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Implementation of CAD/CAM system CATIA V5 in Simulation of CNC Machining Process

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Abstract

This paper presents improved methodology of the virtual simulation of turning and milling technological process of the selected components in the graphical interface of CAD/CAM system CATIA V5. Each technology of machining must be supported by selection of the relevant geometry of cutting tools and cutting materials. The technological system CNC machine tool – progressive cutting tool – workpiece – fixture must ensure sufficient stiffness, which has a direct influence on the quality of the surface finish of the final product. This technological process is important to enter the process of manufacturing of various precision engineering components of complex shape designed for the needs of automotive and aerospace industries. Using the highest level of programming through graphical engineering system is designed for creating the ISO programs using CNC machine tools. All simulations of cutting experiments show that the proposed clear-up tool path works well in the real cutting process and can improve the machining efficiency of the machining process. At the same time, the manufacturing must have the advanced process equipment and method due to the requirements of the design performance and machining efficiency. Then, the total cutting volumes of an impeller are conventionally machined by (CNC) machine that is set up.

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Keywords: CAD/CAM system; turning; milling; machining strategy; postprocessor; virtual simulation

1. Implementation of CAD/CAM system CATIA V5

Great importance of all new technologies, materials, machinery, progressive methods and information tools that enable more efficient use of starting materials, produces a more cost effective, fast enough to produce reliable and quality products and above all succeed in the market in an increasingly challenging competitive environment [8], [10].

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Nomenclature

a_p	depth of cut [mm]
f	feed motion [mm.min^{-1}]
n	spindle speed [min^{-1}]
CAD	Computer Aided Design
CAM	Computer Aided Manufacturing
CATIA	Computer Aided Three dimensional Interactive Application
CNC	Computer Numerical Control
HSC	High Speed Cutting

The development of innovative products and their realisation by means of advanced manufacturing methods and process combinations is a key issue in international competitiveness. In the field of the new Rapid Technologies, represented today mainly through various Additive Manufacturing processes, the progress of the High-Speed Cutting (HSC) technology is of remarkable and specific importance [3]. Today's trend component is machining in one setup, with minimal clamping. This is achieved by the time reduction of the final machining, while eliminating inaccuracies caused by manual switching of the part [10]. The introduction of computer graphics allows a computer to create pieces, handle and examine them. Universal application of computer aided systems brings significant benefits. CAM is most closely associated with functions in manufacturing engineering, such as process planning and numerical control (CNC) part programming. [12]. Programming of CNC machines is carried out at three levels:

- DIN ISO programming, which is manually inserted G and M codes. The advantage is accurate programming of each movement. The disadvantage is the error of the human factor [13].
- Higher workshop programming. Application with these systems required minimum knowledge of DIN ISO programming. All information about the workpiece are graphically entered which makes operating the machine. The advantage is the possibility of simulation [10].
- CAD/CAM systems, which are a complex preparation of production process. CAD is computer-supported design for preparation production. Capable drawings of a 3dimensional model of the product, creating technical documentation, conversion from 2D to 3D display. CAM is computer aided manufacturing. It is a kind of automation through computerization. CAM brings the possibility of simulation machine cycles. Integration of CAD/CAM offers the possibility to generate CNC codes of design components [7, 9].

Machining of various components of complex shape using milling, drilling holes and so on, is possible to implement more gripping and carrying out each operation machining on various machine tools. By automating the production process of such components can be performed all the operations to a minimum number of clamping in more axial CNC machining centres. CAD Support/CAM system makes the entire production process faster, and what is particularly important in those parts of complex shape of scale [2, 4]. Moreover CAM systems provide access to a large variety of data such as the programmed feedrate, the tool length, the number of flutes, etc. These parameters can be of interest for the evaluation for which the prototypical realisation of the assistance approach allows collecting this data without the need for user interaction [1].

2. Simulation of CNC Machining Process

Individual acts necessary for the design of machining on CNC machining centres in the CAD/CAM system CATIA V5 Lathe and Prismatic Machining modules are as follows:

- Determining appropriate and Geometric BODY SET for parts and semi-on working tree. Create a 3D model.
- Creating a blank (rough stock) with those allowances which will be model.
- Going to the machine mode of CAD/CAM system and set tool parameter, which will perform roughing model.
- Defining the strategy of machining operations, which are determined semi-alone model, the machining allowance, tolerances, cutting speed and feed motion of the machine tool path.
- Setting the proper parameters of the other cutting tools which will perform profiling.

