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ASSEMBLY MODELING USING CATIA SYSTEM

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ABSTRACT

CATIA is one of the most performance graphics programs that has been created for the production of technical drawings in the 2d and 3d with a high precision. In this paper, we have presented a simple assembly that is a kitchen table. Elected assembly is composed of several component parts that are modelling in the CATIA Sketcher and Part Design modules. To carry out the final assembly made of the component parts brought into the contact module we have used CATIA Assembly Design.

KEYWORDS: CATIA, Assembly, Pad, Pocket, Modeling.

1. Introduction

CATIA V5 (Computer Aided Three dimensional Interactive Applications), is one of the products of Dassault Systèmes company. At present, this is one of the most used CAD/CAM/CAE Systems, supplying a large variety of integrated solutions to satisfy all the aspects related to design and manufacture.

The CATIA soft divides into more modules the Sketcher, Part Design and Drafting ones being emphasized in a special way. CATIA Part Design is used to construct individual part models. Part models are constructed by adding and removing material from a base feature.

The present paper makes evident the block elimination towards 3D, making less difficult the migration from the part module to the Draft one. But its main objective is modeling of component parts and putting them in contact with using the CATIA Sketcher, Part Design and Assembly modules.

In this paper, the notion of assembly appears. An assembly is a document that stores several components which can be parts or other assemblies. The components used in an assembly can be pre-existing ones or components created within the assembly. Like a part, an assembly contains a specification tree. The tree shows the inserted components, and the constraints used to fix the components [1, 2].

2. Modeling of parts using CATIA Sketcher and Part Design modules

As a first step we will open the application: Mechanical Design → Sketcher Part Design. After opening this application, New Part dialog box appears in which we will click the option Enable hybrid design and then OK button will be activated as in figure 1.

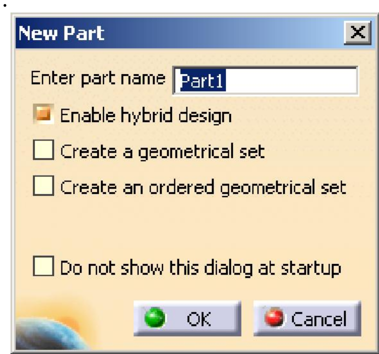


Fig.1 The New Part dialog box

We open the Sketcher and we will construct a rectangle in the xy plane as shown in figure 2.

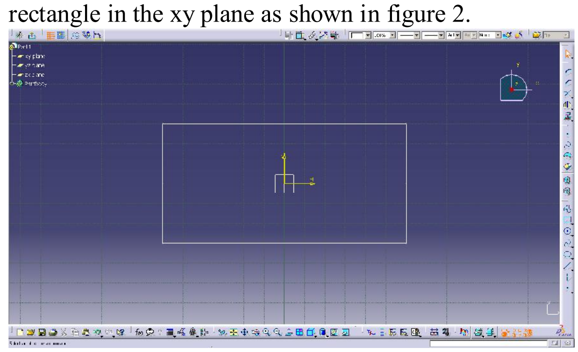
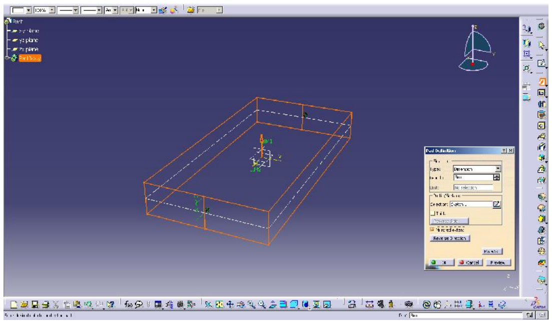


