ANNALS OF THE ORADEA UNIVERSITY
Fascicle of Management and Technological Engineering
ISSUE #1, MAY 2013, http://www.imtuoradea.ro/auo.fmte/

# CAD / CAM MODELING AND SIMULATION, WITH THE CATIA

**Lavinia Ioana CULDA[1], Elena Stela MUNCUT[2], Doina MORTOIU[3] , Gheorghe SIMA[4]**
[1] University "Aurel Vlaicu" of Arad, lavyy_99@yahoo.com
[2] University "Aurel Vlaicu" of Arad, muncutstela@yahoo.com
[3] University "Aurel Vlaicu" of Arad, doina.mortoiu@gmail.com.ro
[4] University "Aurel Vlaicu" of Arad, gheorghe.sima@klastorf.ro

*Abstract-* The CAD / CAM integrated systems experienced a significant development in recent years, creating a close connection between the two adjacent areas: constructive design side - CAD (Computer Aided Design) and technological design side - CAM (Computer Aided Manufacturing). To make a virtual prototype, the CAD module generates models of the processed objects and generates processing and manufacturing related information required for the CAM unit.

*Keywords-* CAD/CAM, prototype, landmark, CATIA

## I. INTRODUCTION

The numeric control concept appeared in 1951 at the Massachusetts Institute of Technology for process engineering applications. [1]

The numbers entered in a specific sequence describes the trajectory of workpiece, the paths of cutting tools, the speeds of working motions, the numbering of tools, but also some related technical specifications. [1]

The obtained model using the CAD software will be geometrically correct, throughout its creation the user must take into account that the model must be conformable with CAM processing on machine tools. [1]

Computer Aided Manufacturing developed in response to the problem of materialization of a geometric pattern existing on the designer's board. The first step in computer aided manufacturing is transposing into the computer of the desired geometric model using a specialized application. There are a number of apps on the market that can help to achieve this step, the optimal solution depends on several factors. [2]

But for the correct choice of an application for modeling mechanical components must be considered also other uses of the developed geometric model: generating work drawings, making assemblies and subassemblies, verification or optimization of shape using a finite element computing program. [2]

Analyzing CAD solutions proposed by different companies (AutoCAD Inventor, SolidWorks, SolidEdge, CATIA, ProENGINEER, etc.), we can see that all are based on 3D geometric parameterized models adding or allowing you to add applications which permit running the realized model. Applications which use the geometric model can be independent or integrated on basic application. [2]

## II. THE COMPLETION OF BUSHING GUIDE IN CATIA V5 ENVIRONMENT

CATIA V5R20 is presented as a set of working environments (Workbenches) that can be activated in sequence to achieve a set of tasks. In this way we could avoid working with a single interface too complex for most user tasks. [2]

Computer aided programming of CNC machine tools development program aims the developing of piece - program starting from the plan of operations to be executed on sample, using electronic computer system for this conversion. [3]

The development of computer-aided programming has resulted in several directions in dozens of programming languages implemented on different computer systems.[3]

Essentially they consist in a set of geometric definitions that may cover more or less satisfactory the flat geometry (2D) and partially, the space geometry (3D) and a second set of moving orders with which generates movements of MUCN mobile subassemblies. This part is the actual processor which generates a dataset in certain intermediate form. [3]

For these data to be used to generate programs, certain specialized programs called postprocessor or simulators are needed, they therefore takes information from intermediate forms generated by the processor itself and process them generating phrases according to syntactic particularities of the numerical control equipment. [3]

The knowledge can be found in the tree structure generated in the design of a particular model or may have general application, being specified in the form of algorithms, expert systems, etc. [3]

The work plan, which will make the profile, is selected with the left tree structure of the document. As a result, the plan selected will be highlighted with a different color

display. [3]

Any construction of the 3D model (Fig. 1.) usually starts by making an outline plan which through a movement (translation or rotation) became a basic form.[2]

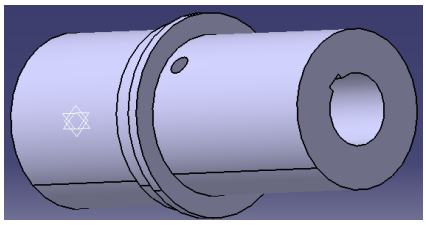

Fig. 1.

The first outline (Fig. 2.) will be drawn on one of the existing planes at the start of application. Thereafter, whenever a form is made based on an outline, it will be built on one of the original coordinate planes on a flat face of the piece or on a helping plan built for this purpose. [2]

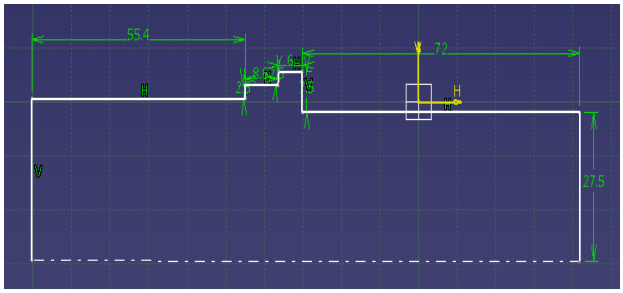


Fig. 2.

