Computer Aided Process Planning Using Neutral File Step for Rotational Parts

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***Abstract*— Present investigation is on process planning using neutral file with format STEP for rotational parts with use of computer. CAE systems involved in every stage of product life cycle mainly uses the product data produced by the CAD systems and integrated manufacturing data produced by CAPP and CAM systems. As the degree of automation and CAD/CAM integration increases, the inclusion of high level information with the product data and its seamless flow in CAD-CAM-CNC chain becomes a necessity. The objective of this work is to develop a Computer Aided Process Planning system for rotational parts using ISO 10303 standard STEP AP224 data exchange file, enabling the inclusion of high level information about the product besides geometry. The developed system aims to incorporate the small and medium sized manufacturing enterprises into the e-manufacturing chain by adopting the NC-code based CNC machine tools without any modification of the controllers.**

***Keywords*— CAPP, CAM, CAE, STEP, ISO, NC Code, Automation**

I. INTRODUCTION

Design and manufacturing are two most important but complementary issues of production, from the engineering point of view. The product design is not just based on good design but it should be producible and include the representation of product data throughout the life-cycle of a product. However, in traditional understanding of production, design and manufacturing are regarded as separate phases and most of the time managed by two different engineering groups. This may lead scrapped parts, need for retooling and totally an increase in cost, due to;

- The unawareness of the design group to anticipate the manufacturing implications of the design,
- The design data package does not reflect the intent of the design group,
- The way of passing the design data to manufacturing group,
- The misinterpretation of design data by manufacturing group.

Generally, obtaining an acceptable part is possible after a few passes back and forth between the two groups and furthermore after a few revisions done by the design group. With improved computer technologies and better understanding of its usages for design and manufacturing, above mentioned methods are largely being replaced by Computer Aided Design (CAD) and Computer Aided Manufacturing (CAM) systems and various databases (Mantyla, Nau, Shah, 1996).

CAD systems are used for engineering of 3D models and/or 2D drawings of physical components since they are interactive, visual, user friendly and powerful. This leads an increase in designer's productivity, quality of design while decreasing total cost of production by creating a database for manufacturing. An advantage of this is the availability of the model data for the direct or indirect utilization in manufacturing processes. Nevertheless, CAD and CAM modules or applications have their own data representation, namely CAD language is different from CAM or CAPP (Computer Aided Process Planning) common language. Besides this, interpreting the CAM common language from the CAD language is very hard for computers.

1.1 Integration of CAD and CAM

Introduction and usage of Computer Aided Engineering (CAE) systems in design and manufacturing increased the engineers' productivity in both design and manufacturing; however, most of the problems regarding the loss of design intent, misinterpretation of design still remain. Moreover in some cases management of the digital data between two distinct CAE systems is a different matter.

To overcome these problems, Computer Integrated Manufacturing (CIM) was introduced to further increase the degree of automation in production processes beyond CAD/CAM systems (Groover, 1987). CIM aims the complete automation of a manufacturing plant with its all processes functioning under computer control but still the CAD/CAM is in the hearth since the CAD and CAM systems are essential to reducing cycle times in the organization. So the main concern of CIM is to enable efficient communication and data exchange between CAD/CAM systems for the life cycle of a product.

Together with the idea of complete automation idea, rapid advancement of information technology associated with

manufacturing technology and advanced internet technology introduced a new paradigm of e-manufacturing. By the introduction of e-manufacturing paradigm, realization of design anywhere- build-anywhere (DABA) system, announced to reduce production cost and the lead time while increasing quality, became possible. The key to the success of the e-manufacturing paradigm is a seamless data flow in the CAD-CAM-CNC chain. To achieve this, STEP-NC is developed to be used as the interface between CAM and CNC to the NC-code language currently employed by CNC machine tools. The advantage of STEP-NC is including rich information set of 'what-to-make' (geometry) and 'how-to-make' (process plan) using the power of international STEP definitions (Suh, Chung, Lee, Shin, Choi, Kim, 2006; Shin, Suh, Stroud, 2007). However, a drawback of STEP-NC arises as cost due to the fundamental change in CNC controllers required to adapt new standard.

1.2 Part Representation Scheme

Digitalization of the product data with the introduction of CAE systems brought the need of representations and exchange of product data. Over the years, the increasing popularity and variety of CAE systems, lead to the introduction of different product data models such as IGES, SET, DXF, etc. These product data models are focused on the geometry solely and most of them achieved this using different approaches and output formats.

The integration of CAD/CAM systems, complete automation of a manufacturing plant and a seamless data flow in CAD-CAM-CNC all requires a standardized product data representation; comprehensible, usable, editable by each individual system. Representation of product data is standardized by ISO 10303 (STEP: Standard for the Exchange of the Product data) and the concept carried beyond the geometry exchange, with the inclusion of application specific high level information required for downstream applications.