- Milling and turning machine tools to be stored in the machining centre in preparation for the fabrication and inspection of the CAM [4]. Running simulations output control machining (can see on Fig. 1, 2).
- CNC program in the ISO format, as appropriate CNC machine tool and CNC control system [7].

After creating a technological process of production, the choice of an appropriate strategy roughing and finishing of the functional areas of the drilling and milling tools to select suitable material, geometry and cutting insert holder. Then determine the optimum cutting parameters for each roughing and finishing hard milling (feed, cutting speed). A model created with the tool, along with intermediate put into the environment module of CATIA V5 Manufacturing in the work of the tree PRODUCT LIST. Creation of the production process is at the top of the tree under the name PROCESS LIST. Process through the menu is listed in NC Manufacturing module design type CNC machine tool. Then we determine the work origin "W" of the workpiece coordinate system of the machine, the type of post-processors to generate CNC code. Through the function "Auxiliary Operations" is generated optimal tool shape and cutting material, tool holders, cutting inserts and determine the strategy (Fig.3) of whole process of production using Machining milling operations. Using the function "Tool Path Replay" simulates of the process of machining. The function "Generating NC code" then generates the CNC program in ISO format (G and M codes), using a memory card, transferred to a control system for CNC machine tool. The choice of a machining strategy [11] for machining tool path generation has two main objectives: avoid significant federate loss and reducing as possible machining time. To achieve these goals, the machining assistant often associates to a standard of 3-axes HSM two types of machining strategies widely used in CAM software for the machining of forging dies:

- parallel planes strategy: the machining feed direction is defined by parallel planes containing the tool axis,
- Z-level strategy: the machining feed direction is defined in parallel planes perpendicular to the tool axis.

Generally in all CAM software, tool paths computation is initially carried out through a selection of the machined surface and the identification or selection of avoidance shapes. When any avoidance shape is selected, tool paths are limited to the boundaries of the machined surface. In the case of a cavity linked to the machined surface (topological relation), tool paths can be extended in order to maintain the set point of the feed rate [14].

3. Milling Simulation in CATIA V5

The code in CAD/CAM system CATIA is implemented through function Generative CNC code. When activated this icon to formation of the CNC program form the active machining process, which was designed in the previous steps. Before the act of generating the CNC code it is necessary to determine the correct data on the method of machining, tool and defining its holder. The generated code can be further edited on a computer or directly to the CNC machining centre, the second option is not desirable in terms of production efficiency. A necessary early step is to delete the generated CNC code, because these data are set in the CNC machining centres and the abolition of numbering lines. It is important to change the file to CNC program without an extension, to make it readable for CNC machining centre. The last stage is to upload CNC code for portable media and inserted into a machining centre, which operate codes directly from portable media due to lack of internal memory machining centre.

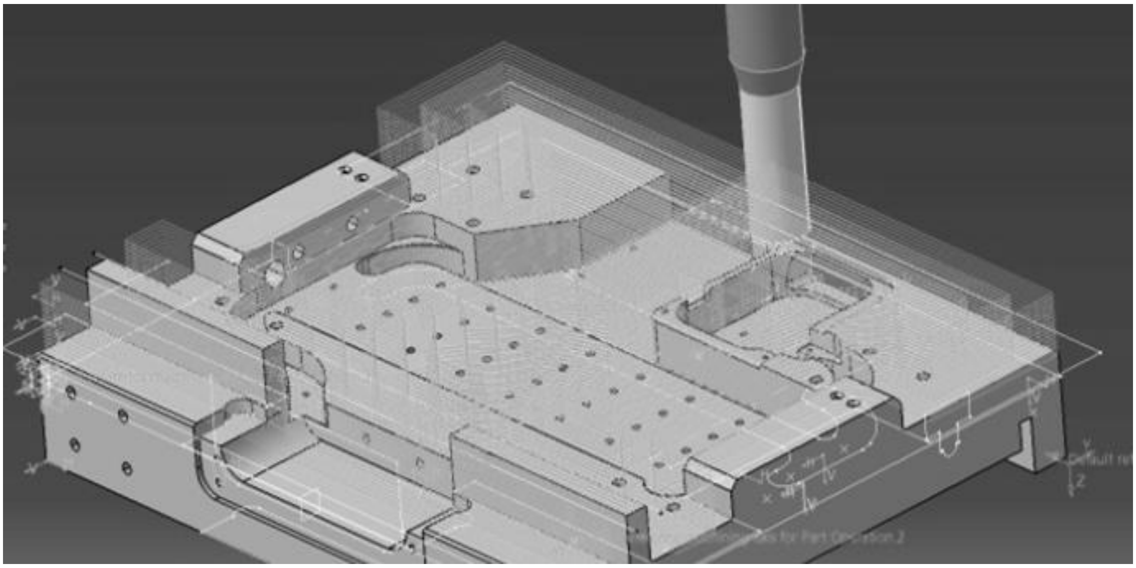

Fig. 1. prismatic milling process on selected CAD component [11].