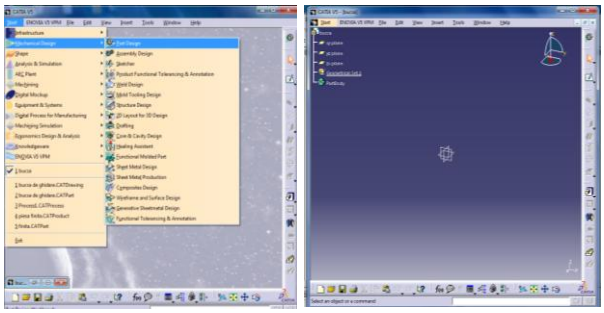

Fig. 3.

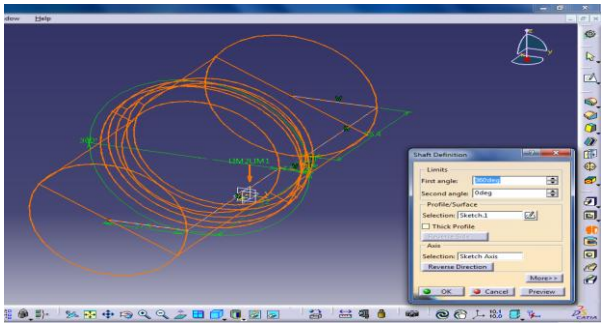

Fig. 4.

The outline plan should be drawn [2]:

- closed;
- possible internal contours must be closed;
- must be properly and completely defined (through quotas and constraints).

Interior contours can be defined along with the casing contours if the operation which generates solid (extrusion or revolution) is performed between the same limits. [2]

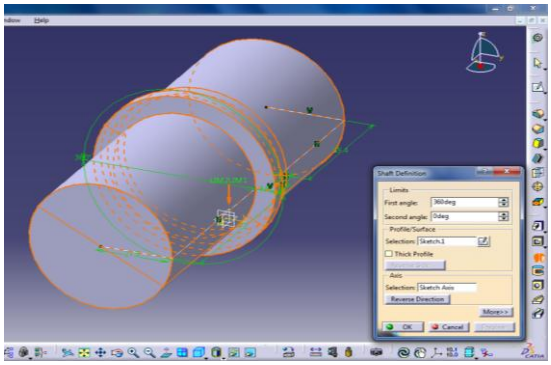

Fig. 5.

Part Design (Fig. 3.) is an environment for modeling parts. The modeler integrated into CATIA is parameterized, so it provides [2]:

- keeping all dimensions as values of parameters;
- automatic highlighting the state of outlines (under defined fully defined or over defined);
- automatic recurrence in case of changing the dimensions;
- notification of the intention of designer (automatic adding constraints);
- possibility of imposing relationships between parameters.

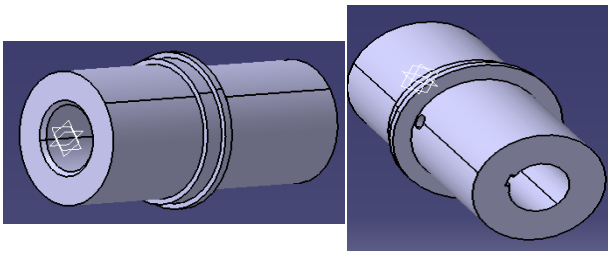

Fig .6.

To realize the bushing guide, first the contour is made in **Part Design** where the **SHAFT** order is applied to achieve an equidistant piece around the axis of symmetry (Fig. 4., Fig. 5.).

The necessary holes and keyway extensions are performed continuing to Part 2 of the processing, namely: the manufacturing of the bushing guide simulation on tool machine. (Fig. 6.)

## III. THE ACCOMPLISHMENT OF BUSHING GUIDE SIMULATION ON TOOL MACHINE IN CATIA V5 ENVIRONMENT

The **Lathe Machining** application easily defines NC programs dedicated to process cylindrical revolution workpieces using turning and drilling operations on 2 axis lathes, both on normal and vertical lathes. [3]

The tools can easily be created and integrated into catalogs. The trajectories described by the tools can be defined using the GUI dialog boxes, being possible to generate, simulate and analyze them. On this basis, finally the NC program dedicated to lathe is obtained, using an integrated postprocessor and technological documentation for the operator, in html format. [3]

We will consider the bushing guide, whose 3D model was made in **Part Design** application.