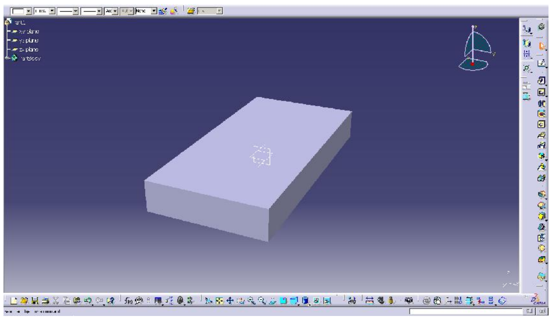
Fig.2 Construction of a rectangle using the Sketcher application

With Pad command the rectangular profile will be extruded as in figure 3 a, b. The move from CATIA

Part Design in a CATIA Sketcher module will be the Exit Workbench from Workbench toolbar.



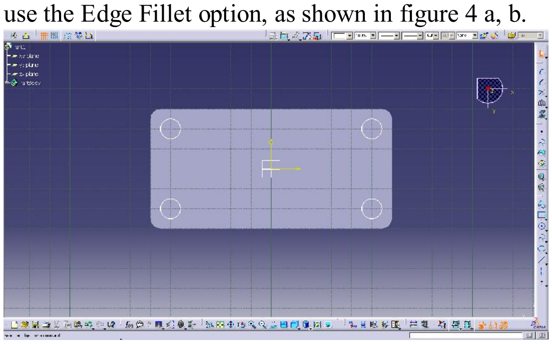
a.



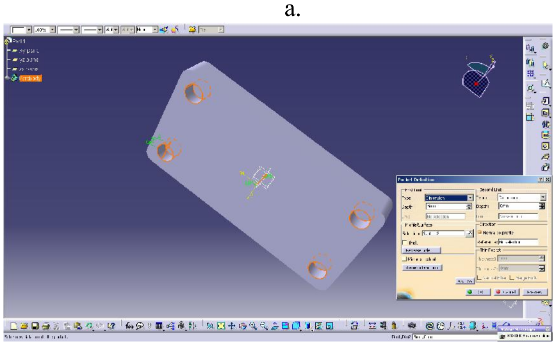
b.

Fig.3 Extruding a rectangle using the Pad application

For the execution of the four holes on the underside of the table we will use the Rotate command. Next, we make the four holes using the Pocket command and for the four connections we will use the Edge Fillet option, as shown in figure 4 a, b.



a.



b.

Fig.4 The Pocket and Edge Fillet applications

The first component part of elected assembly is concluded in accordance with its representation in figure 4.

Then, we will represent the following component parts. At the beginning, we make the four circles in the xy plane in CATIA Sketcher module.

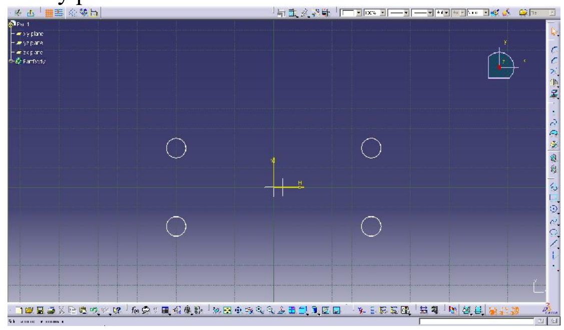


Fig.5 Representation of the four circles using CATIA Sketcher module

Similarly, by extruding the four circles, the figure 5 will become similar to figure 6. In this last figure, the four legs of the kitchen table are represented.

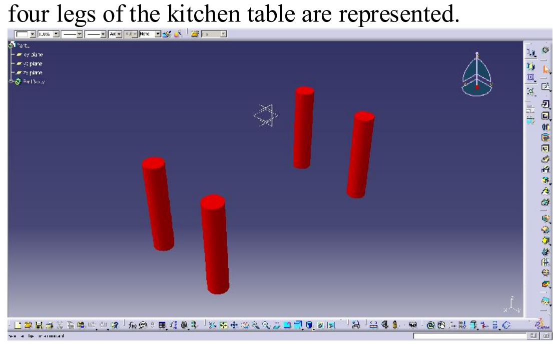


Fig.6 Extruding of the four circles using the PAD application

In order to put into contact the parts of the elected assembly we will use the CATIA Assembly Design module that will open the application: Mechanical Design → Assembly Design. Inthe beginning we will click on the Product 1 name in the specification tree. Then, we can access the Properties dialog box, by clicking this option. In the Part Number section we fill the field by writing the name Kitchen Table, as shown in figure 7.

