DATA INPUT AND DATA IMPLEMENTATION FOR NC PROGRAMS

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Abstract: *The article deals with the data input and data implementation for NC program. To store all information about production is very important today. It represents know-how of the plant what is the basis of its competitiveness. The flexible and continuous exchanging of information within the plant enables the increasing of production efficiency. The utilization of CAD/CAM system for NC data obtaining is suitable mainly at the complex shaped parts machining, where the accurate dimensions are unknown during the whole process production.*

Key words: *NC program, format, postprocessor, data, CAD/CAM.*

1. INTRODUCTION

Many of companies have already a CAD/CAM system and generate CNC-programs for various CNC-Machines. The way from designing to manufacturing the part is:

- To draw (construct) the part with the CAD-program available, either 2D or 3D and save the data of the contours in a file. The export-formats for contour-data are set up within the system, e.g. DXF in 2D or 3D, or STL in 3D.
- The contour-data are read in a CAM-program and completed with the machining data (which tool, which contour corrections, which feed, which depth per step etc.).
- A so-called postprocessor, which belongs to the CAM-program, generates one- or more CNC-programs for the control of the CNC-machine, e.g. a CNC-program for roughing and one for finishing.
- The CNC-program is read by the operational system of the machine and machining is done.

In order to manufacture a part, typically three different software programs are used (Fig. 1):

- CAD software to make the design of the part
- CAM software to calculate the tool paths based on the design, compensating for the cutter's geometry, adding spindle commands, etc.
- Control software to read the tool path and let the machine actually move along these paths

Communication between these three programs is done by means of special files. File translation is a process in which one data format is converted into another. Transferring CAD files electronically is the norm for today's manufacturing world.

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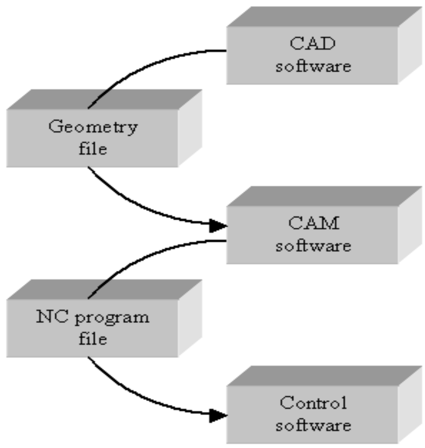

Fig. 1. Software programs and file translators needed for the part manufacturing.

2. FROM CAD TO CAM

In the beginning, there were only CAD systems. Engineers used CAD systems to draw pictures of parts. These CAD systems have been primarily 2D drafting and 3D wireframe, with a few surface modelling products. Today designers use CAD software to create a solid model of part. This model can be represented as a drawing or a CAD data file.

Only 8% of all parts were programmed from a drawing. This number includes all big and small NC using companies, as well as all PC and Unix CAM systems. Parts defined by a physical model or prototype make up 2%, while the remaining 90% of parts are programmed from electronic data [5].

Data translation is the change of electronic part data from one format into another format. From CAD to CAM the design is transferred using a file format translator for geometry data exchange. These are standard formats that in most cases can be used without any special configuring needed.

In the group of parts programmed from electronic data, there are three approaches to moving data between CAD and CAM: common database, direct read, and neutral file format.

In a *common database*, CAD and CAM software store their data in the same database. This is usually the case when the CAD and CAM products are from the same company, or when the products use the same solid modelling kernel, such as ACIS. The data, in this case, is the geometry, surfaces, and solids used to define the part. Since both the CAD and CAM software use the same data, it can be said that no translation is performed.

Direct read CAM software reads the CAD data in exactly the format used by the CAD software. Direct translators (Fig. 2) can exchange data directly into the CAD software's proprietary format to another. The ability to integrate seamlessly between software packages is the best situation for manufacturing shops because data transfer errors are virtually eliminated. By directly transferring files it is possible to eliminate the "middleman." In the end, less time is expended on data transfer issues so it is effectively reduced the product's manufacturing cycle. Direct translations generally work better because data transfer errors are virtually eliminated.

In *neutral file format*, CAD software writes the data into a form usually specified by a committee and the CAM file reads the same format. Examples of neutral file formats are IGES and STEP. Neutral file usage always requires two translations, one by the CAD product into IGES, and one by the CAM product out of IGES. Neutral translators (Fig. 3) are used to convert a proprietary CAD or CAM data format into a generally agreed upon industry standard. They are documented and available for anyone to use.

