Modeling of a drawing in three-dimensional space using CAD System

HARAGA GEORGETA, GHELAŞE DANIELA, DASCHIEVICI LUIZA
Descriptive Geometry and Engineering Graphics Department
POLITEHNICA University of Bucharest
Splaiul Independentei 313
Bucharest, 060042
Romania
g_haraga@yahoo.com

Abstract: - This paper refers to three-dimensional modeling of a drawing in Solid Edge software using a 2d sketch. For 3D modeling of a drawing we need a profile that represents a combination of a number of two-dimensional entities such as lines, arcs, circles, rectangles, and so on. In this article we will highlight the ease of using basic commands in creating a 3D drawing in Solid Edge Draft and Solid Edge Part environments. We insist on how to make threads in three-dimensional space and on automatic extraction of views or sections of the selected 3D model. Toward the end of the paper we point the importance of using Material Table command to change the material and its properties of the proposed 3D model.

Key-Words: - Solid Edge, modeling, protrusion, cutout, Solid Edge Material Table dialog box.

1 Introduction

Solid Edge is powerful software from CAD systems which has four environments: Part, Sheet Metal, Assembly, and Draft. Each of these environments creates a different type of Solid Edge document. For modeling the proposed drawing was selected the Solid Edge Part that is used to construct individual part models. The Solid Edge has an interface that contains a menu bar, toolbars, and ribbon bars that make it easy to access commands and set options. It is assumed that we are familiar with the basics of Solid Edge Part modeling understanding of how to produce a 3D model using Solid Edge [1, 3].

Not only can 2D drawings be produced easily from the 3D model, but having a 3D representation of the object or part of it provides more details. 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 us to develop the various steps of the generation process on paper before executing them on the computer [2, 6].

The planning process also allows us 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, Hole, etc.

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

2 Modeling of a drawing in 3D space

We create a new Solid Edge Part file. We will start through opening the Solid Edge Part and at the beginning we will create a profile, using Protrusion, Hole, Cutout, Round and Chamfer commands, as shown in figure 1.

Fig. 1 Using Protrusion, Hole, Cutout, Round and Chamfer commands in Solid Edge Part

The Protrusion command can permit to construct a protrusion by extruding a profile along a straight path.

The command named Cutout permits to construct a cutout by extruding a profile along a straight path.
Another similar command is the Hole command which is defined through constructing one or more holes.

Next, we will use the basic commands named Round and Chamfer, from the same Solid Edge Part environment. With the help of the Round command we used a constant rounding radius for the selected drawing. The Chamfer command constructs a chamfer between two faces along their common edge.

2 Working Type

In order to realize a thread we can use the Helical Cutout command.

At the beginning we draw a line of axis with the necessary value using Axis of Revolution command. Then we draw a profile in triangular form as shown in figure 2.a. Figure 2.b illustrates the finalizing of the Helical Cutout command.

Similarly, we will draw the two remaining 3D threads. The final modeling is shown in figure 3.

The Divide Part command allows us to split one part into multiple parts using in principal a reference plane. Each new part represents a base feature in its new document, and the new part documents are associative to the original part. On the Divide Part dialog box, in the Filename options, we will click the names for the new part documents.

In the Cutting Surface options, we can specify the geometry we want to use for a cutting surface. We can select a reference plane as shown in figure 4.

We will click a reference plane to define where we want to divide the part. Thus, we will click to define the cutting direction. For other additional division, we can click the Next Cut button on the ribbon bar and then define the cutting plane and the cutting direction. After we have finished making all the necessary divisions, we will click the Finish button on the Ribbon bar.

On the Divide Part dialog box, in the Filename column, we can type the names for the new part documents. Finally, we will click the Select All button, and then we will click the Save Selected Files button [7].

In figure 5 the effects of Divide Part command can be seen.

The Solid Edge software was designed especially for technical drawing generation and assures an exact representation, detailed describing, annotations and quotations which corresponds to the used standard.
A draft file from Solid Edge Draft environment consists of the 3D model projected to one or more 2D views of a part or assembly file.

In Solid Edge Draft we can automatically extract views or sections from a 3D model realized in Part module. We will begin by creating a section using the Drawing View Wizard command to view a 3-D part.

The Open File dialog box allows us to choose a 3-D part. The Drawing View Wizard Options dialog box allows us to set drawing view options for any type of model [4, 5].

The Drawing View Orientation dialog box enables us to either select a named view or create a custom orientation for the view. The Custom Orientation dialog box contains view manipulation commands that allow us to create a custom view as the primary view. The Drawing View Layout dialog box allows us to choose companion orthographic views and to place them with the primary view. In order to obtain the projections of the 3D model we will open a new page in Draft module.

On the Main toolbar, we will click the Fit command to display the entire drawing border.

For creating some primary drawing projections, we will click the Drawing View Creation wizard button, which captures the training information of the drawing and guides to projections location.

In the Drawing View Creation Wizard, we will set the following necessary options to capture information in order to achieve the finished product in two-dimensional space.

Figure 6 is realized in Draft module, by getting through detailed steps for the selected solid, but besides this the notion of automatic generation of a section is implied.

In the Solid Edge Part we can define the material and mechanical properties for a selected model with the Material Table command as shown in fig.7. In the dialog box from figure 7, we can select a material from the list, material and mechanical properties for the material such as face style, fill style, density, coefficient of thermal expansion, and so forth which automatically are assigned to the proposed model.

![Fig.7 The Solid Edge Material Table dialog box](image)

We can use the Solid Edge Material Table dialog box to do the followings: creating, editing, and deleting material property sets which are stored in the material library; assigning an existing material to the current document; creating a local material for use only in the current document.

The material and mechanical properties are used when we calculate the physical properties for a part or assembly, place the part in an assembly, render the assembly with Advanced Rendering, create a parts list on a drawing, define a bill of materials, and so forth.

When working with a sheet metal part, we can also use the material table to define the properties for the sheet metal, such as material thickness, bend radius, and so forth.

In the Material Table dialog box, in the Material Library Property option the material names and property sets are stored in an external material database file that is used to populate the property set for each material on the Solid Edge Material Table dialog box.

We can use these materials to define a material for any document on one’s own computer.

These options from Material Table dialog box make it easy for us, like users who work with a consistent set of materials and properties while providing the capability to customize the materials list [7].
For defining a local material we can create one local material name and property set for a document. This can be useful when we need a variation of a common material to be displayed in the Material column in a parts list, bill of materials, or in Property Manager. For the selected model we will do the following settings: we choose Steel for Material option and Material Properties for Settings option. After typing these properties, we click the Apply to Model button. All these can be seen easily in Fig.8.

![Fig.8 The Solid Edge Material Table dialog box, after properties choosing](image)

Figure 3 will become figure 9.

![Fig.9 The 3D model after applying selected properties using Material Table command](image)

3 Conclusion

This paper is addressed to all the CAD systems users, the authors’ main objective being to remove blockage of 3D and 2D space. In the final part of the paper, were illustrated aspects and properties of the selected model by using the Solid Edge Material Table dialog box.

Today the CAD software has changed the traditional working mode and thus at first we will begin by 3D modeling conception and afterwards by automatic extraction of the projections. There are many advantages in using a 3D modeling package for producing engineering drawings. The present paper makes evident the block elimination towards 3D, making the migration from the part module to the Draft one less difficult. The Solid Edge associated to Drafting automatically creates 3D models also bringing them up-to-date. As the models modify, the associated drawings are automatically brought up-to-date by also showing the model changes.

References:


