6th INTERNATIONAL MULTIDISCIPLINARY CONFERENCE

THE METHODS OF NC PROGRAM CREATING

Katarína, Monková, Technical University of Košice, Faculty of ManufacturingTechnologies, Bayerova 1, 080 01 Prešov, Slovakia

Abstract: *The article describes the basic methods of NC program creating, including the basic steps from part geometry creation by means of CAD system to NC program creation and the basic types of data files pipelines, which translate one data format into another in this process. It deals with advantages of these methods and with the problems, which originate in the process of creation.*

Key words: *STEP-NC, ISO 6983, CNC control, CAD/CAM systems*

1. INTRODUCTION

For the majority of NC users, NC is about productivity and flexibility - making a lot of parts, and many different parts, on one machine tool. This was true even before computer numerical control (CNC) superseded an earlier generation of machine tools that did not have the benefit of microprocessor-based control technology. With the level of automation being used in CNC machining the level of consistency and quality increased. CNC automation eliminated errors and provided CNC operators with time to perform more tasks. The CNC automation also allowed for more flexibility in set-up and job changes and today's CNC machines are productive, capable and flexible, too.

What makes CNC so flexible and productive is the ability to run different workpiece programs. With the right program, machining is a pushbutton affair. Provided that the proper cutting tools and set up fixtures are available, all a CNC machine needs to begin automatically machining a new and different program. [5]

How to create those new and different programs has taken various approaches. Many CNC machines can be programmed on the shop floor, with the operator entering data at the control panel. This method has been very popular, especially for simpler workpieces. Programs can also be prepared "off-line," away from the machine tool, using computer-aided manufacturing (CAM) software. This method is most often used for more complex workpieces. The latest CAM software for the PC (personal computer) provides many automated features that make NC programming largely a push-button affair, regardless of how simple or complex the workpiece might be.

However, the biggest change in recent times for CNC systems lies with the introduction of STEP-NC as the interface between CAD/CAM systems and CNC programs.

2. BEFORE STEP-NC

In the beginning, there were only CAD systems. Engineers used CAD systems to draw pictures of parts. CNC program was developed manually. That is, a programmer was sitting down to write the program armed only with pencil, paper, and calculator. When the applications became more complicated and especially when new programs were required on a regular basis, the writing of programs manually became much more difficult. To simplify the programming process, a computer aided manufacturing (CAM) system can be used.
A CAM system is a software program that runs on a computer (commonly a PC) that helps the CNC programmer/machinist/manufacturing engineer to program from the drawings and with the whole programming process. The making of drawings, and programming parts from drawings, was (and still is) time consuming and subject to a lot of human error. Someone got the bright idea to eliminate this to-and-from drawing step, and integrated CAD/CAM was born. Integrating computer-aided design with computer-aided manufacturing (CAD/CAM) system produces quicker and more efficient manufacturing processes. This compatibility of CAD/CAM systems eliminates the need for redefining the work piece configuration to the CAM system.

Most computer numerical control (CNC) machines are programmed in the ISO 6983 “G and M code” language. Programs are typically generated by computer-aided manufacturing (CAM) systems that use computer aided design (CAD) information. However, ISO 6983 limits program portability for three reasons:

- First, the language focuses on programming the tool center path with respect to machine axes, rather than the machining process with respect to the part.
- Second, the standard defines the syntax of program statements, but in most cases leaves the semantics ambiguous.
- Third, vendors usually supplement the language with extensions that are not covered in the limited scope of ISO 6983. [3]

Fig. 1 shows how design data is communicated to manufacturing in current practice. Design creates the specification for a product as a 3D model. Detailing decides the manufacturing requirements for the product by making a drawing. Path planning generates tools paths. Manufacturing controls production. The job of design is performed using a CAD (Computer Aided Design) system, the job of detailing is performed using a drawing CADD (Computer Aided Design Drafting) system, the job of path planning is performed using a CAM (Computer Aided Manufacturing) system, and job of manufacturing is controlled using a CNC system. In many cases the CAD, CADD and CAM functions are combined into a single integrated CAD/CAM system but in all cases the CNC function is performed by a separate system. [4]