The primary neutral translators today are *IGES* for surface data and *X_T* or *SAT*, which were developed for solid modelling data. Neutral translators convert a proprietary CAD/CAM data format into an industry-standard format that is then read by another system and converted into its own format. The problem is that most CAD/CAM systems have their own "flavours" of standards such as IGES, requiring CAM systems to still interpret data. The idea of using neutral file formats is a good theory. In reality, however, each CAD package has its own exclusive properties and methods for packaging geometric data. A common term in the industry is "flavours," which means that a CAD export translator will start with a neutral format, such as IGES, and may add its own unique slant. Therefore, the results are not truly common to all in the industry. A CAM package must interpret the modified neutral translator. As new versions are introduced, the CAD translators must also be revised. So, in turn, CAM companies must upgrade the importing translator specifically for each new CAD version to eliminate compatibility issues.

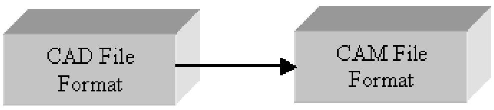

Fig. 2. Direct translator.

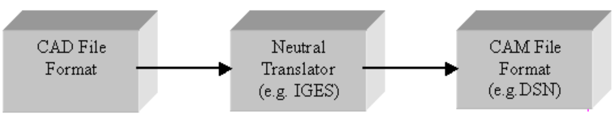

Fig. 3. Neutral translator.

File translators are written independently by each CAD or CAM Software Company. Some design teams may use several different types of CAD packages to communicate with different teams or complete specific functions. One design model may be exported and imported into several CAD packages before the final file reaches manufacturing. Also, the CAD/CAM software manufacturers develop proprietary formats for their software.

A well-built CAM translator translates all types of files quickly and accurately. Most often problems originate from an incomplete design model. A common source of error in part files is the receiving incomplete designs due to inexperienced or overworked CAD designers. The companies may not receive a complete model, or details that the translator is expecting may be omitted or lost. Or the CAM translator may not be fine-tuned enough for the specific CAD file format that is importing. So it will find the translating process offers plenty of room for error and miscommunication.

A solution of problems is actively being studied. The International Standards Organization (ISO) is working with American, European and Asian businesses and government agencies to implement a standard for transferring electronic data, called the International *ST*andard for the *E*xchange of *P*roduct Model Data (STEP). Essentially, STEP is a common structure, operating as a template, for sharing data among multiple users, across all functional areas. **STEP** (STandard for Exchange of Product model data) describes how to represent and exchange digital product information. STEP replaces IGES as the means by which graphical information is shared among unlike computer systems around the world. The big difference is that STEP is designed so that virtually all essential information about a product, not just CAD files, can be passed back and forth among users.

Digital product data must contain sufficient information to cover a product's entire life cycle, spanning design, analysis, manufacture, quality control testing, inspection and product support functions. STEP must, therefore, cover geometry, topology, tolerances, relationships, attributes, assemblies, configuration and more.

3. CAD/CAM SOFTWARE

A manufacturing engineer or NC programmer uses CAD software [5] to:

- Develop a computer model of a part that was defined by a drawing.
- Evaluate and repair the design CAD data to manufacturing tolerances. This is a surprisingly common task.
- Create new part models from the original design to allow for manufacturability. This would include adding draft angles or developing models of the part for different steps in multi-process manufacturing.
- Design models of fixtures, mould cavities, mould cores, mould bases, and other tooling.

A manufacturing engineer or NC programmer uses CAM software to select tools, methods, and procedures to machine the models defined in the manufacturing modelling section described above. Note that the user that performs manufacturing modelling is usually the same user that performs NC programming.

The biggest change in recent times for the CAD/CAM industry lies with the term "integration". Integration plays a very important role in the future of CAD/CAM products. Data integration is the ability to share part models (common data files or a common database). This is the most important type of integration for CAD/CAM.

The first CAM systems helped an NC programmer/machinist/manufacturing engineer program from the drawings. This 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.

Until recently, integrated CAD/CAM meant buying the same brand CAD and CAM products. Many companies provide such products today. Companies like Parametric Technologies (Pro/E and Pro/Manufacturing), Unigraphics, Dassault (Catia), SDRC (IDEAS), Computervision (CADDS 5), and others. All provide high sophistication, high power, and high cost solutions. These products typically provide data, interface, and application integration. Because of their cost and complexity, these products do not provide ideal solutions for everyone. In addition, once the customer picks the CAD product he likes best, he's kind of stuck with whatever CAM product they have. No mixing and matching of products is allowed [8].

The disadvantages of the traditional integrated workstation CAD/CAM system has contributed to the growth of the standalone CAM market as we know it today. These CAM products focus on NC programs, or both manufacturing modelling and NC programming. In general they are faster, easier, and far less expensive than their workstation-based integrated brethren. This class of products has grown in sophistication to rival the capabilities of the traditional integrated CAD/CAM products, while maintaining their lead in simplicity, efficiency, and cost. The only problems they have suffered from is a lack of integration with the original design modelling CAD system, and a lack of ability to access the CAD market. Now that is changing.

4. FROM CAM TO NC PROGRAM

Communication from CAM to Control software is done using NC program files, for which many formats do exist. In most cases the format will be a variation of the ISO/DIN G-code format. G-code is supposed to be a standard; however in practice each manufacturer chooses a slight different implementation. In other cases, a proprietary format is used. So for this communication the CAM software has to fine tune its output in order to meet the requirements of the used NC controller. This fine tuning is done by the Postprocessor.

The **Postprocessor** is the part of the CAM software that translates the tool path data into the correct file format when saving (in fact an export filter). In many current CAM systems the postprocessor can be configured by the user, making it easy to connect to any new machine. This is in contrast to the older CAM packages, where user had to pay to supplier money to obtain new postprocessor.

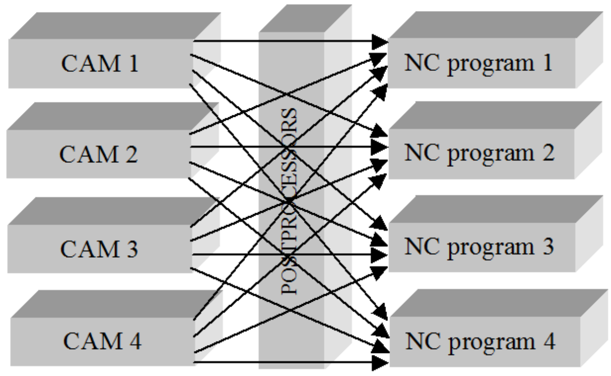

Fig. 4. Common used form of postprocessors for various machine/control combinations.

A separate program, generally referred to as the postprocessor (because processing takes place after the tool path has been generated), is used to format the neutral CL file into an NC program that is suitable for a specific machine/control combination.

General purpose CAM systems produce programs to run a variety of different types of NC machines including two-axis lathes; three-, four- and five-axis mills; wire EDM machines; punch presses; and turn/mill machines. In addition, there are virtually an unlimited number of machine and control software combinations, for which NC program is created. It follows that one NC program depends not only on the type of NC machine, but also on the type of used control software and on the type of used CAM or CAD/CAM software (Fig. 4).

This means that two different companies with exactly the same machine/control combination may choose to program in completely different way, using different combinations of available codes and functions. It is *one of a number of very common post processing problems* that CAM users typically face.

The second problem is simply getting correct codes output in the desired order at critical places in the NC program. The critical areas normally come at the start of the program, at a tool change, and at the end of the program. Getting the correct cutter compensation codes (diameter and length) output at the appropriate place is also difficult task. As previously stated, individual companies and even departments within the same company have different requirements and frequently adopt different methods for employing such things as tool changes and cutter diameter compensation. Thus, a postprocessor configured for one company may not be suitable for another. The CAM user is then faced with three undesirable choices:

- accept the output as generated,
- have the NC programmer edit and modify each individual NC program before it is sent to the machine tool, or modify the postprocessor configuration (requires personnel with appropriate expertise or aid from the CAM vendor).

The third problem is that an organization will purchase a machine tool, which their current CAM system cannot support. Many of the low to moderately priced CAM systems do not fully address the requirements of multi-axis machine tools, for instance. Limitations include the inability to control multiple rotary axes such as two or more rotary tables or heads. Even in cases where the rotary axes can be controlled, the CAM software's postprocessor may not be able to accurately calculate the feed rates necessary to keep the tool tip moving at the programmed feed rate as the rotary axes are moving simultaneously. Problems such as these are frequently overlooked until after machine tool is purchased and is on the floor waiting for a program. Changing or upgrading The CAM system is sometimes the only way to overcome such problems.