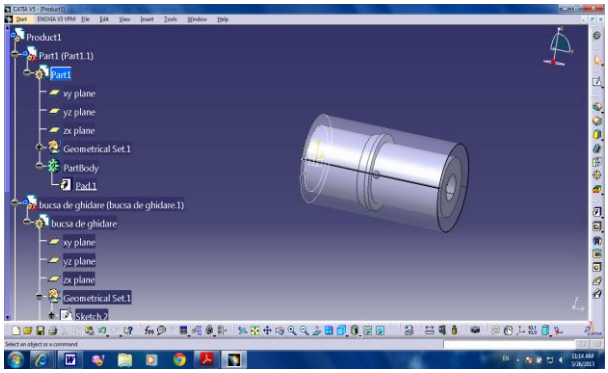


Fig. 7.

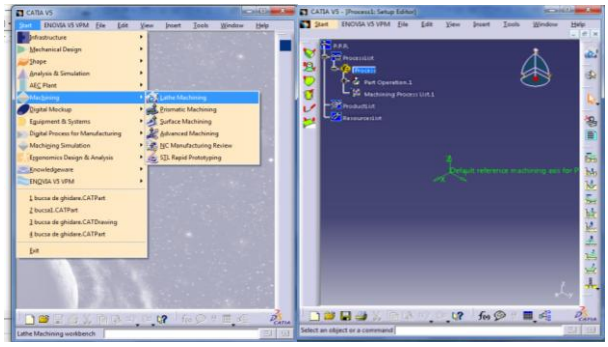

Fig. 8.

In **Mechanical Design - Assembly Design** application the representation of unfinished piece is realized which looks in Fig. 7. like a "coat" covering the piece.

From the **Star**t menu, the bushing guide's CATPart file is opened, followed by the Lathe **Machining** application from the **NC Manufacturing** module (Fig. 8.) next double-clicking on **Part Operation.1.** From the tree structure, the **Part Operation** window will open to define the main data necessary for simulation. (Fig. 9.)

Pushing the **Machine** icon the **Machine Editor** dialog box will be displayed, which allows you to select the processing machine (**Horizontal Lathe Machine** icon), the machine's axes and the piece which will be processed are defined.

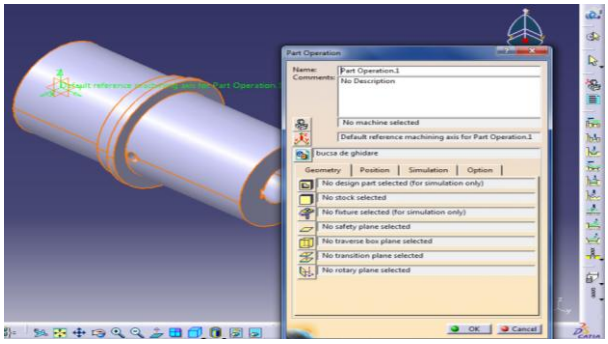


Fig. 9.

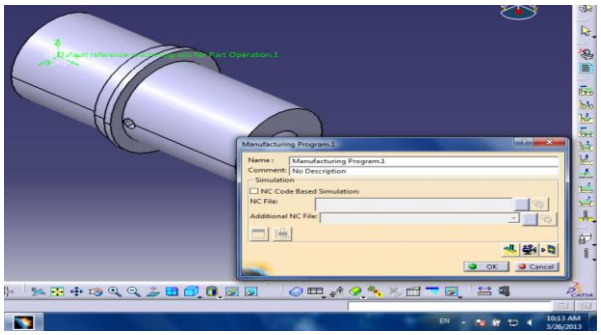


Fig. 10.

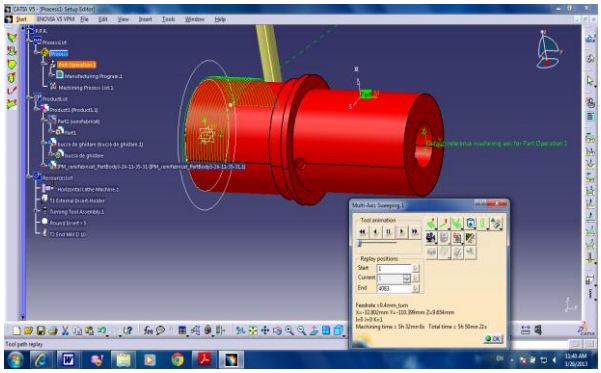

Fig. 11.

From the tree structure the **Manufacturing Program.1** (Fig. 10.) is selected, there will be the commands inserted for different successive processing operations to achieve the workpiece.