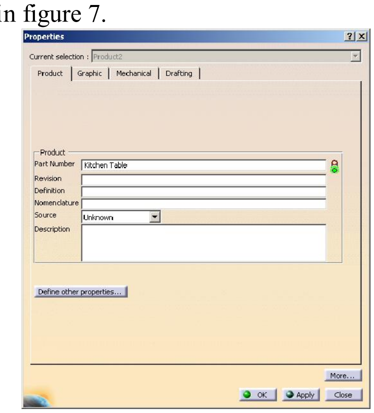


Fig. 7 The Properties dialog box

During the next stage, we will access OK button and the interface of the tree can appear as in figure 8.

Fig. 8 The interface after OK button accessing

Next, we click right on the Kitchen Table application using the mouse and then we select Components → Existing component…

Following this command selection, it appears the File Selection dialog box. Here, we can insert the two existing parts and then we press the Open button, as in figure 9.

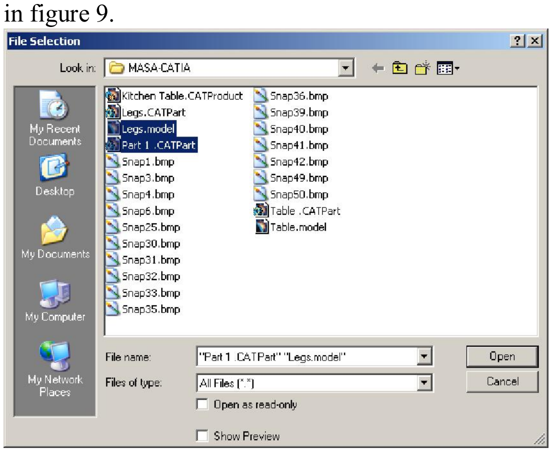


Fig. 9 The File Selection dialog box

Automatically, the component parts are put into in contact, having as a result the final assembly like in figure 10.

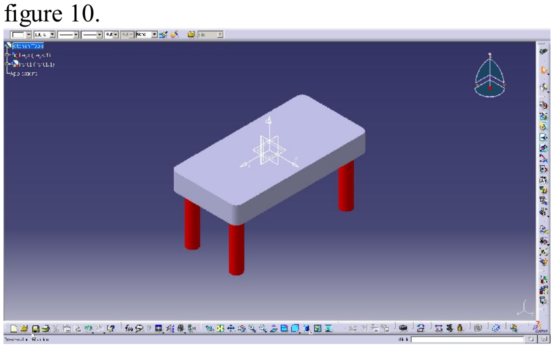


Fig.10 Final assembly made in CATIA Assembly Design module

The final assembly is a simple kitchen table which can have different colors depending on preference of each user. This can be done using the Graphic Properties bar as shown in figure 11.

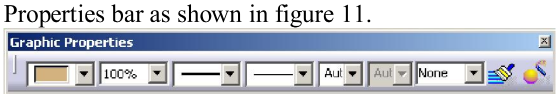


Fig.11 The Graphic Properties bar

For a more beautiful design, we can have all planes hidden using Hide/Show option.

After the application of the new color properties the kitchen table design will become an image according to figure 12.

