12th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2008, Istanbul, Turkey, 26-30 August, 2008

CATIA V5 AS MULTITASKING MACHINE TOOLS PROGRAMMING ENVIRONMENT

Ph.D., Eng. Arkadiusz Kolka
Silesian Technical University
Machine Technology Department
ul. Konarskiego 18a, Gliwice
Poland

ABSTRACT

In The Industry and Universities CATIA V5 is very well known drafting and modeling software, but it has very strong CAM abilities also. Author of the paper using CAM system since a lot of years, and in his opinion CATIA CAM is very powerful product. Main intention of the author is to introduce present CATIA V5 CAM functionality from aspect of programming of Multitasking Turning-Milling Machine tools. As an example Okuma Multus B300 with OSP-P200L have been taken. Specialized postprocessor have been prepared and methodology of process planning was presented by the author. NC code prepared during works have been tested on the machine tool by simulation and machining.
Keywords: CAD, CAM, CNC, Multitasking Machine tools, Programming

1. INTRODUCTION

Multitasking Turning-Milling Machining Centers are very powerful machine tools. First this kind machine tools have been designed in 70ties of XX century. Main goal of this construction it was to make possible complex machining of workpieces by turning and milling operations. It was necessary to join potential of turning centers (with programmed X,Z,C axis) and milling centers (with programmed X,Y,Z,A,B axis). One of the first this kind machine tool was LMT70-AT (Fig.1.) made in Japan by OKUMA factory. This machine tool had moving support with eight tool turret and milling spindle. For milling tools it was possible to use ten pots tool store. Range of the Y axis it was +/-150mm. It was possible to machining by use X,Y,Z,B and C axis. During last 30 years constructions of this kind of machine tools have been evolved. One of the newest one on the market is MULTUS (Fig.2.) multitasking machine tool offered by OKUMA.

In many cases customers decides to buy less complicated lathes and milling machines to do complex workpieces. Cardinal reasons of those decision are big price of MTS machine tools and complication of programming. The second one can be solved by use specialized CAM software or use

Figure 1. LMT70-AT

Figure 2. MULTUS B300

multipurpose CAM software with specialized postprocessors. Problem with use well known CATIA V5 Manufacturing and specialized postprocessor application are presented in the paper.

2. CHARACTERISTIC OF TURNING-MILING MACHINE TOOL MULTUS

Main elements of MULTUS construction are presented in the fig.3. On the body (1) are fixed headstock (2) with electrospindle module, and horizontal pair of slideways. On the slideways are moving cross slide (3) with support column (4). Side slideways are used for cross moving by complete support column. Column are moving as Ys axis. Support is moving (X axis) on the slant slideways fixed to the column. By compose of Ys and X axis is realized Y axis. Additionally center works are done by use of tailstock or subspindle.

