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CAD - CAM DATA TRANSFER AS A PART OF PRODUCT LIFE CYCLE

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Abstract. *Availability and accessibility of proper and correct information is a precondition nowadays for launching a successful product to the world market. Every piece of information will help in creating a new and improved product at any segment of its life cycle. If it is a computer aided manufacturing, then in the phase of data transfer from the geometrical model to the production model, errors should be minimized. A large number of standards, protocols and scripts for the software data exchange confirm how important this area is. On the other hand, the end user is always able to look for better and more favorable solutions, with the least possibility for errors, in our case, in data transfer. Thus all the input requirements of the designer are met, giving the product the desired geometry and quality. That is only a precondition for its further development and implementation in the complex technical systems, so that it would pass into the next phase of its life cycle, which is its actual service and fulfillment of end users' requirements.*

Key words: *CAD CAM Technology, Data Transfer, STEP Standard*

1. INTRODUCTION

In the modern technical product life cycle, the importing and exporting CAD data files is a fundamental function of the CAM software. An understanding of this process can help you find an answer to the questions regarding your expectations and the kind of aid you will get when making a CAM purchase. The CAM software today is development with quality new features. One of your primary concerns when using the CAM software is your ability to import your customers' CAD files so that you can generate tool paths and ultimately cut parts.

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Transferring CAD files software is the norm, rather than the exception, in today's manufacturing world. In the past, the key process was related to blueprints and technical documents. But today you don't want to “reinvent the wheel” by re-creating a paper or lonely electronic model. Beware of poorly developed CAM data translators. Import a file from every customer you work with to determine that there are no data-transferring issues before you cut the salesman a check.

2. CAD CAM DATA TRANSFER

In design and manufacturing, many systems are used to manage technical product data. Each system has its own data formats so that the same information has to be entered multiple times into multiple systems leading to redundancy and errors. The problem is not unique to manufacturing but more acute because design data is complex and 3D leading to increased possible for errors and missing information or lack of understanding between operators. The National Institute of Standards of USA has estimated that data incompatibility reaches a 90 million dollar per year and that is a major problem for manufacturing industry [1].

Data translation is a process in which one data format is converted into another. Data 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 support teams or complete specific functions. One design model may be exported and imported into several CAD packages before the final file reaches manufacturing. Then you're counting on your CAM system's import translator to successfully filter through the result of the previous translations.

A common source of error in part files is receiving incomplete designs due to inexperienced or overworked CAD designers. You may not receive a complete model, or details that the translator is expecting may be omitted or lost. Or your CAM translator may not be fine-tuned enough for the specific CAD file format you are importing. So you will find the translating process offers plenty of room for error and miscommunication.

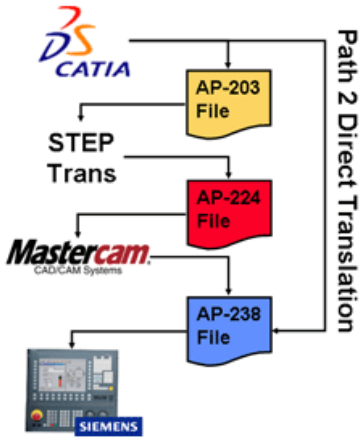


Fig. 1 Data Translations

There are numerous file formats on the market today. You can encounter IGES, CADL, DXF, ASCII, NCAL, PAR, PRT, X_T, MODEL, various European and American automotive formats, and other file extensions. Also, the CAD CAM software manufacturers develop proprietary formats for their software. So one true conversion standard is not yet possible [6,7,8].

Fig. 1 shows data translations - a CAD model of the rough form was first exported from Catia as STEP AP203. This was annotated with STEP manufacturing features to produce AP224 data. Using plugging designed by STEP Tools, the AP224 data was read in to MasterCAM, a workplan with toolpaths was generated from the feature data, and an AP238 file containing all of the geometry, feature, and toolpath data was written out. Last-minute changes to the cutting tools were easily accommodated and demonstrated flexibility [11].

Currently, there are basically two types of file translators–neutral and direct. Neutral translators 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 modeling data [5].

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. It 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 projects must upgrade the importing translator specifically for each new CAD version to eliminate compatibility issues.