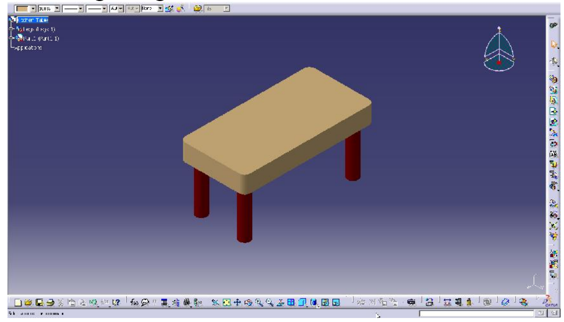


Fig.12 The Kitchen table after the application of the new properties

The CATIA software contains commands that work together in managing part colors and styles. By applying texture or color on the entire surface of the model or only on the desired surfaces the final product is obtained. For a high quality of the image in terms of color, the 3D models can be assigned a lot of distinct colors that are found by using Properties command.

If we select, by example, the superior part of a table and we click the Properties option and automatically, the Properties dialog box opens, as shown in figure 13.

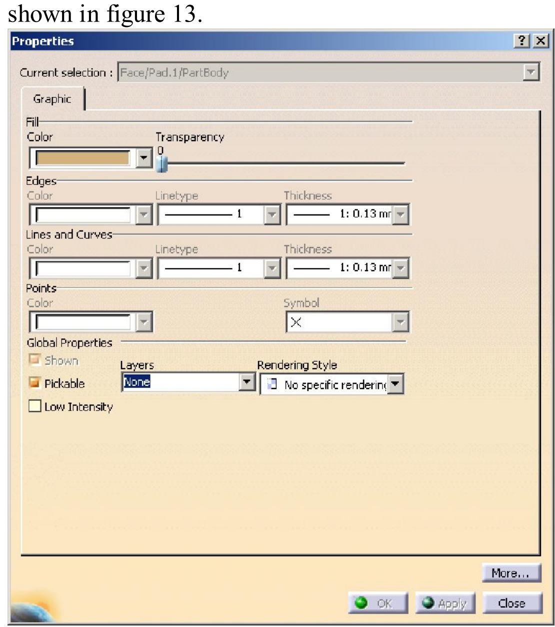


Fig.13 The Properties dialog box

For a better visualization we will create more views using the Create Multi-view button and drawing can become, as shown in figure 13.

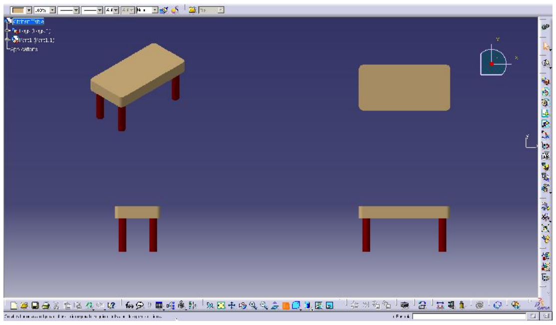


Fig.13 The Create Multi-view application

4. Conclusions

The CATIA is also a power graphics program that helps users to create simple or complex shapes in bi-dimensional or three-dimensional space. If we work with a solid component, with a model of the assembly or with a blank sheet of drawing, CATIA Drafting and its detailing instruments help us to make the drawings faster and more easily than with any other CAD system [3,4].

The present paper makes evident the block elimination towards 3D, making less difficult the migration from the part module to the Draft one. But its main objective is modeling of component parts and putting them in contact with using the CATIA Sketcher, Part Design and Assembly modules.

All the assembly applications of CATIA system are based on images of pieces or subassemblies. The important characteristics of assembly applications in CATIA system are:

- each assembly is defined by each component spatial orientation and by the connections created between subassemblies and the component parts;
- more parts and subassemblies can be opened and edited at the same time;
- the component geometry of an assembly can be created and edited in the assembly context;
- the geometry change of a piece of assembly is reflected in all the assemblies or subassemblies which contain this piece image;
- the assembly is automatically brought update in order to reflect the latest version of references pieces [5].

5. References

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