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METHOD OF CREATING THE SURFACE OF THE BODY OF PRODUCTS

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Abstract

Awareness and formulation of the principles underlying the creative activity of students and conditioned by the laws of development of technology and technology, the process of creative activity are necessary to achieve success in the process of educational design.

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 used to manufacture an educational design object, as well as to apply and improve their practical knowledge and skills.

The objective of this study is to resolve the arising methodological differences in the process of teaching solid modeling of the components of complex products to students in the first years of higher education. It has been established that before proceeding with the design of the product, it is necessary to analyze the product being created, select individual surface areas and outline methods for their construction, including the construction of complete sketches of individual elements.

Introduction

Using the example of the proposed product shown in Fig. 1, 2, we will consider the construction of the surfaces of individual elements that are most often found in products of the aviation industry [1] and heavy engineering [2]. We have divided the entire product into three main areas for this purpose: bodies, handles and attachments. Each of them is a set of different complex and simple surfaces [3] [4]. Since each area is modeled using specific commands, the process of creating the entire model consists of three stages, corresponding to the three main parts of the product. A sketch of the selected surface is drawn for each element, then a surface 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 [5] [6].

Next, let's take a look at sketching the Body part for an example.

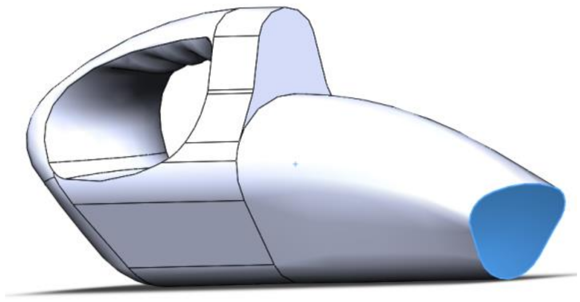

Figure 1. Product with a complex surface

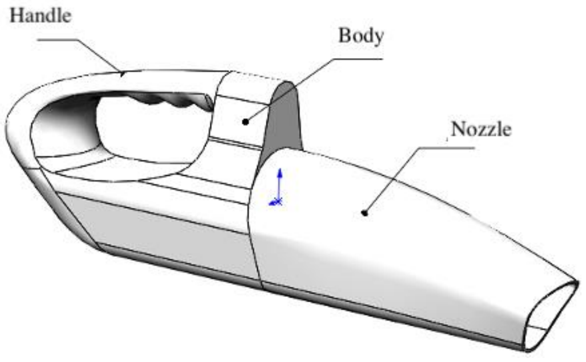

Figure 2 Three main parts of the product: body, handle, nozzle

Sketching methodology

Centerline sketch. Sketching usually begins with construction of construction lines, for example, with a centerline. To do this, activate the command - **Line** and select the line - **Center line**, which is drawn vertically, as shown in Fig. 3.

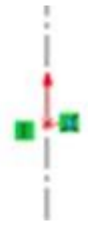

Figure 3.Centerline.

Two green squares, **near the origin**, are the display of geometrical references [7], which the program automatically superimposed on this centerline. A vertical line on a green background is a **verticality relationship**, which limits the rotation of the centerline in space and tells the user that this line is vertical. The intersection of two lines on a green background is a coincidence relationship, which in this case tells the user that the origin and centerline are the same, or that the origin belongs to the given centerline. Both links are **green**; this means that these geometric relationships are fulfilled. **Yellow color** - executed with an error or partially executed.**Red** - not executed.

If the centerline was not drawn vertically or through **the center of coordinates**, as shown in Fig. 4, then this must be corrected. In this case, the axial relationship is superimposed [8] [9].

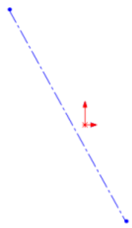

Figure 4. Oblique axial.