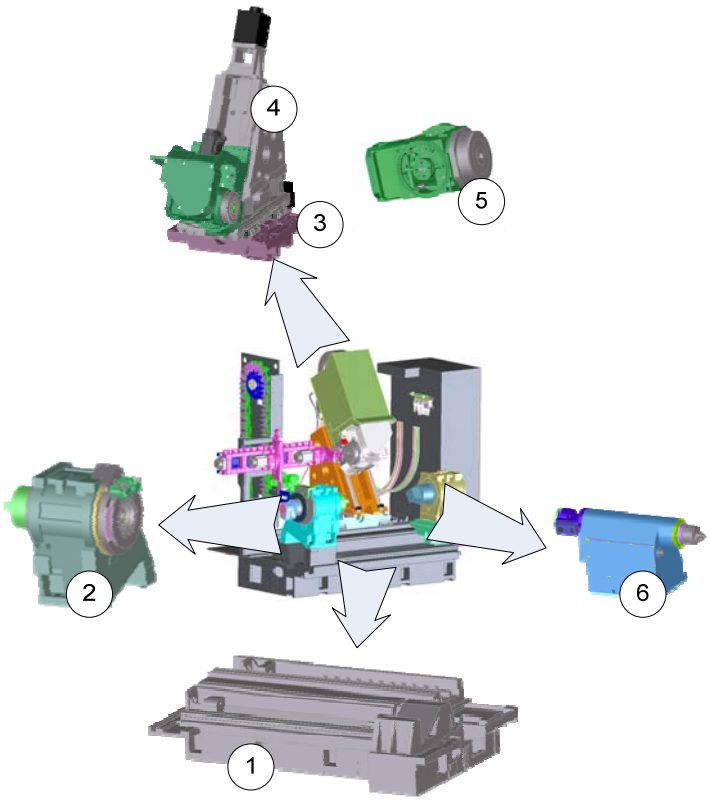


Figure 3. Main elements of the Multus Machine tool

Headstock is typical for horizontal lathes and is equipped with electrospindle. Different then lathes is construction of turret. Typical turret have been replaced by rotating milling tool headstock. This turret is rotating around Y axis by using BA axis. BA axis indexing by 0.001 deg. To the turret are mounted milling tools with diameter up to 125mm and turning tools. M-tools are driven by electrospindle. Tool change system is typical for milling machines. Automatic tool change system is in form of chain float position system with tool changer. Big advantage of this solution is larger number of tools then in lathe turret. Longer tool exchange time is a price of the one. To make tool exchange time shorter multi indexing spindle is used. For lathe tool spindle is indexed I 12 angular positions. To use this function a special multi positions tools are needed.

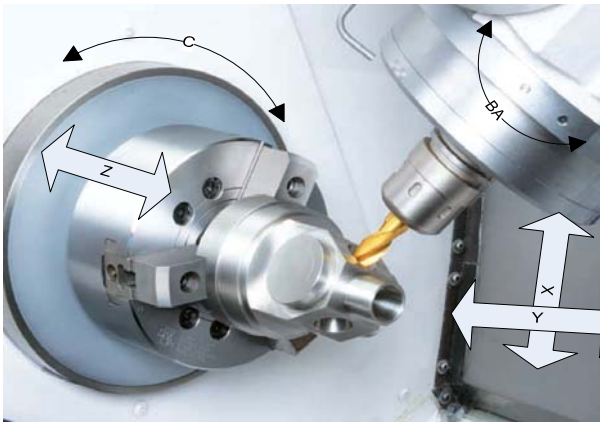


Figure 4. Example of machining

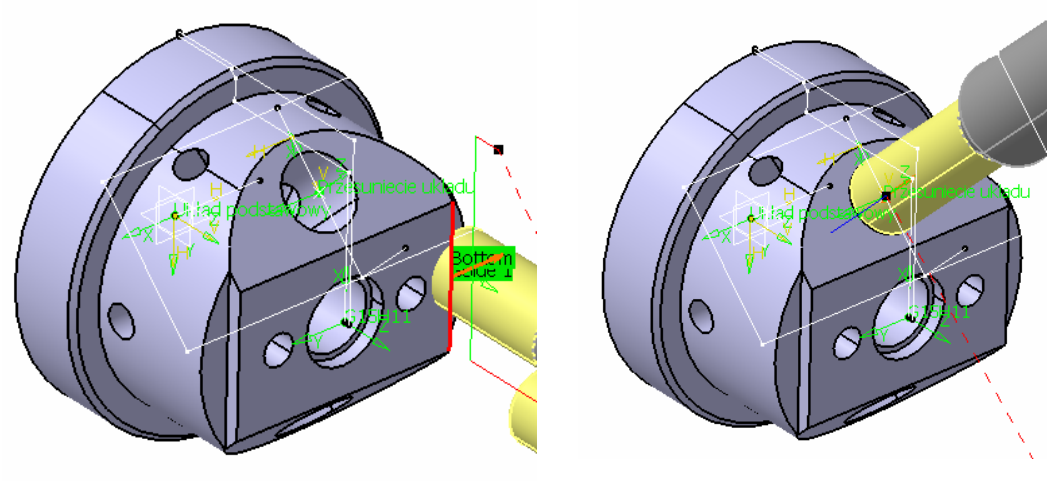
Machining capabilities are presented in the fig.4. It is easy to understanding programming difficulties when imagine the process. There are a few solution for milling operations. Some functions of coordinate system conversions are offered by the control. We can use milling like coordinate system when Z axis is aligned with axis of rotating tool spindle and interpolation at virtual X,Y (X+Ys) planar is done. Big problem in this case is non proportional working area volume, because limits of the each axis are differ. Like at all lathes X axis is situated mainly above workpiece rotation axis. This is a reason that Y axis (X+Ys) is relatively

short (+/-70mm in this case). It makes impossible to machining face or side contours by use X+Y axis kinematics when Y dimension is bigger than 140mm or machining position is outside the Y axis limits. But for the face contouring it is possible to use X+C kinematics which gives possible to machine bigger contours when limitation is only maximum turning diameter. Both methods have some advantages. By use the real Y axis better linearity and flatness arte achieved. By use C+X axis larger face contours have been generated. For both method programming is completely different and decision which one of the method is selected depend on programmer. This illustrate main problem for CAM systems - how to select proper methods of machining – only one solution is a highly specialized postprocessor.