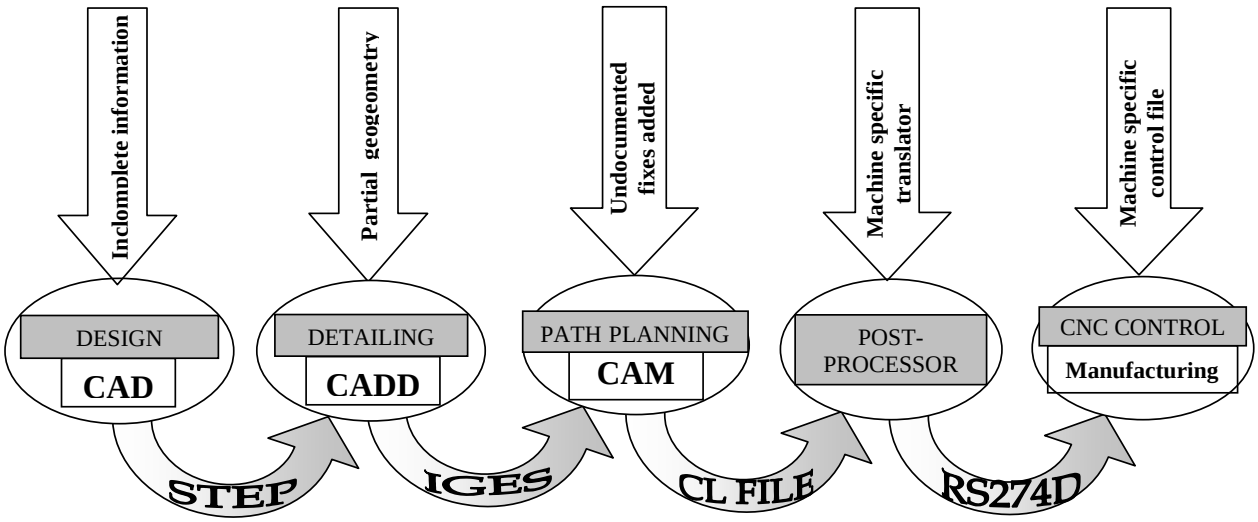

Fig.1 The actual interface between CAD and CNC control

Data flow in mechanical engineering:

- First the information about a product is imported into the CAM system. Usually 3D CAD model is imported.
- In mechanical engineering CAM is used to calculate toolpaths to cut material. The CNC programmer just specifies the machining operations and the CAM system creates the toolpath, usually written in CL data (Cutter Location data) file.
- Calculated toolpath is imported to the postprocessor which converts the CL data to the NC program - the specific machine codes that are required to operate numerically controlled machine tools. Machine codes vary by machine tool. The output from a postprocessor should be usable in the controller without further modification.
- NC program written in a notation called G-code is exported to the NC machine and the manufacturing process can begin.

One of the most important links in the CNC machining process is postprocessor. The company can have best programming system, the best DNC system, and the most accurate machines, but without dependable postprocessors the code getting to its machine could cause problems. Most machines require some tweaking to the postprocessor to make it to produce code to user's likings and programming habits. Most CAM software comes with built in postprocessor; however there are many stand alone postprocessor. It is the last software link between an ideal CAD model and a “real” machined part.

Some problems on the subject of the postprocessors are:

- Absence in a CAD/CAM system of the ready postprocessor for specific "machine tool /machine control" combination, and also far too high cost of new postprocessor development.

- As a rule, it is very difficult and often impossible, to modify the postprocessor available in the CAD/CAM system to the specific "machine tool / machine control" combination. In general, the technologist is forced to permanently correct wrong NC program file with the text processor.
- Usually generalized postprocessors of majority of the CAD/CAM systems are very difficult to adapt to NC/CNC equipment, made in 70 - 80s.
- Each new CAD/CAM system requires an individual postprocessor for the same NC equipment, so the company has to pay twice for the same stuff.
- Creating a custom postprocessor for each unique machine is often difficult and costly.

