3D MODELLING A TRUCK USING SOLID EDGE

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ABSTRACT

The purpose of this paper is to give a basic understanding of how to produce a 3d solid modeling using Solid Edge. Example of this manufacturing information includes features such as protrusion, revolved protrusion, hole that can be created on modern computer. For each approach, we present a detailed description of the commands being employed along with some assessments of the technology. In this paper, we suggest the use of the Part module of Solid Edge 18 software for 3D solid modeling and the guided development of the execution drawing.

1. Introduction

The field of solid modeling has developed a variety of techniques for unambiguous representations of three-dimensional objects.

Solid Edge Part is used to construct individual part models. Part models are constructed by adding and removing material from a base feature. There are many advantages in using a 3D modeling package for producing engineering drawings.

This type of planning is very crucial for solids modeling. It allows you to develop the various steps of the generation process on paper before executing them on the computer. The planning process also allows you to optimize the steps.

The Solid Edge uses interface that contains a menu bar, toolbars, and ribbon bars that make it easy to access commands and set options. Examples of features used in paper are Protrusion, Revolved Protrusion, Hole, etc.

In this paper we will plan and execute several sweep operations to generate a solid model of a mechanical part.

2. The stages in producing the solid model

A solid model is created using 2D cross sections called profiles. We will decide which creation method to use before actually beginning the solid. These methods create what is called the Base Feature or raw material of the part. Each of these methods requires creating a 2D cross sections of the material, then the profile is projected, rotated, swept along a path, or lofted.

The purpose of this paper is to give a basic understanding of how to produce a solid model using Solid Edge, the module Part.

The final shape of solid modeling from this paper can be shown in figure 1.
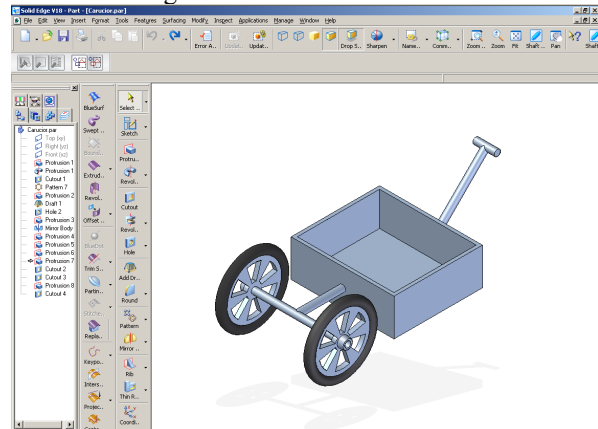

Fig. 1. The final solid modeling using Solid Edge, module Part

Afterwards, we will describe step by step of 3d solid modeling which was presented in figure 1. At the beginning, we will create a new Solid Edge Part file. We will describe the necessary operations for solid modelling in the following sections:

Section 1: Creating of Protrusion 1 command

The Protrusion command is located just below the Sketch command on the Features Menu. The Protrusion is used to take a 2D profile and create a solid model which takes the shape of the profile along vectors perpendicular to the plane of this profile.

In order to realize a protrusion propose, we have gone through the following necessary steps:
- Select the y-z reference plane for profile construction and click the Fit command;

- Draw a circle with its centre at the midpoint of one of the reference planes, then place and edit the dimension with a Smart Dimension;
- Click Return button and then we will set the distance.
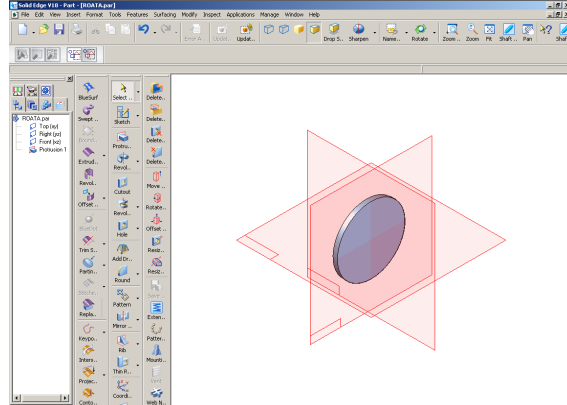In Figure 2, it can be seen the complete Protrusion 1.