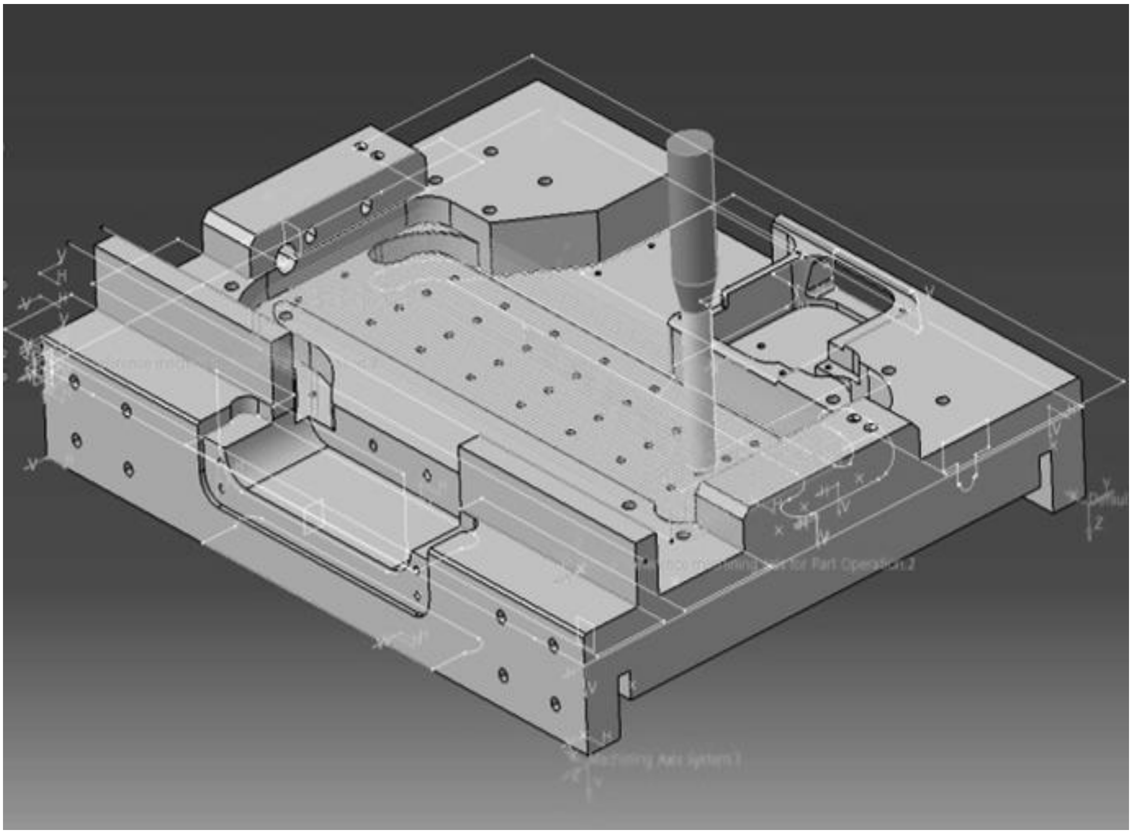

Fig. 2. virtual simulation of prismatic milling process in CATIA V5 [11].

