

# THE METHODIC PROCEDURE FOR CREATION OF NC PROGRAMMS BY USING SINUMERIK 840 D

Dávid TÓTH<sup>1</sup> and Peter KOŠTÁL<sup>1</sup>

**ABSTRACT:** This paper deals with design of the manufacturing process of rotary parts in environment of the drawingless manufacturing by using SIEMENS control system. Aim of this paper was propose the methodology for EmcoTurn105 turning machine by using Sinumerik 840D control system.

**KEY WORDS:** Sinumerik840D, drawingless manufacturing, methodology, NC programm

## 1 INTRODUCTION

There are few ways to create manufacturing programs for different machines using different CAD/CAM programs. In this paper we will show the procedure for programing EmcoTurn105 turning machine by using Sinumerik840D control system. The iCIM3000 system which includes EmcoTurn machine was described in first part of this paper. The machine and its control program Sinumerik 840D used for the machine programing is briefly described.

Machine (EmcoTurn105) is programmable also in other CAM programs but for this paper we chose Sinumerik 840D since it is a part of machine (already included in the machine. New NC programs can be formed simply without any additional corrections. Overall process is divided into four sub-processes in this paper. Some of these processes will be further divided and described independently later.

## 2 ICIM3000 SYSTEM

Production with integration of computer support for all interconnected phases of the production system is called Computer Integrated Manufacturing (CIM). A network of connected computers includes related activities for production, starting with the product marketing and ending with expedition to the customer.

CIM is a concept for structuring of industrial enterprises. Nowadays, the manufacturing technologies demand CIM concepts, which can be realized through the capabilities of information processing.

The idea of integrating different areas of CIM, as production planning and control (PPC), computer aided design (CAD) and computer aided manufacturing (CAM) is explained via operating chains and put into a CIM architecture based on a hierarchy of EDP systems.

The CIM is a computer controlled production and its open system is iCIM. The iCIM 3000 is a training and open model system of CIM made by FESTO company. In Figure 1 is shown the 3D model of iCIM. System in this figure consists from stations which are marked here as numbers 1-8.

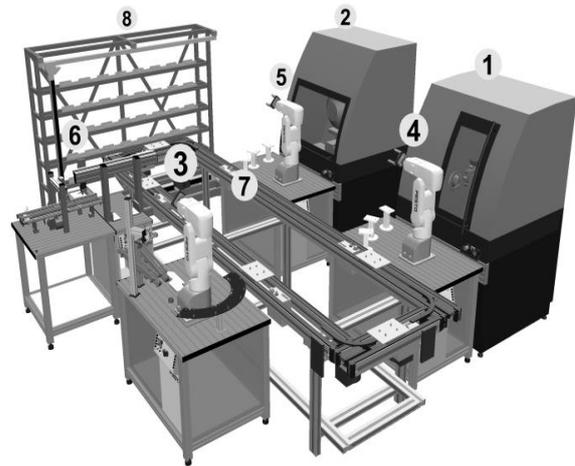


Figure 1 The 3D model of iCIM3000 by FESTO

1 - Concept Turn, 2- Concept Mill, 3 - Flexible Robot Assembly Cells, 4; 5 - Service robots of concept machines, 6 - Pallet Handling and Quality Station, 7 - Pallet Transfer System, 8 - Automatic Storage / Retrieval System

## CNC feeding turn: EmcoTurn105

The CNC feeding turn (Fig. 2) is responsible for production of single parts. The robot takes raw parts from the magazines to equip the turning machine. Parts are finished, corresponding to their order. Since the parts comes to conveyor system,

<sup>1</sup> Slovak University of Technology in Bratislava, Faculty of Material Science and Technology in Trnava  
E-mail: karol.velisek@stuba.sk, peter.kostal@stuba.sk

the finished components are available in pallets. The Concept Turn 105 is a desktop lathe driven by interchangeable CNC control software running on a commercially available PC. This dual purpose turning center is the ideal solution for training students in further education when large industrial machines are not suitable. Its role in education is defined by its interchangeable control systems. The software can be switched in a minute from FANUC 0T/21T control to SIEMENS 810/840 fully supported by Fanuc/Siemens CNC control system site licenses and control keyboard operation. The Concept Turn 105 can be integrated into FMS and CIM systems. The robotics interface can control equipment as automatic door, etc. and also connect to other machines. The DNC interface allows programs to be started from the master computer. The Concept Turn 105 is available with a comprehensive machine stand which has a built cabinet for tooling and PC also swivel table for Control Keyboard and Monitor.