Fig. 2. The Protrusion 1 command

Section 2: Creating Revolved Protrusion command
Thus, we will access the Revolved Protrusion command. The Revolved Protrusion command is used to create this model.
In order to realize a revolved protrusion propose, we have gone through the following necessary steps:
- Select the y-z reference plane for profile construction;
- Click the Fit button;
- Draw a circular profile by a considered diameter and define an axis of rotation that is illustrated in figure 3;
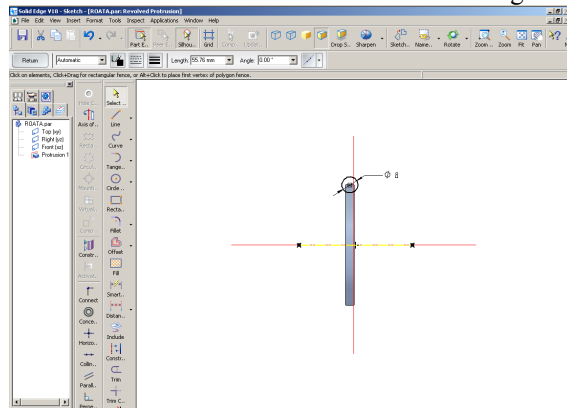

Fig. 3. Drawing the circle and defining the axis of rotation

-Click Return button.
Thus, we will select angle and direction of rotation that can be seen in figure 4.
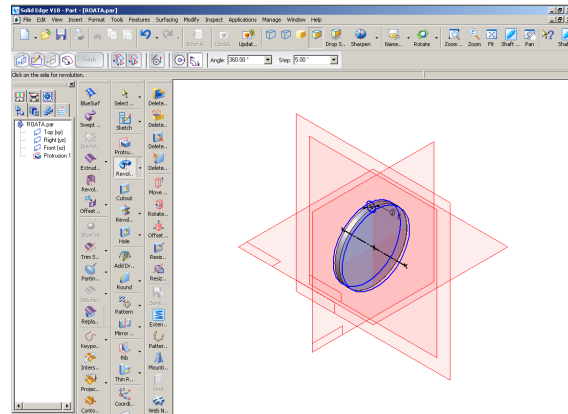

Fig. 4. The select angle and direction of rotation

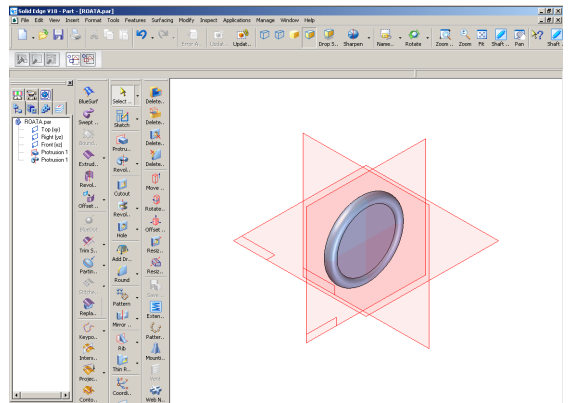- Figure 5 presents the final Revolved Protrusion command final.

Fig. 5. The Revolved Protrusion command

Section 3: Creating a Cutout command
The Cutout command is located two buttons below the Protrusion command on the Features Menu. The Cutout is used to take a 2D profile and remove material from a solid model that is the shape of the profile. The Cutout moves along vectors perpendicular to the plane of the profile. This command cannot be used to create the basic shape of the model. In order to realize a cutout propose, we have gone through the following necessary steps:
- Select the y-z reference plane for profile construction and then click the Fit button;
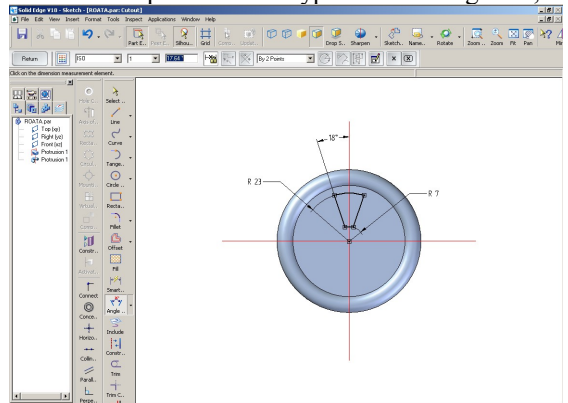- Create one profile of the type shown in figure 6;