1.3 Feature Based Approach

In the early eighties, it was widely considered that the newly emerging geometric modelling techniques, that the CAD systems use, would provide the necessary complete and unambiguous part descriptions (Voelcker, 1992). However, with the rapid evolution of CAD-CAM-CNC chain and with the introduction of concurrent engineering concepts, geometric models need to be enhanced to adapt to product life-cycle. This need introduced new concepts such as parametric modelling, feature based modelling and feature recognition for the sake of integrating design and manufacturing. In feature based modelling, feature based design and feature recognition, product models are represented using generic shapes with which certain properties or attributes and knowledge useful in reasoning about the product (Fidan, 2004).

II. STEP AND ITS APPLICATIONS

2.1 Introduction to STEP

STEP, stands for Standard for the Exchange of Product Model Data, and is officially titled ISO 10303. The main aim of STEP is to provide a representation of product information along with the mechanisms and definitions to enable product data to be exchanged. The exchange is among different computer systems and environments associated with the complete product lifecycle along with design and manufacture. During the process information generated related with product is used for many computer systems, including some that may be located in different organizations. The organizations must be able to represent their product information in a common computer-interpretable form that is required to remain complete and consistent when exchanged among different computer systems. [2]

In STEP the parts are organized in a series, and each published separately. These parts falls under one of the following series: description methods, integrated resources, application protocols (APs), abstract test suites, implementation methods, and conformance testing. STEP uses a forma language, EXPRESS [3], to specify the product information to be represented. The precision and consistency of representation is enabled by the use of formal language and facilitates development of implementations. To transfer this information, STEP usually employs the neutral file approach. Transfer of data from individual application to other is usually a two-step process requiring a post-processing and pre-processing.

The main objective of STEP is to provide a mechanism that is capable of describing product data throughout the life cycle of a product, independent from any particular system. The nature of this description makes it suitable not only for exchange of neutral file, but also useful for implementing and sharing product databases and archiving. The ultimate goal is an integrated product detailed information database that is accessible and useful to all the resources necessary to support a product over its lifecycle.

2.2 STEP Application Protocols

Product information for one or more applications can represented by the protocols The APs define the scope, the information exchanged, by means of testing and a user guide for the application. The STEP Application Protocol of interest for this research is Application Protocol 224. It contains all the information needed to manufacture the required part, including:

1. Geometrical and topological entities required to represent manufacturing features along with implicitly boundary representation format.
2. Explicit representation of manufacturing features.
3. It is necessary to identify the dimensional and geometrical tolerances of the manufacturing features.
4. Information necessary to define material, hardness, surface finish and other technological data.

III. OBJECTIVES

CAE systems involved in every stage of product life cycle mainly uses the product data produced by the CAD systems and integrated manufacturing data produced by CAPP and CAM systems. As the degree of automation and CAD/CAM integration increases, the inclusion of high level information with the product data and its seamless flow in CAD-CAM-CNC chain becomes a necessity.

The objective of this work is to develop a Computer Aided Process Planning system for rotational parts using ISO 10303 standard STEP AP224 data exchange file, enabling the inclusion of high level information about the product besides geometry. The developed system aims to incorporate the small and medium sized manufacturing enterprises into the e-manufacturing chain by adopting the NC-code based CNC machine tools without any modification of the controllers.

IV. LITERATURE SURVEY

A significant number of research papers are reviewed related to, integration of CAD, CAM and CNC through CAPP and also about Product Data Management (PDM), Product Life cycle Management (PLM), Product Data Exchange between different CAD systems, Neutral file concept, ISO standard 10303 Standard for Exchange of Product model data (STEP) and specifically STEP Application Protocol (AP) – 224, which is designed for computer aided process planning system (CAPP). With the above literature survey it is found that, there are issues related to the seamless integration of CAD/CAPP/CAM and CNC. Most of the integration softwares are designed on the proprietary data formats. There is a need to develop software system which should be independent of any proprietary data format.

V. METHODOLOGY

As mentioned above, most of the CAD systems available in market today have built in facility of representing designed part model in STEP AP 224. Using any of these packages the STEP AP 224 representation of CAD model can be obtained. This neutral file is used to integrate the CAD, CAM and CNC systems by developing a CAPP system. For this first the CAD model of the designed part is represented in neutral file format and then process it to make the CAPP system accept this neutral format for the generation of optimal process plan for the given part and finally the output of CAPP system is transferred to a standard CAM system and finally for automatic generation of CNC codes.

The neutral file defines the CAD data of a designed part in terms of low primitive geometric entities such as vertices, lines, curves and surfaces which cannot be used directly for the automation of process planning activities. It is necessary to process the neutral file and reorganize the data to convert it into manufacturing features. It is because, for a product, the manufacturing features are the volumes which are removed by one or a series of operations and it is possible to associate manufacturing information with them. Therefore, features are considered the communication medium between design and manufacture and the automation of process planning begins with the implementation of feature recognition procedures. Accordingly feature recognition is considered as the first and most important activity in the development of a CAPP system which tends to the automation of subsequent manufacturing activities and also shop floor activities such as scheduling.

VI. POSSIBLE OUTCOME

The outcome of the research is the integration of CAD, CAM and CNC through developed CAPP system using STEP AP-224, by seamless data flow from CAD to CNC machines for manufacture of Rotational Parts. This will be achieved by developing software which takes ISO 10303 AP-224 neutral file as its input which is independent of any proprietary data format.

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