Figure 2. CNC feeding Turn: EmcoTurn105 [3]

### Sinumerik 840D

The Sinumerik 810D/840D is a part of the changeable controls in WinNC. WinNC allows the user to learn everything about CNC industry controls that are common in the market on a single machine. One of WinNC controls is Sinumerik 810D/840D controller. It corresponds with the particular control in handling and function. This controller can be used on a commercial PC. The user has the possibility to work on the PC as well as on the machine control and it gets familiar with all aspects of control programming in detail.

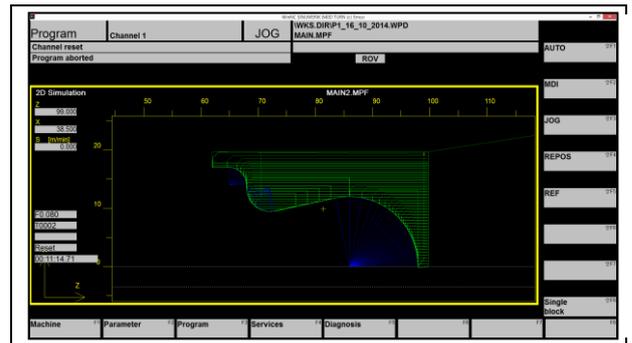


Figure 3. Sinumerik840D

#### Highlights

- Operation like the industrial control Sinumerik 810 D & 840 D MMC 100.2 with horizontal and vertical soft keys (cycle version 6.xx)
- 2D Graphic simulation with auto zoom
- Detailed error message about drive problems and programming faults
- Operation via PC keyboard, digitizer, control keyboard or mouse
- Languages: German, English, French, Spanish, Italian, Swedish, Russian, Chinese, Czech, Dutch, Hungarian
- Main programs, sub-programs tool data, register and workpiece management will be stored permanently
- Position offset will be stored by pressing the store button
- Program titles can be given without specifications
- Window oriented, modern user interface
- Comprehensive editor with intermediate memory storage function
- Large memory of NC programs (limited only by the size of the hard disc)

### 3 DEFINITION OF THE MAIN POINTS FOR DESIGN OF THE MANUFACTURING PROCESS IN SINUMERIK 840D

1. Checking the drawings and clamping
2. Defining the applied tools
3. Setting of the 3D simulator
4. Programming

#### Checking the drawings and clamping

Prior the performing of any kind of operations in Sinumerik 840D or other CAM software we must check the drawings and clamping. This will be realized in first step of design. It is necessary e.g. for choosing the zero point position or drawing use to transfer information about the shape, dimensions and manufacturing requirements of the product,

which are necessary for further contour drawing or for choosing the correct processing cycles.

The drawing and the clamping must be checked since the program will started. To avoid collisions, the position of the raw part in the clamping device must be checked. If the position is wrong, the position of zero point is changed and evaluated.