Fig. 6. The profile realization for a Cutout command

- Click Return button and we will show direction and the respective setting of distance.
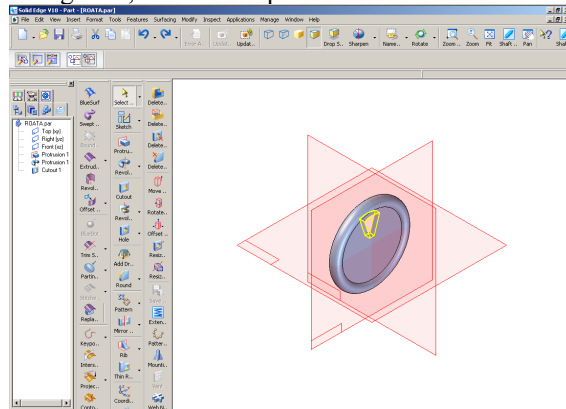
In Figure 8, see the complete Cutout.

Fig. 8. The Cutout command

Section 3: Creating a Pattern command

In order we click the Circular Pattern button on the Feature toolbar and we select the profile for multiplication.

Thus, we will click where we want the center of the pattern circle to be and we click where we want the pattern to start. In order to realize the proposed Pattern, we have gone through the following necessary steps:

- Select the y-z reference plane for profile construction;
- Draw profile, divide circle in 6 counts and setting radius value.
- Click the Fit button;
- Click Return button.

In figure 9, we can see the final Pattern command.

Thus, in this figure we show the yellow profiles that signify the multiplication of the profile that is drawn in figure 8.
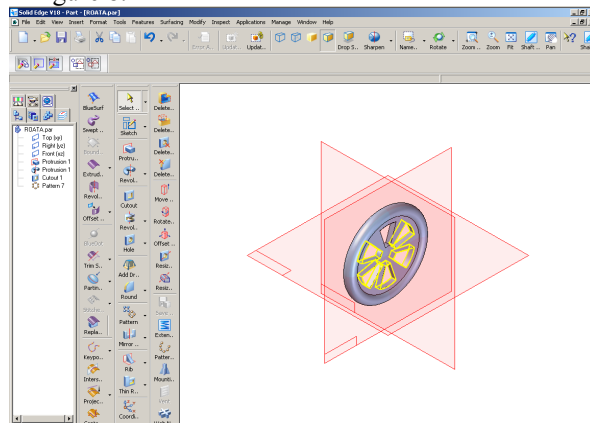

Fig. 9. The final Pattern command

Similarly with the first protrusion that it was drawn in figure 2, we will draw Protrusion 2 that can be seen in figure 10.
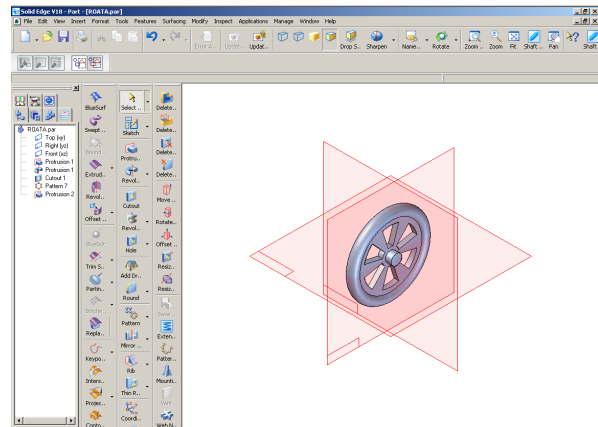

