DESIGN OF ROTATIONAL PARTS USING STEP AP224 FEATURES WITH AUTOMATIC NC-CODE GENERATION

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ABSTRACT

DESIGN OF ROTATIONAL PARTS USING STEP AP224 FEATURES WITH AUTOMATIC NC-CODE GENERATION

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The rapid advancement of information technology and its integration with the manufacturing technology increased the necessity of consistent and coherent data flow in the chain of Computer Aided Design (CAD)-Computer aided Manufacturing (CAM)-Computer Numerical Control (CNC). To achieve this, ISO 10303 standard (STEP), developed by ISO, is seen as a solution since STEP is independent of the environment on which design data, manufacturing data or machining data produced. In this thesis, efficiency of NC-code generation, with the inclusion of process planning data, from a STEP based CAD data is investigated. For the investigation purposes, software responsible for both building the STEP based CAD data and generating related NC-code automatically is developed. Using this software, several parts are designed; generated NC-codes are verified via CNC simulators and some test parts are produced. STEP AP224 based feature modeler, developed specifically for 2-axis rotational part design, includes; feature library, feature modeler employing SW2007 via API for visualization and preprocessor responsible for generation of STEP file in neutral format, called STEP Part 21. The NC-code generator includes; postprocessor responsible for STEP Part 21 interpretation, CNC machine tool and cutting tool database and preprocessor responsible for NC-code generation.

Keywords: STEP, AP224, Feature Based Modeling, Rotational Parts, NC-Code

ÖZ

STEP 224 UNSURLARI KULLANARAK DÖNEL PARÇA TASARIMI VE OTOMATIK NC-KOD OLUŞTURULMASI

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Bilgi teknolojilerindeki hızlı ilerleme ve üretim teknolojileri ile bütünleştirilmesi, Bilgisayar Destekli Tasarım (BDT)-Bilgisayar Destekli İmalat (BDİ)- Bilgisayar Nümerik Kontrol (CNC) zincirindeki veri akışının uyum ve tutarlılık içerisinde olma gerekliliğini artırmıştır. Bunu gerçekleştirmek için, ISO tarafından geliştirilen ISO 10303 standardı (STEP), tasarım, üretim ve talaşlı imalat verilerinin oluşturulduğu ortamdan bağımsız olmasından dolayı, bir çözüm olarak görülmektedir. Bu tezde, STEP'e dayalı BDT verisinden NC-kodu oluşturulmasının verimliliği, süreç planlama verileri de eklenerek incelenecektir. Bu inceleme kapsamında, STEP'e dayalı BDT verisini oluşturarak, ilgili NC-kodunu otomatik olarak derleyecek bir bilgisayar programı hazırlanmıştır. Bu program kullanılarak çok sayıda parça tasarlanmış; oluşturulan NC-kodlar, CNC simülasyon programları ile doğrulanmış ve bazı test parçaları üretilmiştir. Özellikle 2 eksenli dönel parça tasarımı için geliştirilen STEP AP224 tabanlı modelleyici bünyesinde; unsur kitaplığı, API aracılığıyla SW2007 kullanan unsur modelleyici ve STEP Part 21 olarak adlandırılan nötr formattaki STEP dosyasını oluşturmakla görevli ön işlemci bulunmaktadır. NC-kodu üretici bünyesinde ise; STEP Part 21 algılamakla görevli son işlemci, CNC tezgah ve kesici takım bilgi bankası ve NC-kodu oluşturulmasından sorumlu bir ön işlemci bulunmaktadır.

Anahtar Kelimeler: STEP, AP224, Unsur Tabanlı Modelleme, Dönel Parçalar, NC-kodu

To the two precious women in my life…
To my mom and my wife.

And to the loving memory of my dad.

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CHAPTER 1

INTRODUCTION

1.1 Design and Manufacturing

Design and manufacturing are two 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, the methods mentioned above 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.

1.2 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.

1.3 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: **ST**andard for the **E**xchange of the **P**roduct data) and the concept carried beyond the geometry exchange, with the inclusion of application specific high level information required for downstream applications.

1.4 Feature Based Approach

In the early eighties, it was widely considered that the newly emerging geometric modeling 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 modeling, feature based modeling and feature recognition for the sake of integrating design and manufacturing.

In feature based modeling, feature based design and feature recognition, product models are represented using generic shapes with which engineers associate certain properties or attributes and knowledge useful in reasoning about the product (Fidan, 2004).

1.5 Objective of the Thesis

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 system for rotational part design using STEP AP224 features, enabling the inclusion of high level information about the product besides geometry, and automatic generation of NC-code using this data in neutral format. 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.

1.6 Scope of the Thesis

The scope of this work is to develop a system to design rotational parts for NC-code generation using AP224 features of STEP. The system development includes two main phases with their sub steps: (1) feature based modeler development phase, (2) NC-code generator development phase.

