Automation methodology of the surface creation of products thin-walled shell

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Abstract. To succeed in the process of educational design, it is necessary to understand and formulate the principles that underlie the creative activity of students and are conditioned by the patterns of technique and technology development. In the process of educational design, opportunities arise for students to use their knowledge about the mechanical, physical, chemical, technological and operational properties of products for the manufacture of an educational design object, as well as for the application and improvement of their practical knowledge and skills. In this work, the authors set themselves the task of resolving the arising methodological disagreements in the process of teaching junior students solid-state modeling of the constituent parts of complex products. It has been established that before proceeding with the design of a unit, it is necessary to analyze it, select individual surface areas and outline methods for their construction, including the construction of complete sketches of individual elements.

1. Introduction
Using the example of the proposed product shown in figures 1, 2, we will consider the construction of the surfaces of individual elements that are most often found in products of heavy engineering. For this purpose, we will divide the entire product into three main sections: body, handle, and attachment. Each of them is a collection of different complex and simple surfaces. Since each area is modeled using specific commands, the process of creating the entire model consists of three steps, corresponding to the three main parts of the product. For each element, a sketch of the selected surface is drawn, then the surface itself is created, which is a thin shell of the product, and then all the created surfaces are combined, mated and converted into a solid phase [1].

2. Construction methodology
After decomposing a complex technical product, let us consider each of the three main elements: the body (1), the handle (2) and the attachment (3) (figures 1 and 2) [2].

The necessary assumption is that at this stage the modeling may not be connected with the final material of manufacture. To date, experimental materials consisting of composites seem to be more promising due to their greater wear resistance and less influence of aggressive media, as, for example, it happens with industrial rubber goods [3, 4].
The use of standard and unified fragments is widespread in mechanical engineering [5], but in this case we considered the design of completely new surfaces to provide the methodological base.

The design of new parts or assembly units can be justified by structural or technological necessity, in cases where flotation increase or improvement of any other properties is required [6, 7].

Body. In the SolidWorks program, the creation of model surfaces, as well as the creation of solid parts, begins with sketching the outline of the individual elements of the product in the Sketch mode, and then it is put in the Surfaces mode to obtain a volumetric image of the shell. Creation of the initial body surface. As a result of applying the full set of geometric constructions of the sketch and editing operations, the sketch of the body was created, shown in figure 3.

Using command Extruded surface a body shell is created (figure 3 a). It should be noted that when any command is turned on, the properties panel of this command opens on the interface, in which the program asks the conditions for creating an object, for example, such as the location of the object, its parameters, with which construction elements it is necessary to carry out transformations, etc. (figure 3 b). In this case, in Direction 1 section, the parameter to the given distance - 400 mm is set [8].

Creation of the location of the body back wall and a place for the handle After creating the surface of the body, spaces are created for the body back wall and for the upper part of the handle (figure 4).
A sketch of the location of the back wall of the body and the place under the handle. Right-click points at the plane on the Right in the Design Tree. In the window that opens, create sketch button and the icon Perpendicular are selected. This allows you to position the created sketch in front of the observer. First, the sketch of the back wall of the body is drawn. To do this, by opening the command Line a straight line is drawn at the right end of the body from a point at height of 140 mm and at an angle of 35 °. The place under the handle is drawn according to the sketch shown on the right in figure 5. After creating a sketch, you need to exit from it [9].

Cutting. When cutting a part of the body, the command Cut Surface is used in the Surfaces menu. On the properties panel of this command a dot is placed in the active window Cut tool, in the circle of the Delete selected parameters line. The part of the surface to be removed is indicated on the body. To do this, the cursor is moved over the cut area, then this surface will be reflected in the Selection section in the Cut tool window. Then the command ends (figure 6).

The surface of the body back will connect two edges that are located on two planes at an angle to each other. The bottom edge of the back of the body has already been drawn, and the other must be
created as an arc on reference Plane 1 through the centerline that connects the two protruding points of the body. In this case, Plane 1 is located at an angle of 10° to the plane Top in DK.

