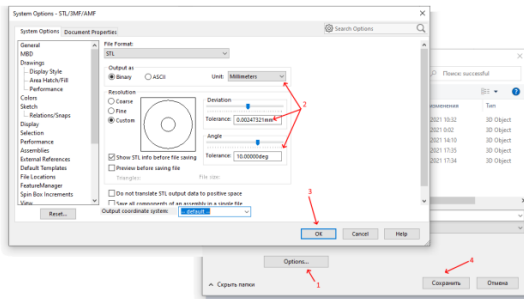
The disadvantages of using SolidWorks to prepare models for printing include certain limitations on the complexity of the model, and the peculiarities of working with solid mesh models, in which, for example, cylindrical surfaces are represented as faceted instead of continuous (smooth). The latter, however, in no way limits the possibility of using other tools of the system to prepare the model.

References:
1. 2021 Best 3D Printer Slicer Software [Electronic resource]. - Access mode: https://all3dp.com/1/best-3d-slicer-software-3d-printer/