All problems with the postprocessors eliminate STEP NC, which replaces RS274D, M and G code standard, as the primary interface between CAM and CNC systems.

3. STEP-NC

The concept behind STEP NC is simple. It enables a product model database to serve as direct input to a CNC machine tool. No separate files of tool paths. No G or M codes. No post processors. This is a radically different approach to CNC programming. STEP NC is an extension to STEP, the STandard for the Exchange of Product model data.

STEP is the international standard that specifies a neutral data format for digital information about a product. STEP allows this data to be shared and exchanged among different and otherwise incompatible computer platforms.

STEP NC standardizes how information about CNC machining can be added to parts represented in the STEP product model. [2]

STEP NC defines a CNC part program as a series of operations that remove material defined by features. The features supported include holes, slots, pockets and volumes defined by 3D surfaces. Each operation contributes to the manufacture of a feature by defining the volume of material to be removed, the tolerances, the type of tool required and some basic characteristics such as whether this is a roughing or finishing operation. The operations are then sequenced into a work plan that converts the stock into the final part. The work plan may be sophisticated and include conditional operations that depend on the results of probing operations, and it may be divided into sub-plans to be executed concurrently on machines that have multiple cutting heads.

STEP NC has the Application Protocol number AP-238 within the STEP framework. A key feature of STEP NC, AP-238 programs, is that they are machine and organization independent. If a machine has the underlying capabilities (axes, table size etc), then a STEP NC "compiler" should be able convert the part program into a sequence of tool movements for that machine. If a CNC has a Tool Cutter Programming (TCP) interface then the tool movements can be executed directly without converting to axis movements. This has significant consequences for industry. [7]

In the new method enterprises can continue to use their existing systems for CAD, CADD and CAM, but the end result is sent to the CNC as a STEP NC AP-238 file instead of an RS274D file. **Fig. 2** shows the modified pipeline. [4]