Note: *There are several options for creating relationships, but this tutorial will cover only one of them.*

To give the center line a vertical position, it is necessary to select it with the left mouse button (LMB) to begin with. As soon as this is done, **the panel of properties** of this line will open (Fig. 5 a). In this panel of line properties in the Add relationships section, select **the vertical** geometric relationship. After that, the centerline will change its orientation and a green symbol of this geometric constraint will appear (Fig. 5 b). ab

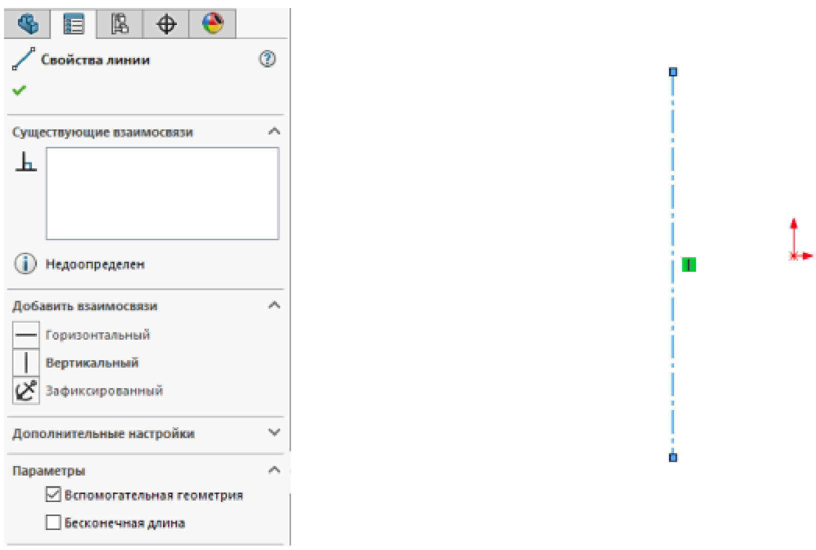

Figure 5 a) **Properties panel** of the **Line properties command**, b) Vertical centerline.

To connect the centerline with the starting point, select the axial LMB, and then, holding down **the Ctrl** key on the keyboard and not releasing it, select LMB the starting point, which is colored red and has two perpendicular vectors symbolizing this working coordinate system. After such a choice, **the properties panel** will open (Fig. 6 a) for the two selected objects. It is necessary to specify the **Coincidence** relationship and the centerline will take the desired position in the **Add relationships section** [8] (Fig. 6 b).

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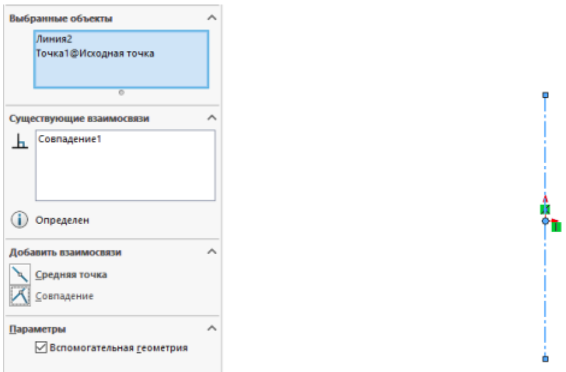

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Figure 6 a) **Properties panel** of **the Selected objects** command, b) Vertical line.

Note: *holding down the **Ctrl** key allows you to select two or more objects at a time. If you click on the centerline, and stomp on a point, without holding down the **Ctrl** key, then when you select the starting point with the LMB, the selection of the centerline will be canceled, and only the starting point will appear in the selected objects.*

Sketching an arc R40. Sketching a body begins by creating an upper arc and setting its parameters. The location of the center of the arc is selected on the centerline. A horizontal centerline is drawn anywhere and its distance from the center point to the horizontal centerline is set with the command - **Automatic dimensioning.** In the window that appears, the size is set to 150 mm.

Select the command - **Arc Center** LMB and indicate the center of the arc and draw an arc in any form with the mouse. Turning on the command (Fig, 7) **Automatic dimensioning**, the cursor indicates the arc, in the window that appears, the program presents the natural size, which is replaced by the arc size of 40 mm.

This element is often used in aircraft construction. Arcs of both constant and variable radius are used [10].

Also, this tool can serve as a description of the law of motion of a point on a plane [11].

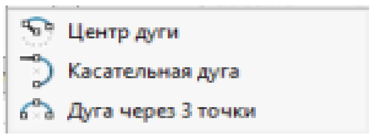
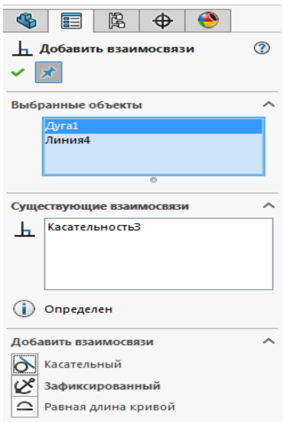


Figure 7.**Arc Center** command.