A fourth problem frustrating CAM users is that postprocessors frequently do not support special features contained in their machine control unit. Examples include advanced probing cycles, the support for variables within the NC program, and the calling of some sub programs. In most cases the staff will be trained on the advanced features of their control only to find out later that their CAM system cannot take advantage of these capabilities.

The need to replace some older postprocessors is a problem faced mostly by larger companies. These postprocessors are typically mainframe based and were written many years ago. In this age companies find themselves with no one to maintain the old postprocessors and/or they want them to run on local computing platforms. Porting the old postprocessors to run them on modern computing platforms is dreadful task. Thus, usually new postprocessors are written and/or purchased. The difficulty in this is producing the same output as the older posts and being compatible with existing CAM data.

Some other general problems with postprocessors are the followings [6]:

- Absence in CAD/CAM system of the ready postprocessor for specific “machine tool / machine control” combination.
- Too high cost of new postprocessor development.
- 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 and so the technologists are forced to correct wrong NC program file with text processor.
- Each new CAD/CAM system requires an individual postprocessor for the same NC equipment.

5. UTILIZATION OF CAD/CAM SYSTEM FOR NC PROGRAM GENERATING COMPLEX SHAPE PART

The example of hand-made prototype is the screw blade for wind-power plant (Fig. 5). It was made by the hand lamination in to the split form the epoxy resin and glass cloth.

The long-time practices of the producer show the reliability and stability of the screw blade, but the manufacturing of this part with complicated shape was very time consuming and toilsome what decreases the production costs. To obtain the digital data from real existed

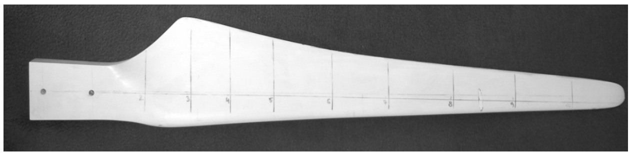

Fig. 5. The hand-made screw blade for wind-power plant.

part is very difficult, especially if the part topology of part is complicated. It can be done by various devices; one of them is 3D scanner.

3D scanners are devices which enable the conversion of real three dimensional objects to the digital versions. The principles of most of them is based on the object shape scanning in the discrete points, what means that the digitizing object is represented in the space as the so called “points cloud”. According to the used technology of the scanning the scanners can be divided on contact and non-contact. The most common scanners are optical, mechanical or magnetically-resonant. The choosing of the scanner type is depended on the precision requests between real and digitized model. The precision at the machining part is request rather high. Other important factors that affect the scanner choosing are the time of scanning and the size of the scanned object. The most quick are the laser scanners.

In Fig.6 is shown one scanned part of the screw blade for wind-power plant.

In this case the problem was about too big dimensions of screw blade for the scanner LPX 250 with the Dr. PICZA software that is on the Faculty of manufacturing technology with a seat in Prešov of Technical University in Košice. It can be needed to cut the real part on 4 parts, to scan individual parts, to process the digital data in PIXFORM software and consequently it was necessary to import the data into CAD/CAM system.

At the creation of 3D model was used various tools and techniques that the CAD/CAM system Pro/Engineer offers, such as the work with the curves and surfaces or quilts. The first was created the solid model of the screw blade, but it was used in praxis as shall. The modelled shell part is shown in Fig. 7. The obtained digital data of the prototype enable to suggest various kinds of model production.

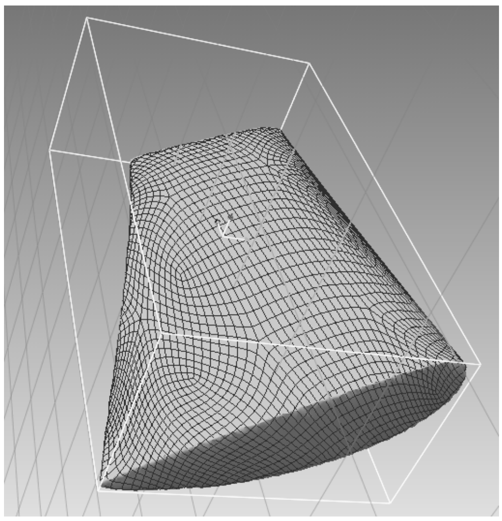

Fig. 6. One scanned part of the screw blade for wind-power plant.