3. PROCESS PLANING

To illustrate the example problem (fig 5) was used. Final effects of toolpath planning are shown in the figure 7. Machining operation contains: facing, rough outside turning, finish turning, axial central drilling, axial non central drilling, slant facing by milling, slant drilling, side drilling, milling of contours and milling of pockets.

First problem to solve is central axial drilling. On lathes with live tools are two ways. By rotation of workpiece and use non rotating tool - powerful method for big bores by use power of stock spindle power. Second method is to use live tool. Rotating tool is better for smaller diameters which requires bigger rotational speed. Usually tools are rotating with bigger rotational speed than workpiece. Most of CAM systems didn't recognize difference in this case. Decision is taken by postprocessor and generally one method is permanent.



a) *Axial Contouring* b) *Slant drilling*

Figure 5. Example of drilling and milling cutting operations

This is permanent because native APT generated for drilling cycle look like:

CYCLE/BRKCHP, *75.000000,, 25.000000,,$*
1.000000, 0.200000, 90.000000,ON,0,0, 1.000000,

There are no difference depending on method. In the control different cycles are used for those purpose. For this case special code should be added for cycles which permits recognize kind of cycle and use proper method of NC cycle.

Second example is drilling of two non center axial holes. It is possible by use both noticed above methods using XY or XC axis. In this case programmer should known machine limitations and decides which of the method is better and possible to use.

Third examples of problem are slant machining. CAD systems changing angular position of tool only and generating APT which describes normal vector of tool axis:

TLAXIS*/ 0.707107, 0.000000, 0.707107*

The problem is that in the multitasking machine tools real angular position is the result of turret rotation (a few reference positions) and tool holder and method of correction. In true normal vector of tool not always is equal to turret vector. For this purpose it is necessary to make special Postprocessor Words. Situation is more complex when machine tool is equipped with subspindle, lower turret etc.

4. TESTING OF SOFTWARE

Problems presented in point 3 are the reason to build specialized postprocessor application for CATIA V5 Manufacturing environment. Application is made as stand alone. Postprocessor required proper APT CATIA V5 native format. This postprocessor was dedicated for OKUMA Multitasking machines MULTUS, MACTURN series witch OSP CNC Control. Postprocessing results have been successfully tested on OKUMA MULTUS B300 machine tools.

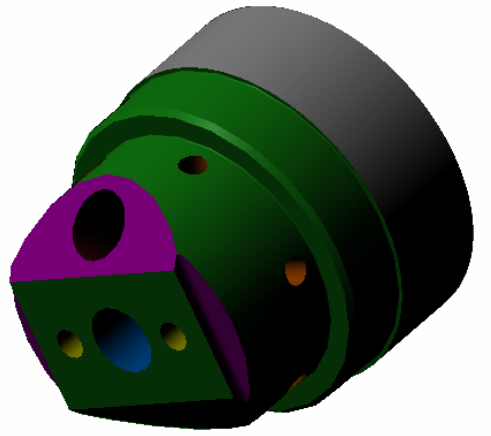


Figure 6. Vizualization made by OSP controller

Figure 7. View of premachined workpiece

5. SUMMARY

Result of works is postprocessing application which permits to comfortably programming multitasking machine tools by use of CAM system without any compromises. It is very important to know machine limitations and capabilities and relations between CAM and Postprocessor to make programming effective. Long time experience in CNC application and knowledge of CAM system by author of the paper it was a way to built up solutions that is possible easy to adopt for any other multitasking machines and another APT based CAM systems.

6. REFERENCES

[1] CATIA Documentation, Version 5 Relase 18, Dessault Systemes 2007
[2] OKUMA OSP-P200L, manufacturer manuals
[3] Dessault System on-line: http://www.3ds.com