Creates an arc tangent sketch R40. Next, you need to build a tangent line to the arc. To do this, select the command - **Line**, which is drawn arbitrarily on the free area of the drawing.

To set the orientation of the line in space, the command - **Automatic dimensioning** is selected, the drawn line is indicated by LMB and then the axial one. In the window that appears, the angle of the line's position in space is set - 12°. After that, it will take the required position in the work area, as shown in Fig. 8 a.

For the line to be tangent to the arc, it is necessary to impose a geometric constraint on it - **Tangent**. The menu line **Relationships** turns on, - **Add relationships**, select objects are indicated by LMB - arc, then straight. In the properties panel in the section - **Add relationships** (Fig. 8 b), suggestions will appear, from which you must select the relationship - **Tangent**. After connecting, the free ends of the lines will be found. You can trim them using the **Trim Objects** command.



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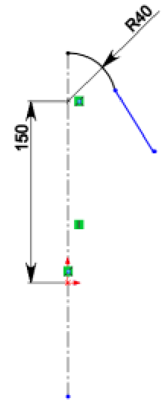
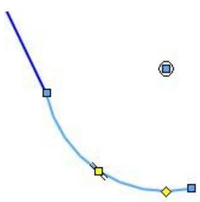


Figure 8 a) **Properties panel** of the **Add relations command**, b) Sketch of an arc R40 and a tangent.

Sketching an arc R25 and a tangent to the arc. The R25 arc is drawn from the free end of the already drawn line using the command - **Arc through 3 points**. To do this, on the toolbar in the **Sketch** menu, expand the Draw arcs command and select - **Arc through 3 points**. Having selected the command, first the free end of the line is indicated by LMB, and the remaining two points are placed in an arbitrary position so that the convexity of the arc looks towards the center line (Fig. 9 a).

The point of a straight line and an arc is fixed, in a known way, with the connection command **- Tangent** and the size of the arc radius is set to 25 mm.

Using the command - **Automatic dimensioning**, the inclination of the straight line at an angle of 23º to the axis is set.



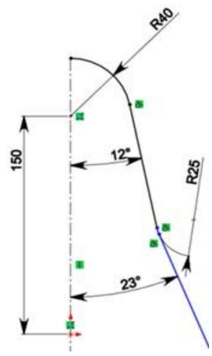


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Figure 9 a) Sketch of arc R25, b) Sketch of a part of the body with arc R25 and tangent.

Sketching an arc R85. An arc of radius R85 is drawn using the command - **Arc Center**. When constructing an arc from the center of coordinates: the first point is selected - the center of coordinates, the second is the free end of the line, and the third point is placed in an arbitrary position, as shown in Fig. 10, then the relationship - **Tangent** is applied. Sets the arc radius to 85 mm.

The free end of the arc is finally fixed, measuring 83° between the centerline and the free end of the arc [12]. To do this, select the command - **Automatic dimensioning,** indicate the starting point of the coordinate system by LMB; thus, we tell the program that the angle will be measured relative to this point, then the upper point on the center line and the free end of the arc are selected by LMB, the required size of 83° is placed between them.

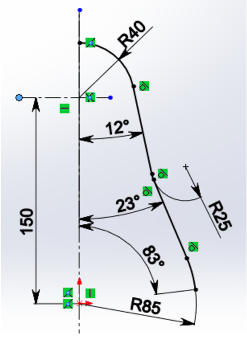

Fig. 10. R85 arc and tangent sketch.

Sketching an arc R90 and a tangent. Next, with the command - **Arc through 3 points,** you need to build a tangent arc and set the size of its radius to 90 mm.

A line is drawn from the free end of the arc. The relationship - **Tangent** is applied to it and to the arc. The free end of the line is fixed at a distance of 40 mm from the axis, as shown in Figure 11.

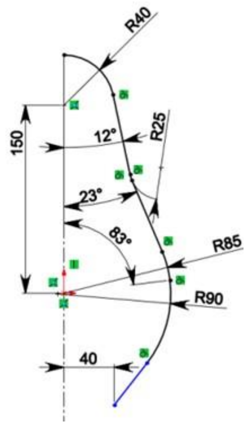

Figure 11.Sketch of arc R90 and tangent.