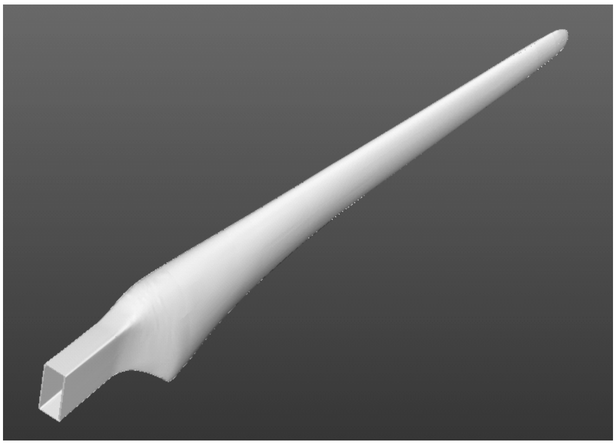

Fig. 7. Created 3D model as shell.

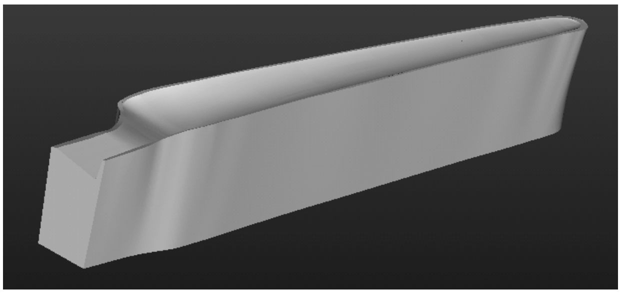

Fig. 8. One part of negative geometry for the form designing.

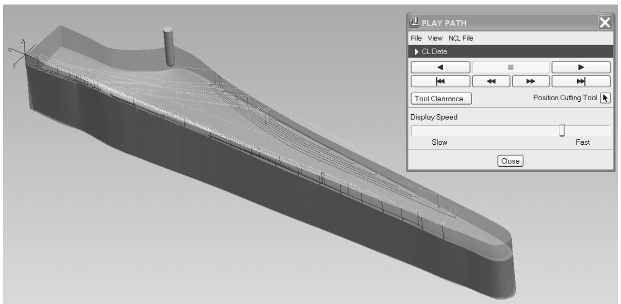

Fig. 9. The simulation of the form manufacturing on the basis CL data.

It was suggested the pressure die casting technology for the manufacturing of screw blade for wind-power plant and so it was needed to generate the negative shape geometry for the case mould preparing (Fig.8).

Consequently, the CAD/CAM system Pro/Engineer enables to simulate the manufacturing process of the form (Fig. 9) and so to detect eventual mistakes and collisions. Geometrical data are automatically loads from CAD module of CAD/CAM system ProEngineer.

On the basis of this simulation of tool path can be generated CL data by CAM module of Pro/Engineer system for the form within the choosing technology [1].

The automation of the manufacturing by means of NC program creation and the complex manufacturing with the utilization of CNC machines is needful for the every plant that wants to be a success with its products on the market today. The generating of CL data as the output of CAM software enables creating to create NC program for selected control system very simply by postprocessor [7].

The manufacturing of new part by means of CNC machine is very quickly and simple today. The basis for computer CL data is 3D model that enables:

- The visualisation of the object.
- The optimization of its structural solution before the production, its quick modification (the dimensions editing), eventually the quick suggestion of the similar objects inside the group technology.
- The utilisation of the object in the assembly allows detecting the conflicts with other components not only in static, but in kinematics state, too, so in marginal constrains of the motion.
- The defining of the couples, loadings, materials and other 3D model properties enables to execute the various types of analysis (structural, thermal, dynamic…) on the object and so predicts the object behaviour in real conditions.
- It is possible to simulate the machining process by means of the created 3D models and so to find out the collisions between the tool and the piece.
- One of the major advantages of 3D model is the possibility to generate CL data and with the utilization of postprocessor to make the NC program for the selected control system in very short time.
- Very simple preparation of the negative shape of 3D geometry for the skillets manufacturing and other.
- 3D model can be analyzed from various approaches in CAD/CAM system (for example static, kinematic, dynamic analysis can be done for the verification of the blade behaviour).