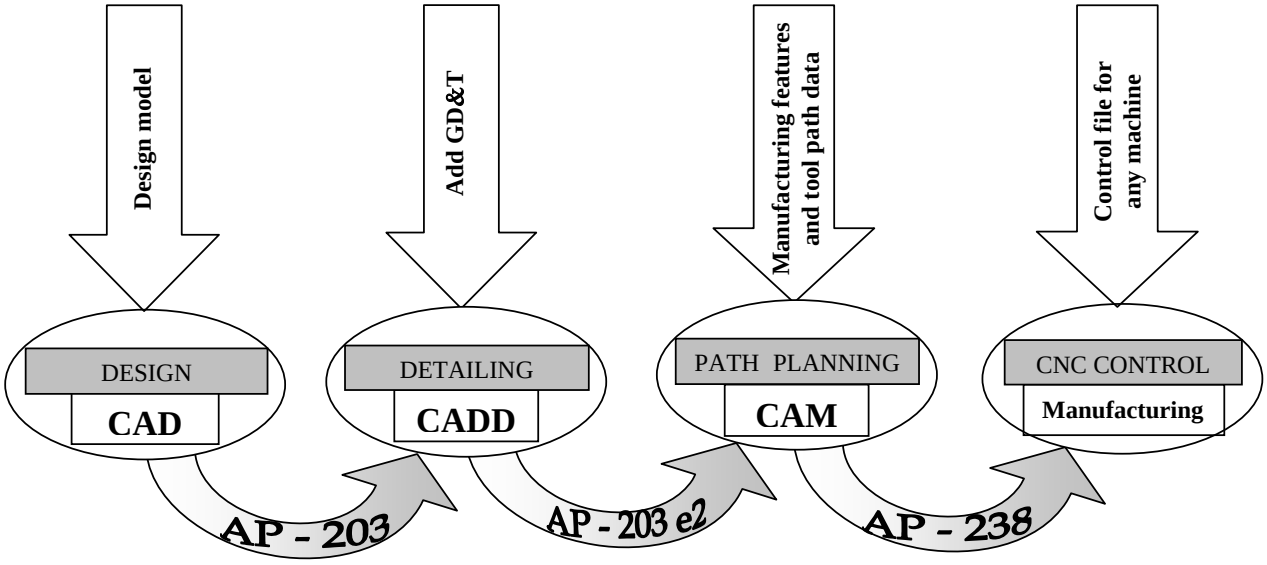

Fig.2 The new interface between CAD and CNC control

The change is small because no systems need to change only the interfaces, but the advantages are significant:

- The AP-238 file can make developing a CNC part program more efficient because the programmer only has to describe the tasks to be performed on the machine and not the tool motions necessary to achieve those tasks.
- The AP-238 file allows a CNC to optimise and check a part program for the tooling available at the time of manufacturing instead of having it fixed at the time of planning.
- The AP-238 file reduces the requirement for drawings on the shop floor and it allows manufacturing to send requests for changes back to design by annotating the original full fidelity design information.
- The AP-238 file makes manufacturing data portable between machines and organizations and allows a part to be made on any machine with sufficient resources (axes, table size etc).

STEP NC allows a complete database of machining information to be built around the digital product model and ultimately makes it possible for this enhanced product model to serve as machine tool input. This database is structured such that part features are linked to specific "working steps," generic descriptions of various machining operations. STEP NC working steps are roughly equivalent to machining commands formatted as traditional M and G codes. With the concept of "working steps" in place, the manufacturing process becomes

streamlined. Now, a machine tool can receive a file with STEP NC data, "know" what it means, and proceed milling the piece without any more instructions. There will be no more programming the machine tool for each individual piece. Moreover, the benefit of the standard goes further. With a set of standard "working steps" in place, all manufacturers will be able to share information reliably and instantaneously.
STEP NC converted CAD file that is completed on the east coast can be sent over the internet to a machine shop on the west coast and they can immediately start milling the part.

Machine tools with PC-based open architecture control systems may be able to install this software to upgrade to STEP NC compatibility rather effectively. The conventional input/output (I/O) structure and the servo system of the CNC machine do not need to be modified under STEP NC.

4. CONCLUSION

STEP NC (ISO 14649) is a worldwide standard developed by the International Standards Organization (ISO) to extend STEP so that it can be used to define data for NC (numeric control) machines. It is a new model of data transfer between CAD/CAM systems and CNC machines, which replaces ISO 6983.

STEP NC will allow CNC systems to become more intelligent. Today, these systems are forced to be dumb because they are not given any information about the product and process. With STEP-NC they can be given much better information. In the near term, this change will make CNC systems easier and safer to use. In the longer term, the consequences may be dramatic because many new applications will be enabled by the new information.

The rich 3D modelling information in STEP NC makes CNC systems easier to program and safer to use. The project that developed STEP NC has estimated that it can reduce the time required to program a CNC by about 35%, reduce the number of drawings that have to be sent from design to manufacturing by about 75% and decrease the time required to machine parts on CNC tools by about 50% for small to mid-sized job lots. [6]

REFERENCES

[1] KALPAKJIAN, S.: *Manufacturing, Engineering and Technology*. USA: Addison – Wesley Publishing Company, Inc., 1995

[2] KISHINAMI, T.: *XML-based High Level NC data Modelling for Manufacturing Data Management*. ISO TC 184/SC4/WG3, Charleston (2000/10/19)

[3] INTERNATIONAL STANDARD, *Data model for Computerized Numerical Controllers*, ISO-DIS 14649-1

[4] *www.steptools.com*

[5] *www.crdatnc.sk*

[6] *www.step-nc.org*

[7] *www.mmsonline.com*

Reviewer: doc. Ing. Jozef ZAJAC, CSc.