Sketching the last arc.The last arc remains: Using the - Arc center command, an arc of arbitrary radius is constructed, the center of which is at a distance of 100 mm from the center of the coordinate system, as shown in Fig. 12 a.

Next, you need to connect the free ends of the arc and line. To do this, LMB specifies the end point of the arc and the end point of the line, and in the opened properties panel of the Add relationships command (Fig. 12 b), select the relationship - **drain.**

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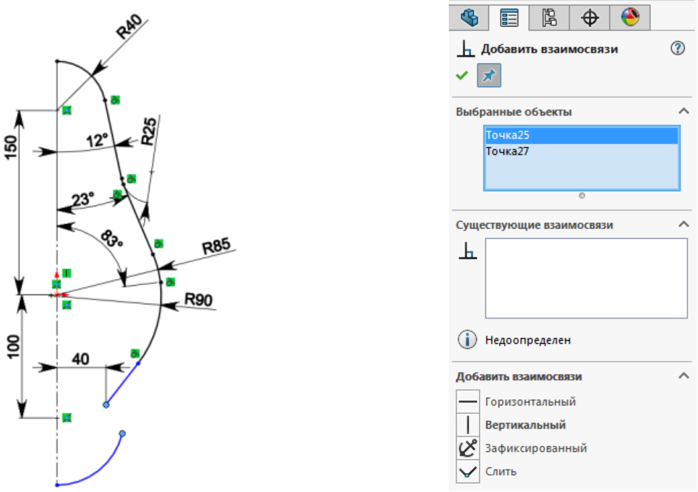

Figure 12 a) Sketch of creating an arc,b) **Properties panel of the Add relationships** command.

After merging the points, you need to set the relationship - **Tangent**, between the arc and the line to complete the construction of half of the sketch, as shown in Figure 12 a.

Complete axisymmetric sketch

To get a complete outline of the sketch, you need to flip the created half about the centerline. To do this, on the **Sketch** toolbar, select the command - **Mirror objects**. LMB will be asked to indicate first the sections of the sketch for mirroring (they will be reflected in the **Objects for mirroring section** on the property panel of this command), and in the second section, line below - **Mirror relative** and with the mouse show the vertical axis - the element relative to which the objects are mirrored sketch (Fig. 13 a). Having filled in all the active windows of this command correctly, you can see the following result, shown in Fig. 13 b [12]

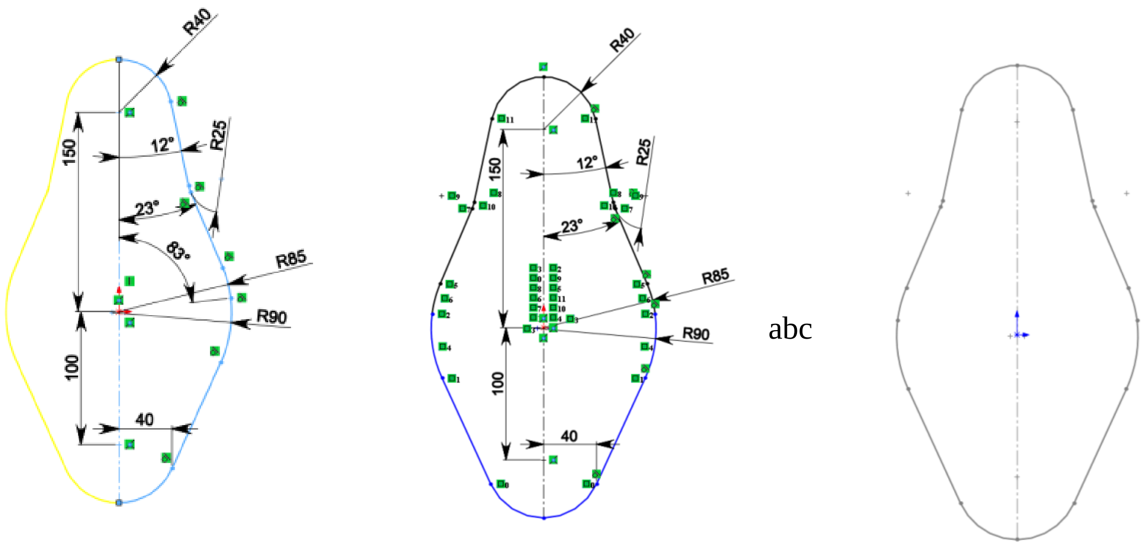

Figure 13.a) Sketch of half of the body and its mirror reflection, b) Complete sketch of the product body with dimensions, c) Sketch of the body outline.