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 group of standards such as IGES, requiring CAM systems to still interpret data. Direct translators can exchange data directly from one CAD-CAM system's proprietary format to another. Direct translations generally work better because data transfer errors are virtually eliminated.

Direct translators can exchange data directly into the CAD software's proprietary format. The ability to integrate seamlessly between software packages is the best situation for manufacturing shops because data transfer errors are virtually eliminated. In the end, less time is spent on data transfer issues so you're effectively reducing the product's manufacturing cycle [3].

3. DATA TRANSFER IN CAM

A well-built CAM translator translates all types of files quickly and accurately. You may need to make simple adjustments to your importing translator or to a specific CAD package that ensures optimal conditions for importing files. Most often, problems originate from an incomplete design model that requires correction by a CAM programmer before proper tool path can be generated. Your CAM product should provide you with basic design functionality for correcting defective models.

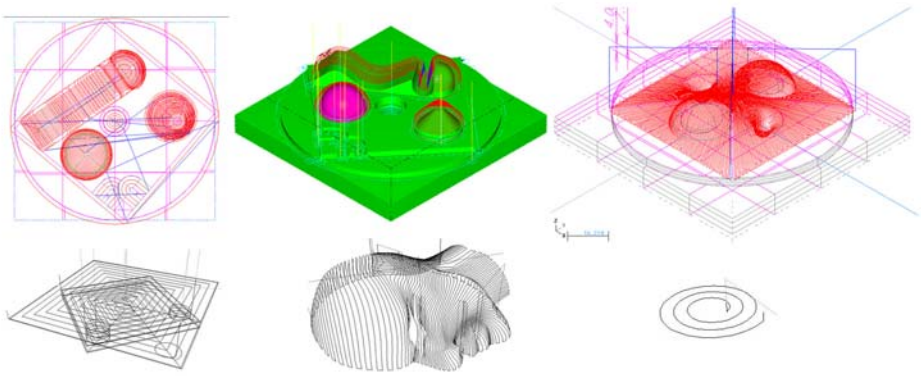


Fig. 2 Spatial Approximation with 3D Lines in the Shape of Surface

You should expect your CAM translator to easily import both solid and surface models. Design analysis and simulation teams use the solid properties and prismatic shapes that can be created in the CAD solid modeler. Usually, these types of part features can be calculated against approximations (Fig. 2). Tool path calculations need a more precise representation of the true shape of the part to avoid part gouging or tool interference. To obtain three-dimensional calculations, files are most often imported into CAM approximate surfaces. Even CAM packages claiming solid modeling capabilities may first translate model files into surface data, then reconstruct the solid, before generating tool path data.

4. CAM TECHNOLOGY IN FUTURE

Leading edge CAM manufacturers are driving the future of CAM software by providing direct "push-button product linking" with CAD manufacturers. Eliminating an extra conversion process of a neutral standard aids the manufacturing shop in obtaining the most seamless method for transferring data. Tool paths can be immediately assigned to the same design model. Usually manufacturing shops run on separate, isolated systems. An ideal solution would provide shops with flexible, interactive methods of data sharing which combining of all product information into a common data format throughout a product's entire life cycle. Data transferred cannot be limited to just geometry, such as curves, surfaces and solids. To manufacture a consistent and tenable, quality product also requires integrated analysis, manufacturing, implementation and testing procedures. Moreover, integrated management systems allude associate standards ISO 9001:2000 for quality management system, ISO 14001:2004 for environment management system and OHSAS 18001:1999 for Occupational Health and Safety Advisory Services.

Fig. 3 DMU 70 eVo linear, Deckel Maho, High-end 5-axis Simultaneous Machining with a Linear Drive

A solution 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 STandard for the Exchange of Product Model Data (STEP). Over the years many solutions have been proposed. The most successful have been standards for data exchange. The first ones were national and focused on geometric data exchange. They included SET in France, VDAFS in Germany and the Initial Graphics Exchange Specification (IGE was started under the International Standards Organization (ISO) to produce one International Standard for all aspects of technical product data. Essentially, STEP is a common structure, operating as a template, for sharing data among multiple users, across all functional areas.