The program offers tools and operations, requiring only the specification of machined surfaces and some tool's parameters. Each programmed tool can be simulated, so that we have control over the final processing of the workpiece. (Fig. 11., Fig. 12., Fig. 13., Fig. 14.)

Final piece, after all settings can be simulated (Fig. 15.) and may be already generated part program. (Fig. 16.)

Integrated CAD / CAM systems has developed

strongly in recent years, having an important development not only for the CAD but for the CAM module too, enforcing an ingenious and useful mixture of the two modules.

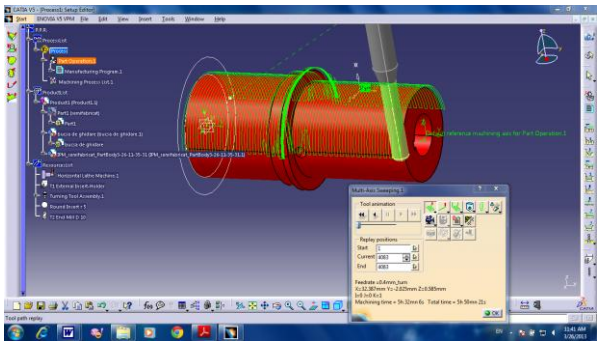

Fig. 12.

The development of design environments by introducing CAM type applications was particularly beneficial for economic environment which prefers virtual prototype instead of the expensive physical ones.

To achieve a virtual prototype the CAD, CATIA module, generates graphical models of parts and manufacturing related information required for the CAM module.

The CATIA (Computer Aided Three-dimensional Interactive Applications) Version5 is made by the IBM and Dassault Systèmes and is currently worldwide leader on the CAD / CAM / CAE software market.

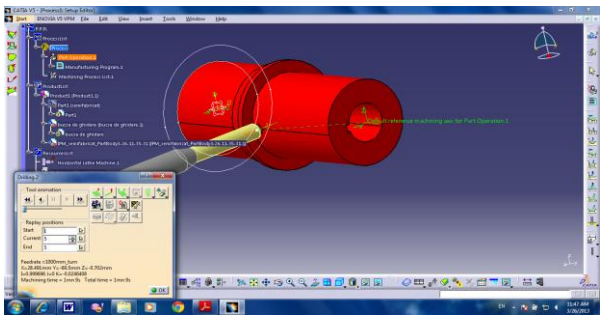

Fig. 13.

CATIA started to conquer the industrial environment founding its application in diverse areas, from engineering industry, aerospace and automotive industries, and ending with the design and construction of parts for the factories producing interior and exterior doors.

## IV. CONCLUSION

Finally, this environment is effective as teaching tool, covering a wide range of disciplines that can use the environment to illustrate the concepts presented in the class.

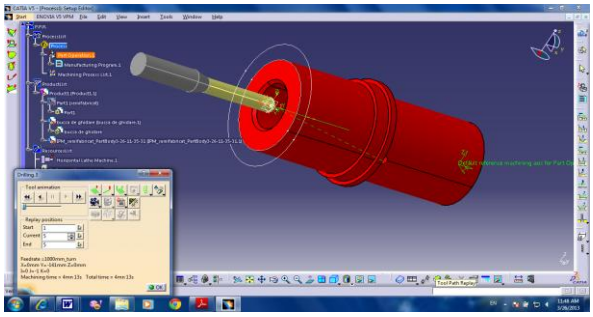

Fig. 14.

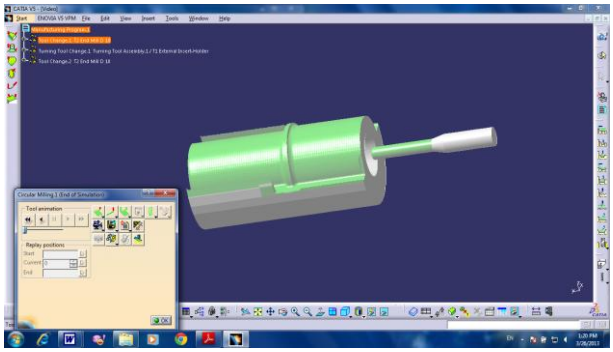

Fig. 15.

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$$ CATIA APT VERSION 1.0¶
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$$ Manufacturing Program.1¶
$$ Part Operation.1¶
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$$ Manufacturing Program.1¶
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GOTO  /  -34.87516,    0.02500,    0.82381¶
GOTO  /  -34.62598,    0.02500,    2.87824¶
GOTO  /  -34.17609,    0.02500,    5.21466¶
GOTO  /  -33.56847,    0.02500,    7.46408¶
GOTO  /  -32.80226,    0.02500,    9.65440¶
GOTO  /  -31.88190,    0.02500,   11.77810¶
Etc.¶
$$  End of generation of : Circular Milling.1¶
FINI¶
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Fig. 16.

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