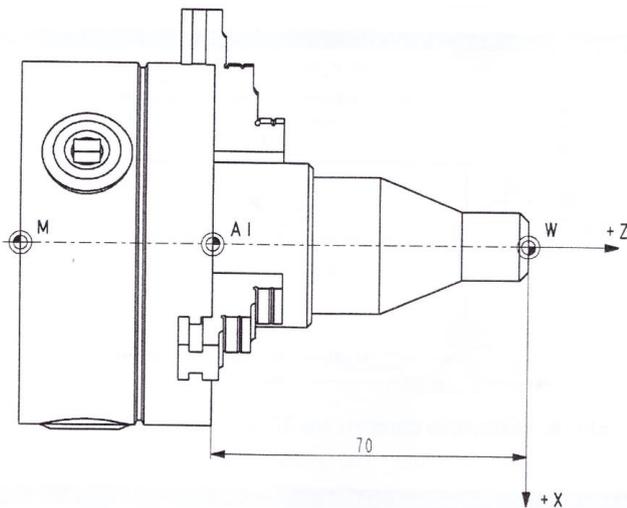
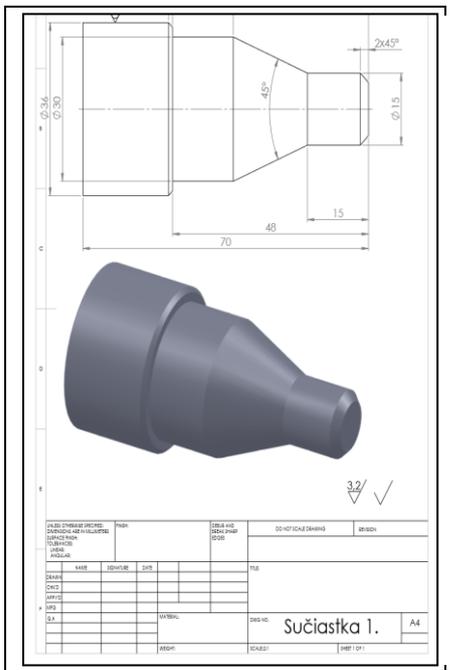


Figure 4. Examples of drawing and clamping devices

### Defining the applied tools

We must set the tools that are available in revolving turret in Sinumerik. The positions in application software must correspond with the tools position in the machine. If the required position of tool is occupied, we must remove and replace it

with a new tool. The replaced tool needs to re-enter new parameters. When the required position is free, we can add the tool with parameters directly.

In Sinumerik 840D we have to choose the function key called PARAMETER (F2); TOOLOFFSET (F1) in main “funcion menu”; where we enter the tool position. We need to select Tno.+(↑F1) or Tno.-(↑F1). When the position is occupied we must delete the current tool with function key DELETE (↑F5); and DELETE TOOL (↑F4). We can use this procedure to delete the cutting edges of the current tool only.

When we have the position free, we can add the given tool. We push the function key in tool offset menu selecting a button NEW (↑F8); then NEW TOOL (↑F4) where we set the tool number. The tool number is the position of tool in the revolving turret. After setting the tool type using the list that the software offers us, we can accept our selection by OK button (↑F8).

Next procedure is to set the tool parameters. For example tool length, compensation or radius compensation. This step is important when we want to turning parts but it is not necessary to set these parameters for the 3D simulator. However, we must set all tools that we will need during the programming.

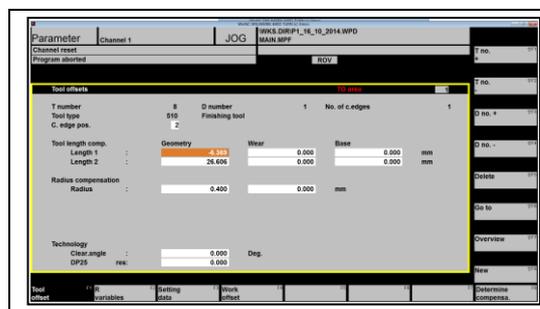


Figure 5. Tool offset

### Setting of the 3D simulator

When we want to turn in the real machine it is important to simulate the overall process earlier. This simulation is realized with the 3D view module in Sinumerik. We can avoid collisions of the tool with the stock part or with the clamping devices or avoid many other problems since we start to carry out the production in real environment. To set the parameters of 3D simulator according to setting of the machine is very important.

After finishing of mentioned steps we can create the basic folder, where our sub-programs will be saved. In the main function menu we have to

choose a function key PROGRAM (F3). We find the primary program menu list here. There is possible to create new folders and does many operations with them e.g. selecting, copying, renaming and removing. Now we create a new basic folder in the next step. We choose the function key NEW ( $\uparrow$ F1), name this basic folder, his type will be WPD and confirm it by OK button ( $\uparrow$ F8).