Communication among different departments and different software is a problem. Interaction is important for shortening the product life cycle. STEP will enable us to iterate designs based on manufacturing suggestions, then evaluate and analyze the results before manufacturing the pieces. It also offers a tremendous benefit for exchange and managing information from several engineering and manufacturing disciplines in an effective way.

There are several sections within STEP, called Application Protocols (APs), which are built into a common data model. These APs include definitions not only of typical geometry and drafting elements, but also of data types and processes for specific industries such as automotive, aerospace, shipbuilding, electronics, plant construction and maintenance. In the Europe the most common AP in current use is AP214 (whereas in USA AP203), which is focused on geometric elements for mechanical CAD. AP214 is further subdivided into classes for defining wireframe geometry, surfaces and solid modeling. To facilitate the transfer of information, STEP is built on a data exchange language, called EXPRESS, which formally describes the structure and correct conditions of a file's engineering information.

5. APPLICATION STEP STANDARDS IN CNC TECHNOLOGY

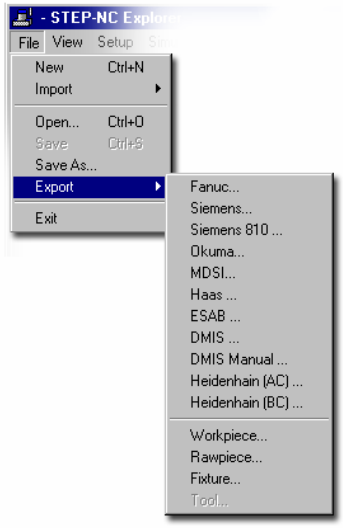


Fig 4 STEP-NC Export Menu

The STEP-NC AP238 standard is the result of a ten year international effort to replace the RS274D (ISO 6983) M and G code standard with a modern associative language that connects the CAD design data used to determine the machining requirements for an operation with the CAM process data that solves those requirements. STEP-NC builds on the previous ten year effort to develop STEP data standards for CAD data and uses the modern geometric constructs in STEP to define device independent tool paths, and CAM independent volume removal features.

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 technology characteristics such as whether this is a roughing or finishing operation, step, radial per minute and similarly. 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.

A key feature of STEP-NC AP-238 programs is that they are machine and technology method 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 two significant consequences for industry.

- If parts can always be rapidly manufactured from an AP-238 description, then there is no longer a requirement to keep copies of those parts in the inventory. A recent study for the UK Navy estimates that at least $4M can be saved for one depot (and up to $640M can be saved for the entire UK Navy) if the depot can store its spare parts as electronic product data instead of as physical items.

- If parts can be made independently of the axis codes, then the same CNC program can be run on many machines. This allows a part program to be made once and run anywhere. Another study has shown that a mid-sized machine shop could save as much as $0.5M per year in reduced CAM costs, less waste, and greater output if it received reliable machine independent CNC data from its customers.