6. CONCLUSIONS AND FUTURE SOLUTION

Up to now, the programming of parts for NC machine tools is normally done by means of standard ISO 6983. This standard dates back to the time of punched cards and does not cover the demands of modern technology. Therefore a new STEP-compliant programming interface that is based on an object-oriented data model (ISO 14649) was developed. This is a radically different approach to CNC programming.

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 postprocessors.

The concept of ISO 14649 is shown in Fig.10 [2].

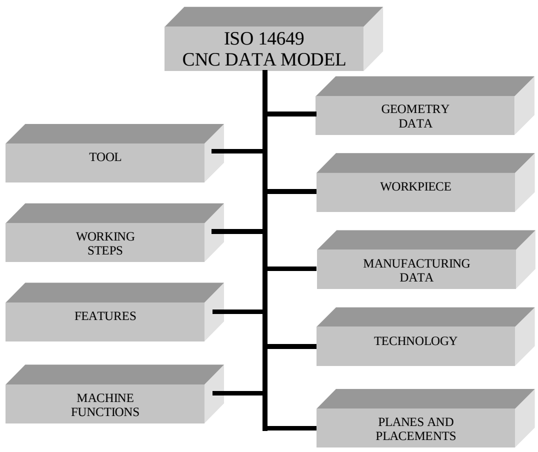

Fig. 10. Comparison of ISO 6983 and ISO 14649.

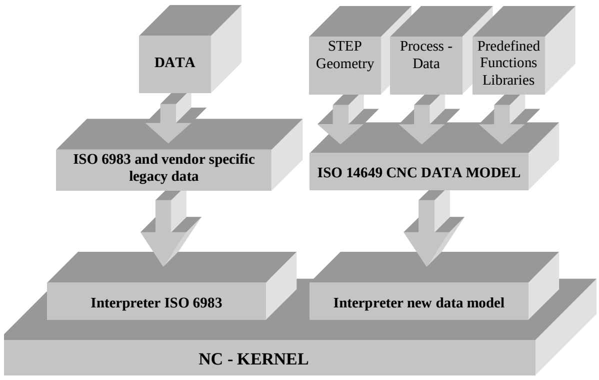

Fig. 11. Data input and data implementation of the new data model.

The company estimates that, by fully implementing STEP NC, machine shops can reduce the time it takes to get jobs onto their machines by 35 percent if they can seamlessly read the 3D product geometry and manufacturing instructions of their customers. Likewise, original equipment manufacturers can reduce the time they spend preparing data for their suppliers by as much as 75 percent if they can seamlessly share the design and manufacturing data in their databases.

STEP Tools also estimates that STEP NC will reduce machining time for small- to mid-sized job lots by as much as 50 percent because STEP NC compliant CNC units will be capable of optimizing speeds and feeds with very little intervention from CNC programmers or machine operators. This factor could make it easier and safer to program high speed and five-axis machines, the company says, making it more likely that they will be used for small- to mid-sized job lots [3].

The new model accepts data from several sources. The data can be generated by CAD/CAM systems, can be part of libraries, can be generated by graphic user interfaces or be input manually. Data generated from CAD/CAM systems or graphical user interfaces is normally processed.

The main characteristic of the new interface presents higher level of information. Whilst a part program according to ISO 6983 describes movements (G1, G2, G3) and switching instructions (M3, M8), the new language covers manufacturing tasks. All operations, which are necessary to produce the finished part from the raw piece, can be described by a sequence of such manufacturing tasks. Since the data model describes tasks, the part program supplies a higher quality of information to the shop floor. Thus modifications at the shop floor can be saved and transferred back to the planning department, which enables a better exchange of experience. Through the fact that geometry of raw piece and finished parts are described by using the STEP syntax, a direct exchange of information between CAD/CAM and NC can be realised. Geometry data can be imported directly from CAD systems. And so, only technology information has to be added in order to generate the part program.

The data model is layer that provides standard interface between the controller interpreter and the different sources of data supplied. The interpreted data supplies the NC-kernel, where axis motions and machine functionality are generated (Fig. 11) [4].

ISO 14649 (CNC data model) is proposed as new data model of transfer between CAD/CAM and CNC machines, but in the future it is expected that it will be able to load the data from various software, which describe the geometric data of the part. So the scan data will not need the data transferring through Pro/Engineer and will be directly load from 3D scanner software to NC machine.

ACKNOWLEDGEMENTS: This article originates with the direct support of Ministry of Education of Slovak republic by grants VEGA num. 1/0885/10 and KEGA num. 270-014TUKE-4/2010.

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