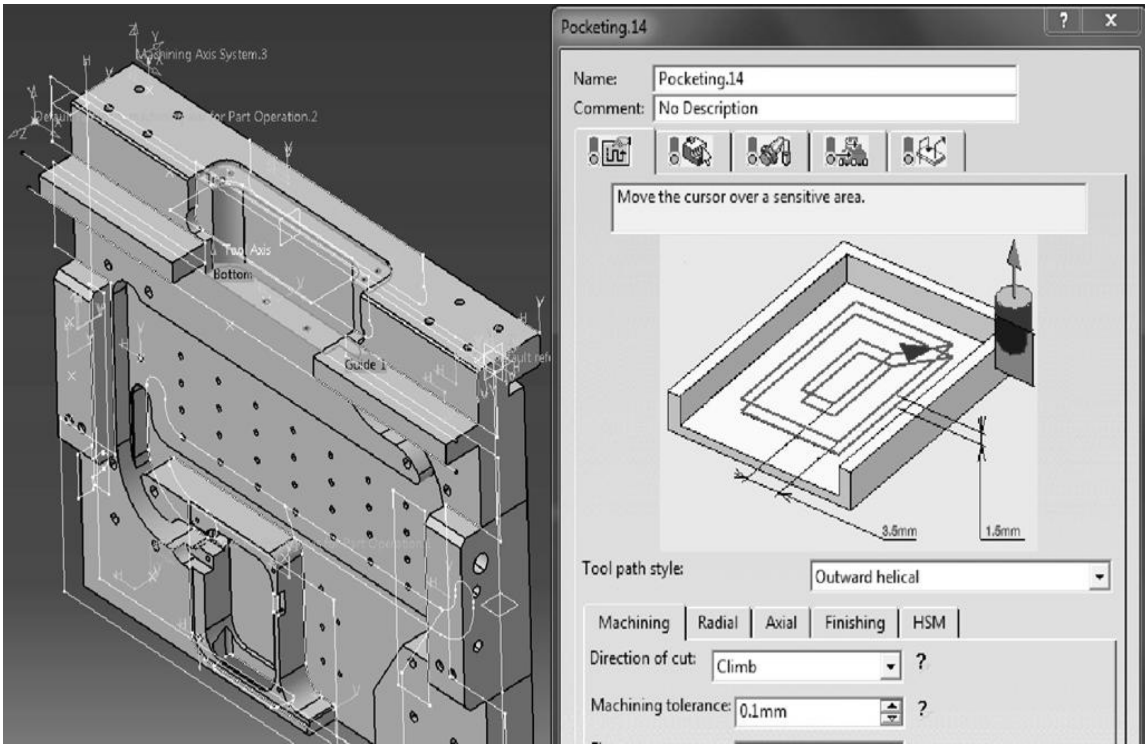

Fig. 3. strategy of prismatic milling process and dialog panel for the settings of machining simulation [11].

Prismatic milling model components were carried out on a vertical machining centre with the SH610 STORM system FANUC Series 0i-MC.

Machine parameters: Working feed f = 1 ÷ 10000 mm.min^{-1}
Workbench 700x450 mm, Table load max 300 kg
Max. spindle speed n = 10 000 min^{-1}, spindle taper SK40
Workspace: X-axis (610mm), Y-axis (460mm), Z-axis (480 mm)
Shifts: Rapid X/Y = 36 m.min^{-1} rapid traverse Z = 24 m.min^{-1}
Roughing: n = 1490 min^{-1}, f = 1700 mm.min^{-1}, a_p = 0.8 mm.
Finishing: n = 2900 min^{-1}, f = 2300 mm.min^{-1}, a_p = 0.15 mm.
Workpiece material: tool steel TOOLOX 44 with 45 HRC

After clamping the stock it important to set up the plane and other corrections that are needed to start machining. Milling of the model we use cylindrical roughing cutter with a diameter of 32 mm carbide insert, stem diameter of 32 mm radius of curvature of 2 mm. For profiling model has been implemented cylindrical cutter with a diameter of 16 mm, 16 mm shank diameter of 2 mm radius. Spherical milling cutter with a diameter of 6 mm was used for removing the additional material from the grooves.

Pointing milling is the most universal method of the conventional free-form surface manufacturing. The dramatic merit to designer and engineer is that almost all the complex surfaces can be point milled accurately. However, a disadvantage is that it is inherently time-consuming. Compared to the point milling, the efficiency of the flank milling is higher than that of the point milling. Then, it is the generally conceived that a free-form surface can be flank milled if it can be closely approximated by a ruled surface [15, 16]. Thus, the CAD/CAM system within which CNC programs are being created, should describe precisely part geometry, have flexibility for the task of relative position of the tool and workpiece and also be able to take into account features of cutting process [5].

4. Turning Simulation in CATIA V5

The CATIA, which is the product of Dassault Systémes Co., are used mostly in medium or large technical companies. Creation of model components using CATIA V5R15 is implemented using the main Start menu, where modules are located, such as SKETCHER, which is creating 2D sketches and shapes, serving as a basis for the subsequent creation of 3D models in other parts of the product CATIA V5R20 (Part Design, Generative Shape Design, Sheet Metal Design, Wireframe and Surface Design, Weld Design). Proper written methodology was created, which would be able to define the ways between single devices during running process [6].