We open the basic folder, which was created. We create a sub-program in this folder. The available operations for this file are similar as operations with basic folder (new, select, copy). We can create two different types of sub-program, MPF and SPF, which are included in WDP folder. Function key NEW ( $\uparrow$ F1) is used for creating new subprograms MPF or SPF. We can name the program as we want but it is good to name the primary part of program MAIN. The basic program will be written here. There is easy to find the main subprogram (SPF) or part of program (MPF).

We open the created sub-program file. Menu of this folder will be demonstrated in the next chapter. To set the 3D simulator we choose the function key 3D-VIEW (F5). This module must be installed, otherwise this function key will not available.

We set the part according to the drawing in first step by using the function key WORKPIECE ( $\uparrow$ F5) thereafter we set up the tools for the 3D simulator. This setup must be the same as the tool setup for the machine and must correspond with the tools in the revolving turret. In 3D-VIEW (F5) menu we choose the TOOL ( $\uparrow$ F6) key. We mark the tool position that we want to set up with tools from the Tools list. We can take ( $\uparrow$ F1) or remove ( $\uparrow$ F2) them, reset the tool colours ( $\uparrow$ F5), change their colours, or assign them in the simulator. When we have finished setup of tool for the simulator, we can confirm it with OK button( $\uparrow$ F8) .

We can leave 3D-View using the RESET key (F6); or modify it by EDIT (F1) keys. We must reset the simulation first, since the software would not let us edit/re-edit the program. Now we can start to edit the CAM program.

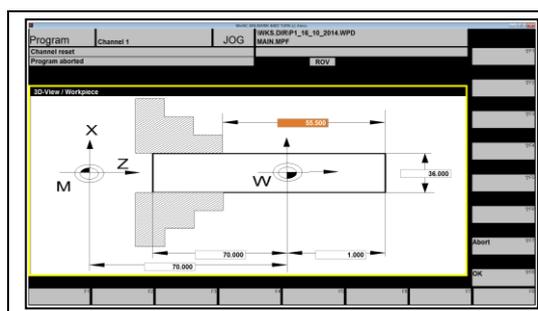


Figure 6. 3D-View set up menu for the tools and for the workpiece

### Programing

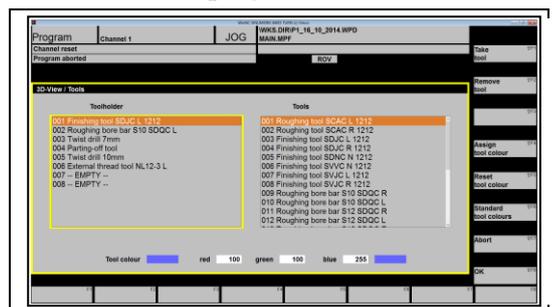
We will use for programing our main program (MPF). The complete programing process could be distributed to three phases. We must set the basic conditions for the overall process in first phase. The contour drawing, if is necessary, will be performing in the second phase. Last phase will consists from the using of cycles for tooling the contour or using the standard cycles for turning operations as groove turning, undercutting, holes drilling, thread cutting.

#### 1. Setting the basic conditions

The first thing we have to do when we start the program writing in Sinumerik 840D software is to set the basic conditions that are necessary for the correct working of the overall process.

*Setting the zero point:* For EMCO turning machine is zero point of the machine "M" on the axis of rotation at the face of the spindle flange. This position is unsuitable as a starting point for dimensions. The coordinate system can be moved to a suitable point in the working area of the machine with zero offset. In the operating area the PARAMETER (F2); and WORK OFFSET (F4) are available in four adjustable zero offsets. When we defined a value in the offset register, this value will be called in program by using a command G54-G57 and the coordinates of zero point will be shifted from zero point of the machine "M" to zero point of piece "W". The piece zero point within a program can be shifted in any number by using command TRANS Xxx Yyy Zzz.