Creation of the centerline. Construction of reference Plane 1 begins with a centerline. To do this, activate the command Reference geometry on the Surfaces toolbar. In the drop-down tab, the line Axis is indicated. The line Two points / vertices is located in the properties panel of the command Axis. Two protruding points are selected and an axial line connecting them is drawn (Fig. 7).

Construction of the reference plane. Next, the reference plane 1, passing through the center line is constructed. The Reference Geometry tab opens again and the command Plane is selected. This command will ask you to give it three reference links to define a plane in space. In this case, we need only two. The newly created centerline is selected, as shown in Figure 5. Select the plane Top as the second reference link. For this, DK is opened and the plane Above is indicated. On the properties panel of the command the line Defined turns green. This means that the plane is fixed in space.

However, we need to create a plane at an angle of 10° to the plane Top. To do this, the value At angle is selected in the Second link and is set it at 10°.

![Figure 7. Construction of the reference plane 1 (A - axis, B - surface).](image1)

Construction of the arc on the reference Plane 1. Plane 1 is indicated. In the window that appears, next to the mouse, the command Create Sketch is selected. For convenient orientation of the sketch in space, you can press the key combination Ctrl + 8. (figure 8)

Using the command Arc by Three Points a curve that connects the two protruding points of the body is drawn. The arc must have ties Tangency with the edges of the body. If they are not there, add, i.e. enable the command Add Interconnect, specify the arc and the tangent to it the edge of the body, as shown in figure 8.

![Figure 8. Sketch of the arc of the back wall of the body.](image2)
Creation of the surface of the back of the body. Since the two edges are ready for the shell creation, the Surfaces toolbar opens, the command Fill surface is activated, and the Patch is created. The edge in the form of an arc on the reference Plane 1 and all end edges of the body are indicated (Fig. 9 a). The sections of the body of each edge are colored blue, and they are reflected on the properties panel of the command in the window Patch border (figure 9).

![Figure 9. Model of the surface of the back part of the body (A-axis).](image)

In the window Setup for edges, you define the contact condition. Replace the mate condition Contact with Tangency.

The surface of the body under the handle should be built between the two edges at the bottom and top of the handle. They are the place where the handle and body meet. Since the handle must be round, it is necessary to have boundary edges at the joints in the form of semicircles or arcs.

For this, in the lower part of the connection of the lower part of the handle and the body, on the reference Plane 1, an arc at 3 points of the radius of 130 mm is drawn (figure 10).

![Figure 10. Sketch of an arc on the reference Plane 1 (A-axis, B-surface).](image)

To construct the edge of the joint of two elements of the product, the reference Plane 2 is created in the upper part of the body, on which a sketch of the arc at three points of the radius of 45 mm is drawn (figure 11).

![Figure 11. Sketch of the arc on the reference Plane 2 (A-axis, B-surface).](image)
The surface under the handle is filled using the command Fill Surface. All edges describing this cavity are indicated then.

The designed product can be adapted for manufacturing using 3D printing or similar additive technologies, thanks to the use of modern design tools [11, 12].

The use of the above methodology for teaching [13] can improve the level of training of specialists in disciplines requiring knowledge in the field of engineering graphics, computer graphics and descriptive geometry. For practical application, it is necessary to take into account factors based on statistical modeling but it can often lead to incorrect results. The functioning of organizational and technical systems can be characterized by uncertainty [14].

3. Conclusion
This paper presents a creation technique of various types of surfaces for aircraft products using the example of the proposed product. Having mastered the methods of constructing surfaces, the user will be able to analyze the product, break it into parts and fragments, select and apply methods for constructing partial fragments, and, thus, construct and design all products as a whole.

The main result of the application of techniques for constructing complex surfaces is the expansion of users training area due to developing skills and experience in creating complex surfaces in the SolidWorks geometric modeling.

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