In order to allow create a precise technical drawings, it is important to be able to draw 3D models of machine parts, which form the basis for further work in the CAD/CAM systems. The advantage of modelling and the creation of the underlying sketches lies in the simplification of control, i.e. using all the assembly features and icons are mostly single-level, creating a much better idea to work, unlike some other CAD systems. After creating a two dimensional sketch, it is possible to get into the 3D interface through the function "Exit Workbench" in modules PART DESIGN, SHAPE DESIGN, etc. In 3D interface, then we use the resulting functions which create PART or parts of the Product, which we can join in the module ASSEMBLY DESIGN. Advantage of CATIA is its simplicity. Icons are placed directly on the sides of the active desktop, or you can activate them via the main menu View-Toolbars. This structure of the whole system is necessary, in view of the scope and job opportunities CATIA. Moreover, CATIA can also do conversion with the documentation created in older CAD systems (e.g. AutoCAD DWG format) directly in special module INTERACTIVE DRAFTING).

The creating procedure is clearly stated in so-called working tree, which is always on the left side of the active desktop (see Fig. 5). Simulation of the machining process includes CATIA V5 Manufacturing module. The menu module is a 2-axis, 2.5-axis, 3-axis, but even 5 axis machining and more. Through the following options various rotating parts can swing aside, but also milled grooves and drilled holes in the rotating parts, which may be particularly applicable if the machine tool contains so-called management accessories C-axis. Milling of complex shapes can realize through main menu module offered by the NC 3-axis and also 5-axis machining. The CAM module is designed to connect a CAD model in CATIA with its own product tree. It can be selected either a blank from which the final piece will be made. 3D model oh part and blank is consequently inserted into working part of the called Process List. Then we realized the choice of the machine tool and control system, cutting tools and processing parameters. After defining the parameters needed CAM software generates data called CL (Cutter Location Data) that is a text entry in the ISO format that can be translated using the postprocessor.

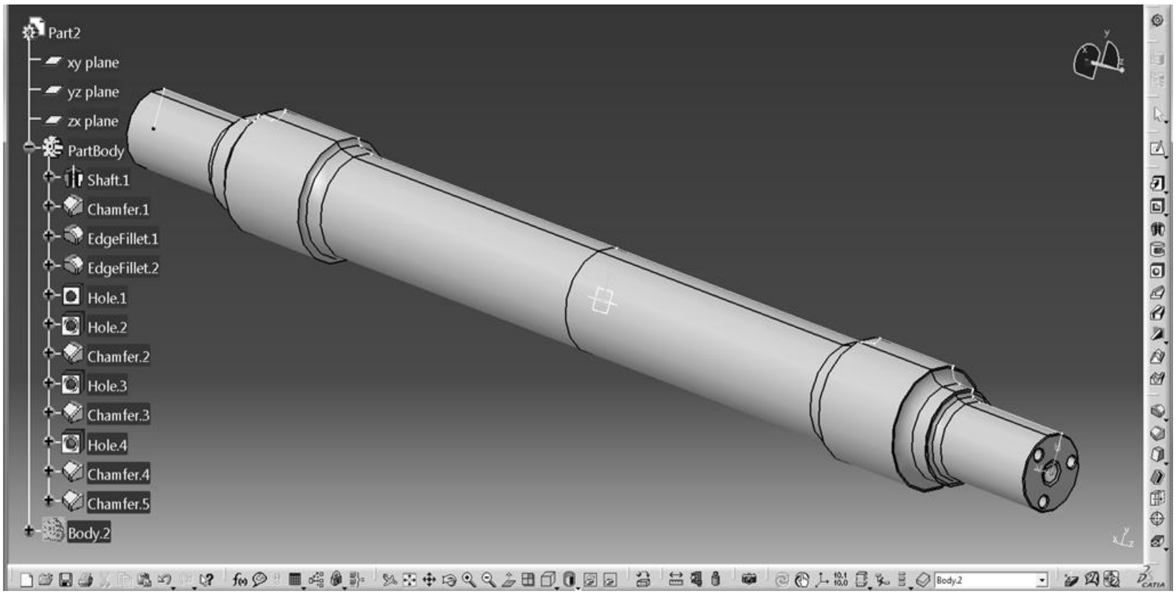

Fig. 4. process of model components in CAD/CAM system CATIA V5 Mechanical Design in the module using a tree as a work of history [6].