*Tool selection:* For successful continuance in turning we must call the correct tool. Mark "T" is marking of the tool number, and D is the tool correction number. To every tool number T up to 9 correction numbers can be assigned. The Sinumerik



840D control describes the correction data “D” as edge. Setting the tool parameters was described earlier. Depending on usage in the program, the tool is called with command e.g. T1 D1 or T1 D2. The command Tx Dx activates the tool correction D and changes the tool. The data for the tool correction (tool length, tool radius, etc.) will reading from previously defined tool called “Data register”.

*Setting the turning process parameters:* After we have selected the correct tool, we need to set up parameters of machine for the turning. The rotation speed for the spindle will in the form “S1500” where number is value of the rotation speed. Starting rotation speed is 1500 rpm. We must then select the turning direction of the spindle with “M3 or M2”. At the end we set the feedrate with command “F100”, where number is the rate of the feed. We can read these values from the tool catalogues or we can choose verified values that were successfully tested.

Feedrate values are not influenced by G70/71, only the machine settings data are valid. After changes G93-G95 commands, the F commands must be every time re-edit. The feedrate F is only valid for path axes, but not for synchronous axes. The address F is the feedrate in mm/min for G94 and for G95 in mm/rev.

When G96 is active, the spindle speed is depends on the respective piece diameter and it is automatically modified. In order, the cutting rate S (mm/min) remains constant the whole edge of the tool. Thus the uniformity is increased with the surface quality of turned parts. If we turning a piece that greatly vary in diameter, it is suitable to specify a speed limit for the spindle. This prevents excessively high speeds with small diameters. LIMS active with G96: G96 S100 LIMS=2500

**2. Free contour programming:** The free contour programming is a support tool for the editor. An integrated contour calculator calculates possibly missing parameters as soon as they result from the other parameters. Additionally, contour transition elements as chamfer are available. The selection is carried out in the operation area PROGRAM (F2); we have to select an already existing program or create a new subprogram (SPF). The contour editor is opened by softkeys SUPPORT(F4); NEW CONTOUR (↑F1) and existing contour can be recompiled by using RECOMPILE (↑F7).

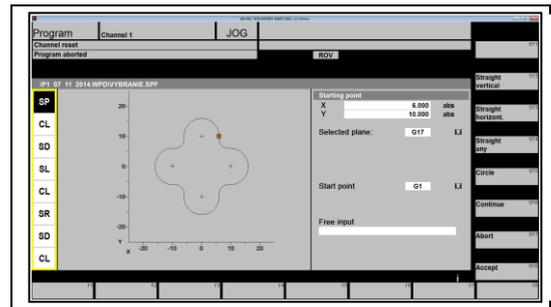


Figure 7. contour programing window

*Determine the starting point:* The input of contours with the starting point is determined. The coordinates for X and Z must be absolutely defined. After selecting the plane (G17/ G18/G19), we will set indication dimension for facing axis and select a start point (G0/G1).

*Contour drawing:* The contour can be drawn by soft keys for the vertical (↑F2), horizontal (↑F3) or circle (↑F5) keys. You can set many adjustments for all of these function keys e.g. end points or points for the transition to the next element. There is possibility to select between a chamfer and radius. We can select between “SELECTION” options with the spacebar or the softkey ALTERNATIVE(↑F1),

CONTINUE(↑F6). The softkey CLOSE CONTOUR is used for closing a contour on one straight line. All previous elements must be accepted.

**3. Standard cycles:** Basic step in programming is using of standard cycles in Sinumerik840D. Standard cycles are called by the SUPPORT (F4) softkey from menu list. There are shown standard cycles e.g. drilling, milling and thread cutting. When we open one of these options, other cycles will appear on screen. We have to choose the correct cycle that we need for turning.

Cycles calling is follows: Cycle (parameter1, parameter2, ...). We must set up all parameters that are required for the program. We can find all parameters for the cycles in the *cycle description*. From the cycles are called only parameter values that were entered. Consequently the sequence of parameter must be kept and for that values defined. If a parameter is not required, an additional comma must be given instead of his place.