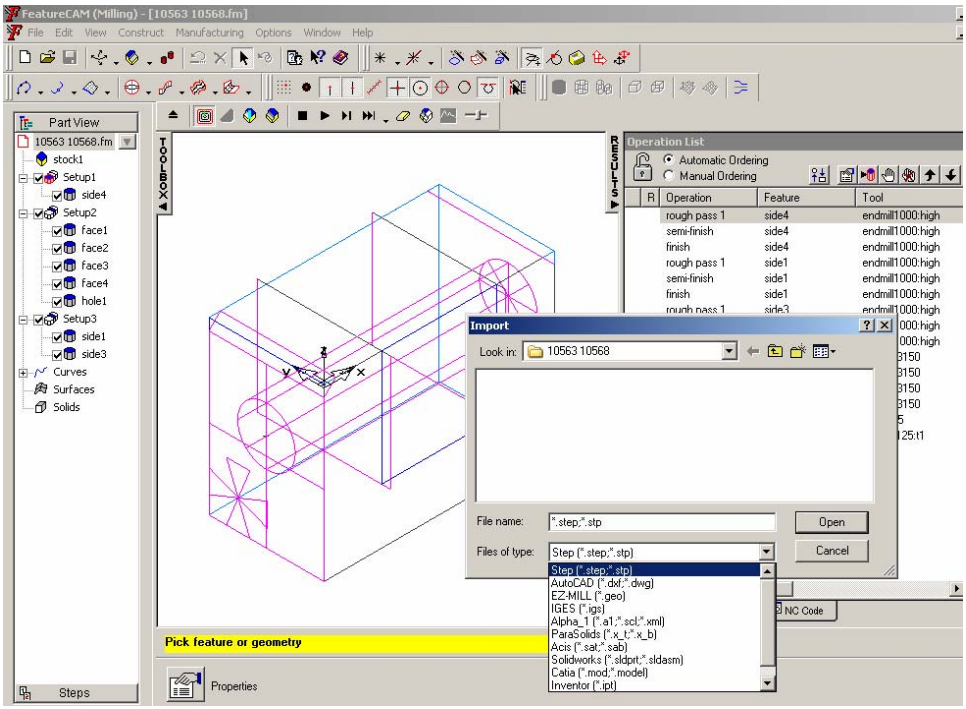


Fig 5 Software Simulation of Milling Process

Fig. 6 shows how design data is communicated to manufacturing on the process model 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

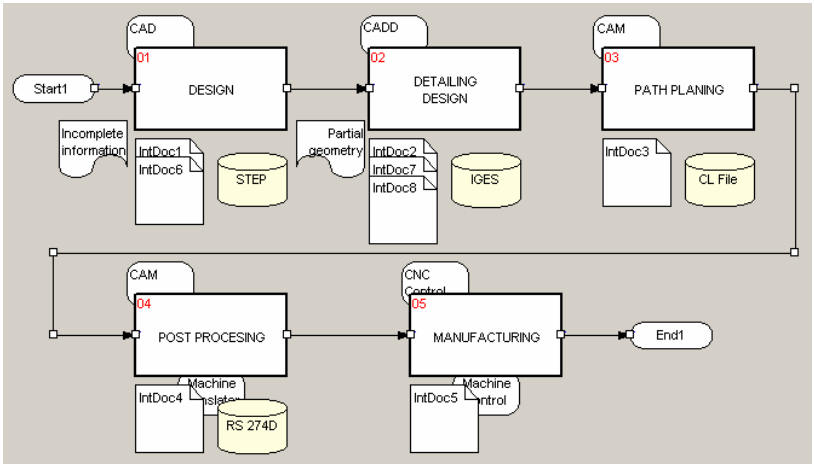


Fig. 6 CNC Process, ©Visual Processes, CIM College

performed using a CAD (Computer Aided Design) system, the job of detailing is performed using a drawing CADD (Computer-aided Design and 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.

Despite the success that the STEP is having, certain questions remain the users' minds concerning the speed of its development and deployment [5]. Many critics point out correctly that the XML standards for e-commerce are being developed much more quickly.

Fundamentally, the product model data is different to other kinds of e-commerce data such as invoices, receipts, etc. The traditional method for communicating product model information is to make a drawing and the traditional method to communicate an invoice is to make a form. When you make a drawing or 3D model you need to define information with many subtle and complex relationships and this makes the STEP data exchange problem more difficult.

An XML data format is being developed for STEP but the STEP architecture requires the information requirements of an Application Protocol to be mapped into the common set of Integrated Resources. This allows all of the protocols to share the same information and is essential if all of the interfaces are going to share and reuse the same set of data (Fig. 7). However, the sharing necessarily divides the original data into multiple entities that are not so easy to understand in XML or any other format. This is disappointing because one of the attractions of XML is that it is self-documenting (at least for programmers and domain experts).