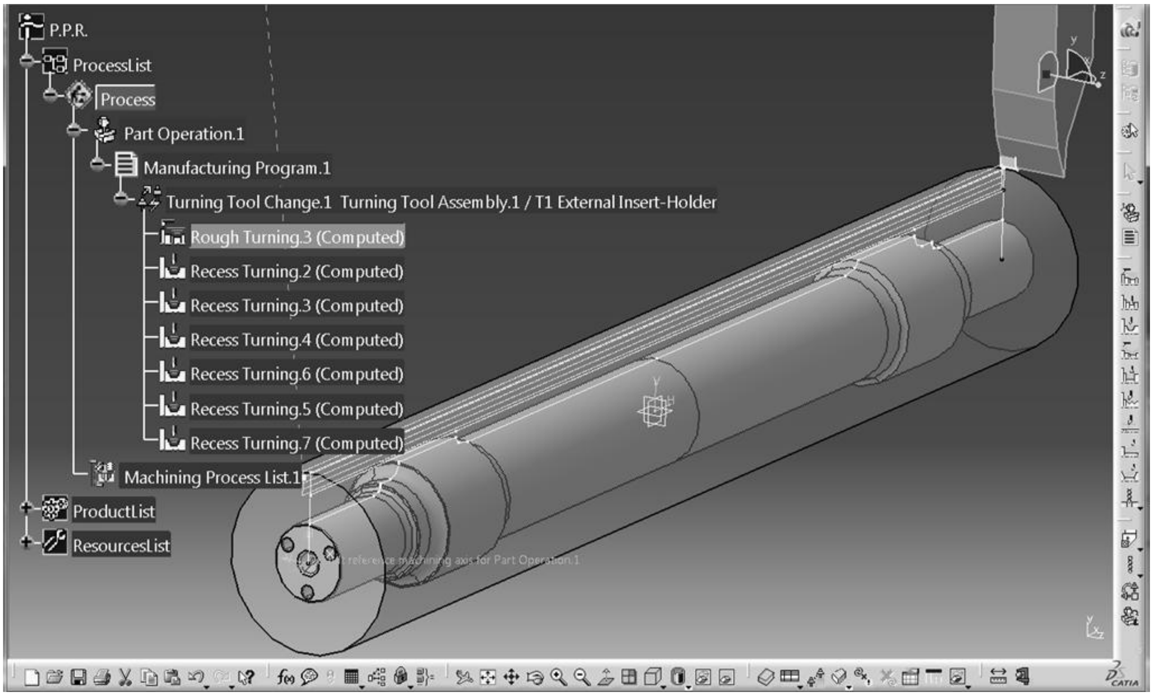

Fig. 5. the method of generating tool paths for the determination of external and internal borders [6].

5. Conclusion

The aim of this paper was realize practical examples of HSC machining technology and programming CNC machines with CAD/CAM system. CATIA proves the relevance and coherence of the new technologies, materials, machinery, progressive methods and information tools that enable more efficient use of starting materials to produces lower costs. One objective of the article was to suggest the possibility of creating CNC program production process components on a rotary chuck, or with minimal handling. This is achieved by shortening the time of the final machining while eliminating inaccuracies caused by manual switching of the workpiece. This paper presents a method of connecting High Speed machining technology HSC with the introduction of computer graphics contained in the CAD/CAM systems. CAM systems are used for preparing data and creating CNC programs for manufacturing components. The first step in the simulation of the manufacturing process is the selection of appropriate shape and semi-finished parts. The choice of components was associated with finding the material composition of parts and determining the class of its workability. Based on the findings of machinability suitable cutting tools and cutting conditions for machining were selected. The second step was to determine the production processes. The manufacturing process was designed to model assemblies constructed in CATIA. The manufacturing process was first designed and simulated using the CAM system. A significant reduction in production time contributed towards lower production costs. Machining process is based on empirical and theoretical relationship between cutting conditions. Manufacturing operations and instruments were chosen with respect to the geometry and the required precision assembly. The results presented in this paper can be further exploited in the process of teaching courses and programming CNC machines and technical practice for upgrading older solutions and processes of the main and additional times, while maintaining the dimensional accuracy of machine parts. In our future research plans e.g. comparing to the existing workshop CNC turn-mill machining centers, have the presented simulation methodology have a number of advantages. First, with the same interpolation steps, the contour accuracy is significantly higher. Second, the programming module is easier to use more simple G code lines. And last, machining cost is lower because of the reduced machining time.

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