The cycle description starts with a survey table containing the cycles and their parameters. Consequently, is given the exact description of the parameter. In the survey table the cycles are based on the previous cycle, it means, that only these parameters are described here that are different to previous or a new cycle. We can also call these cycles with MCALL command.

**EVERY PROGRAM MUST BE ENDED BY COMMAND M30 FOR THE MAIN PROGRAM.**

#### 4 CONCLUDING REMARKS

This part will emphasize the contributions of the paper and the future applications in the research field.

#### 5 ACKNOWLEDGEMENTS

This paper will be created with support of the VEGA 1/0285/12 – Research on the possibilities of „intelligence,, implementation into the assembly process

#### 6 REFERENCES

- ▶ Festo didactic GmbH & Co.KG (2013) Manual iCIM: January 2013. Denkendorf
- ▶ Festo didactic GmbH & Co.KG (2013), Software description EMCO WinNC SINUMERIK810/840D Turning : January 2013. Denkendorf
- ▶ Delgado Sobrino, Daynier Rolando - Košťál, Peter: (2013) A few analysis and customization issues of a new ICIM 3000 system: the case of the material flow, its complexity and a few issues to improve. Annals of MTeM for 2013 & Proceedings of the 11th International MTeM Conference, Cluj-Napoca, Romania, 17th - 19th October 2013, s. 57-62. In: Academic Journal of Manufacturing Engineering. - ISSN 1583-7904. - Vol. 11, Iss. 4 , s. 36-41
- ▶ Delgado Sobrino, Daynier Rolando - Košťál, Peter (2012): Contributions to the design of the material flow at an intelligent manufacturing cell: advantages of the use of simulation. In: Machine Design. - ISSN 1821-1259. - Vol. 4, No. 2 , s. 83-88
- ▶ Holubek, Radovan - Košťál, Peter (2013): The intelligent manufacturing systems. - reg.: Scopus. In: Advanced Science Letters. - ISSN 1936-6612. - Vol. 19, No. 3 , s. 972-975

▶ Danišová, Nina - Ružarovský, Roman - Velíšek, Karol (2011) Design methodology of automation equipment and control system in the intelligent assembly cell. - reg.: Scopus. In: Applied Mechanics and Materials. - ISSN 1660-9336. - Vol. 58-60 , s. 2407-2412

▶ Danišová, Nina - Majerík, Jozef (2011): Sensoric system for identification of jaws in the jaw buffer and intelligent fixture. In: Annals of Faculty of Engineering Hunedoara - Journal of Engineering. - ISSN 1584-2673. - Tom IX, Fasc. 3 , s. 441-442

▶ Danišová, Nina (2014): Digital image processing in the camera system of assembly systems ICIM. - reg.: Web of Science, Scopus. In: Applied Mechanics and Materials. - ISSN 1660-9336. - Vol. 474. - , 2014, s. 173-178

▶ Charbulová, Marcela - Matúšová, Miriam - Cagáňová, Dagmar (2011) Intelligent production systems and clamping systems for intelligent production systems. In: Journal of Production Engineering. - ISSN 1821-4932. - Vol. 14, Number 1 , s. 63-66

▶ Bučányová, Marcela (2014) Component base for CNC processing center EMCO Concept TURN 105. - reg.: Web of Science, Scopus. In: Applied Mechanics and Materials. - ISSN 1660-9336. - Vol. 474. - , 2014, s. 242-248

▶ Mudriková, Andrea - Charbulová, Marcela (2010): Material flow in flexible manufacturing cell. - článok publikovaný v časopise AMO Journal, ISSN 1313-8987, Iss. 3, Vol. 1, 2010, str. 100-105. In: AMO Conference. - ISSN 1313-4264. - 10. International conference, 27 - 29 June 2010 AMO 10, 27 June - 01 July 2010 CEEPUS (2010), s. 1-9

#### 7 NOTATION

The following symbols are used in this paper:

$A_{cc}$  = area of core concrete;

$A_t$  = total area.