Fig.10. The Protrusion 2 command

Section 3: Creating an Add Draft command

In this section we will present the Add Draft command. For this draft, we use the From Plane option that is shown in the Draft Options window. We will select the reference plane that is located at the top of the part for the draft plane. When the surfaces are selected, we will set in 10 degrees for the draft angle in the Draft Angle field on the Ribbon Bar. Select the Next button on the Ribbon Bar to end the select face step of the command. In figure 11 is shown the blue line that signifies the direction of the draft.
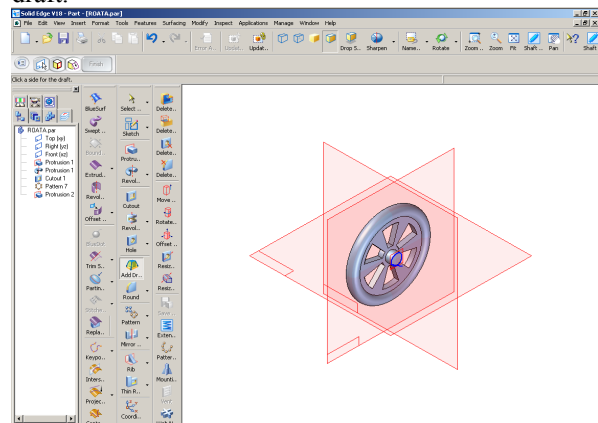

Fig. 11. The Angle Draft select

The completed draft and hole commands are shown in Figure 12. In order to make the drawing clearer, the planes will be turned invisible.
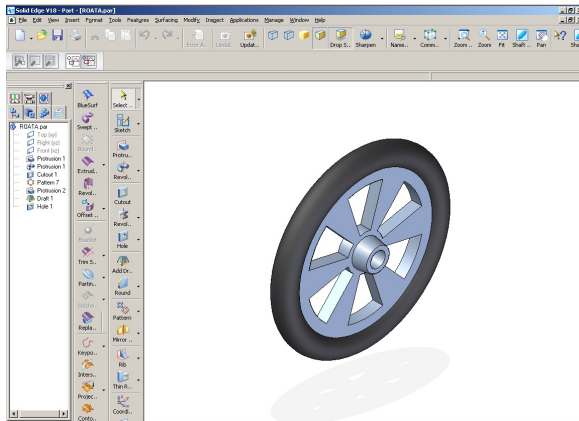
Fig. 12. The Draft and Hole commands

In figure 13 the Mirror and Protrusion commands can be visualized. They guide to the first part of the proposed model construction.
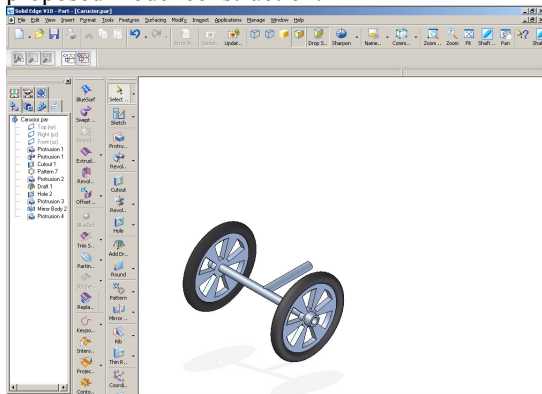

Fig.13. The Mirror and Protrusion 3,4 commands

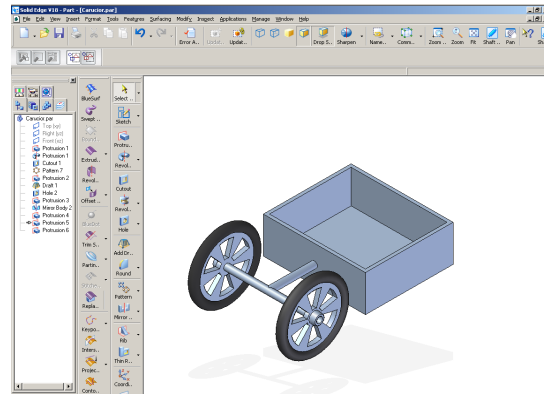In the second part is represented the main part of the truck that can be seen in figure 14.

Fig. 14. The Protrusion 5,6

The last part of the completed drawing is presented in figure 1.

3. Conclusion

This paper summarizes the most important new functions of the Solid Edge Version 18. It also shows their importance for prospective users. The basic commands from Part module are represented, too. They guide to the proposed drawing finalization.

4. Bibliography

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