The sketch shows not only the dimensions of the construction, but also the links of the geometric constraints. In order to hide them, you need to go to **Hide / Show Objects** in the menu located in the work area - views panel, find the line **Show sketch relationships** and turn off this command. The icons in the sketch will disappear (Figure 13 c).

In a particular case, after the completion of this command, the sketch lost one dimension - 83°, because the point from which it was plotted - the end (end) point of the upper arc, changed and became the center of the arc, and not its end, and therefore the size was destroyed (see Figure 13 a). To restore the sketch, you just need to re-place the lost size, as it was done before. LMB on the **Sketch** toolbar the **Exit Sketch button**.

The sketch for creating the appearance of the case is completed (Fig. 13 c)

It is possible to create sketches in another way. Without taking the mouse away from the picture, the outline of the sketch is drawn, and then the dimensions of each of its sections are put down.

Result

Sketching a body.Before starting to create a part, namely a sketch of its body, its geometric features are revealed, such as symmetry, repeating elements, the location of the sketch plane in space and others, without taking into account the material of manufacture of the part [13], its strength or resistance to fracture [14] and climatic [15] operating conditions.

The use of automation tools allows you to correctly select the sketch primitives, draw a sketch faster and better, suitable for further manufacturing using additive technologies [16] and repeated use within the framework of the unification of production [17].

As a result of the decomposition [18], the selected element of the product “body” has satisfactory geometric characteristics and is symmetrical about the vertical plane, therefore the sketch of the body profile will also be symmetrical and should be located on the **Front plane**, while the center of the model should coincide with the center of the interface. Since the sketch is axisymmetric, using this property, you can draw a sketch on one side of the centerline, and then, using the **Mirror command**, get a complete sketch of the part body.

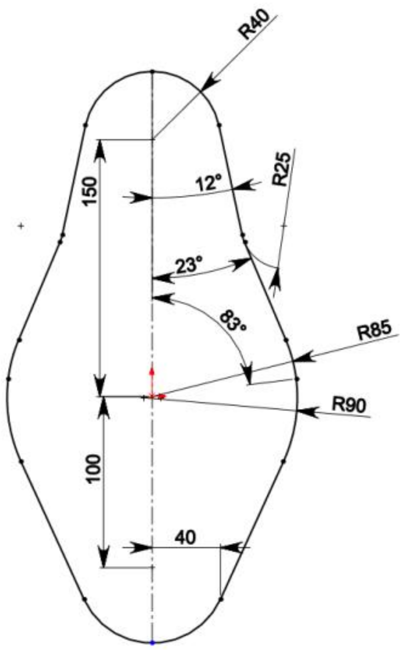

Figure 14. Sketch of the body.

The body sketch profile consists of separate sections of arcs and straight lines, the dimensions of which are shown in Fig. 14.

The sketch of a complex product body is not stochastic [19], has a variable radius and is fundamentally different from the construction in the Descriptive Geometry course, therefore, for the process of creating a complex axisymmetric sketch in the SolidWorks software environment, it is necessary to learn how to use virtual geometric tools.

After turning on the program and selecting a job in the **Detail section**, the working field of the interface opens. On the **Design Tree** (DT), the plane in **Front** is selected and, using the **perpendicular** icon, is placed in front of the user.

Conclusion

The proposed work considers the methods of constructing complex surfaces of aircraft products using the example of the proposed product. Having studied the methods of constructing surfaces, the user will be able to analyze the product, break it up into sections and fragments, select methods for constructing individual fragments, and, thus, design and design all products as a whole.

The main results of the application of techniques for constructing complex surfaces are reduced to expanding the area of user training by gaining skills in creating complex surfaces in the SolidWorks geometric modeling system.

Scientific and methodological novelty lies in the development of a new approach to the creation of complex surfaces when studying the course "Computer Graphics" and is associated with the development of design thinking in design already at the initial stage of training in technical universities of design specialties.

The proposed technique for constructing complex surfaces was tested at the lectures of the Department of Engineering Graphics.

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