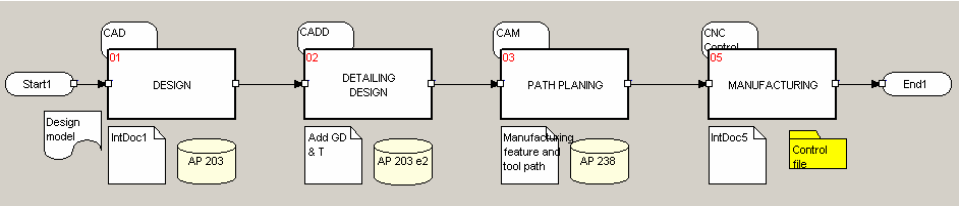
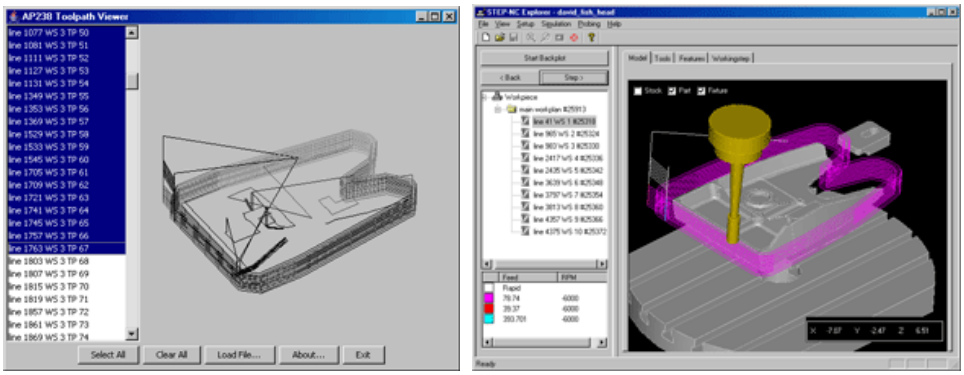


Fig. 7 CNC Process, ©Visual Processes, CIM College

Therefore, a new level of documentation is required in the STEP data to show how the information requirements have been mapped. The required structures are currently being developed and it is anticipated that the STEP will have a self-documenting XML format in the very near future [5].

The "view238" application is a simple tool for graphically displaying STEP-NC toolpaths (Fig. 8). It is a Java application that reads AP-238 XML data using the DOM interface, then finds and display the toolpaths within.



a) Java AP-238 Toolpath Viewer b) STEP-NC Explorer

Fig. 8 Representation of the Same Milling Paths

6. CONCLUSION

The real issue that stops faster STEP deployment is the commitment of those with the resources necessary to define the standards. The government does not like to pick solutions for industry, and industry does not like to fund the development of solutions that can also be used by their competitors. Consequently, much work only gets funded in situations of clear and desperate need such as when the high cost of manufacturing is causing excessive job losses.

The Internet and the World Wide Web broke through this cycle when small applications made the benefits of the new infrastructure clear and compelling for all users. AP-203 made STEP useful by allowing solid models to be exchanged between design systems. AP-238 will make STEP compelling for some users by allowing them to machine parts more efficiently. However, like the early Internet there will be alternatives that are considered more reliable by other users. The killer application that makes STEP ubiquitous has yet to be identified.

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CAD - CAM PRENOS PODATAKA KAO DEO ŽIVOTNOG VEKA PROIZVODA

Saša Ranđelović, Saša Živanović

Dostupnost i raspoloživost pravim i tačnim informacijama danas predstavlja preduslov za lansiranje uspešnog proizvoda na svetsko tržište. Svaka od tih informacija doprineće kreiranju novog i poboljšanog proizvoda u bilo kom segmentu njegovog životnog veka. Ukoliko se radi o računarski podržanoj proizvodnji onda u fazi prenosa podataka sa geometrijskog modela na model izrade prilike za grešku treba svesti na minimum. Veliki broj standarda, protokola i zapisa za međusobnu softversku razmenu podataka danas samo potvrđuje kolika se pažnja poklanja ovoj oblasti. S druge strane, krajnji korisnik je uvek u prilici da traži bolja i povoljnija rešenja, sa što manje prilika za grešku, za prenos podataka u konkretnom slučaju. Time svi ulazni zahtevi projektanta bivaju ispunjeni čime proizvod dobija željenu geometriju i kvalitet. To je samo preduslov za njegovu dalju nadogradnju i implementaciju u složene tehničke sisteme da bi nakon toga prešao u narednu fazu svog životnog veka, odnosno eksploataciju i ispunjavnje zahteva kranjeg korisnika.

Ključne reči: *CAD CAM tehnologije, prenos podataka, STEP standard*