<u>Feature based modeler development phase</u> includes; building a feature library based on AP224 but refined for rotational part design, construction of the modeler using the geometric identities of the refined features, composing a post-processor to pack and store the generated design data in the neutral formatted STEP file.

NC-code generator development phase includes; composing a pre-processor to import the design data to work on, building libraries for machine tools, cutting tools and inserts, constructing NC-code generator algorithm and composing a post-processor to create the machining file.

A feature library is developed using STEP AP224 and refining the machining features defined in AP224 for rotational features those can be manufactured by turning process. The feature library includes definitions, classifications, attributes, generation techniques and parameters, in every detail. Since EXPRESS is the formal specification language of STEP and is an object-oriented data descriptive language, which classifies and constructs product data in terms of entities, and enables precision and consistency of product data representation and facilitates implementation (Amaitik, Kılıç, 2002), the feature library of the system is implemented as an object-oriented data type library. It is created as a "dynamic link library" and compiled by using Visual Basic 6.0.

A feature based rotational part modeler is developed using Visual Basic 6.0 and SolidWorks application inside the Visual Basic to enable visualization of model creation. So the developed feature modeler serves for designing rotational parts easily in a user friendly visual environment, by just picking from the pre-defined application specific features. Also the design data is generated with the inclusion of high level information for manufacturing, thus prevents any manufacturability problem arising from design process.

Post-processor and pre-processor are composed specifically for the system and enables bi-directional transfer of product model data. The output of the rotational part modeler is a neutral STEP Part 21 file, so this output file can be imported to another downstream application or directly to the NC-code generator module of the system.

An NC-code generator is developed to accept feature based design data in neutral STEP Part 21 format. The machine tools library, cutting tools and inserts library are formed to enhance the capabilities of NC-code generator. The machining file generated by the NC-code generator module can be directly used for simulation or machining purposes.

1.7 Outline of the Thesis

In Chapter 2, a review of literature related to product data representation approaches, product data models and automated turning technology using NC-codes is presented.

In Chapter 3, framework of the developed system, feature library, the feature modeler and NC-code generator with its tool and insert libraries is described in detail.

In Chapter 4, the developed software is introduced thoroughly with its running procedures, embedded modules, feature creation forms, tools and inserts library, and capabilities and limitations.

In Chapter 5, sample design and NC-code generation applications executed by the developed system and outputs are presented, to demonstrate the performance of the developed system.

In Chapter 6, concluding remarks and possible future work plans recommended to complete the integration process are given.

In Appendix A, sample source codes are given, describing different steps of the developed software.

In Appendix B, NC-code outputs for the sample parts demonstrated in Chapter 5 are given for the sake of completeness.

In Appendix C, sample source codes are given, describing different steps of the developed software.

In Appendix D, feature library refined from AP224 and used for the basis of system development are presented with their geometry creation parameters.

In Appendix E, cutting tools and inserts library used for NC-code generation are presented with their usage and cutting parameters.

CHAPTER 2

LITERATURE REVIEW

2.1 Product Design Representation

CAD systems use different techniques to visualize the design data. Among these, two main approaches for the representation of the product design are: geometric modeling and feature modeling.

2.1.1 Geometric Modeling

Geometric models are classified as 2D or 3D models. 2D geometric models are wireframe models and 3D geometric models are classified as wire frame, surface or solid models (Gu, Norrie, 1995).

2.1.1.1 Wireframe Models

The wireframe model is perhaps the oldest way of representing solids, dating back to 1960. In wireframe modeling, edge lines or arcs and the corner points of the physical object are used for visualization; indeed, there are no surfaces displayed on the screen. Although wireframe models do not look like solid objects, they do contain an accurate geometric description of the object being modeled (Fidan, 2004). But, visualization of complex geometries may be ambiguous with wireframe models due to the lacking surface information.

Wireframe models were very useful and popular with the earliest computer processors since the number of calculations required is relatively low and rendering is relatively simple. Undoubtedly, for today’s design workstations, visualization and rendering of wireframe models are simple tasks. But, wireframe models are still preferred in cases where a high screen frame rate is needed (for instance, when working with a particularly complex 3D model, or in real-time systems that model exterior phenomena).

2.1.1.2 Surface Models

Surface models are introduced in the early 1960's as an evolution of wireframe models with the inclusion of surface information. Visualization of a 3D object model became unambiguous (Figure 2.1). In a surface model, determining whether or not a point is on the surface is possible unless there is an object represented by several surfaces. In such a case some additional information is required. Vertices, edges and faces are used for visualization of a surface model on the computer screen.

Surface models are useless where representing or analyzing a product's internal structure is required since the interior of the object cannot be represented and visualized as a solid. On the other hand, this is an advantage for surface models since the construction is so easy by creating plane surfaces, by sweeping, revolving, or extruding entities. Surface models are also useful for finding the intersection of surfaces in space and creating models for shaded rendering (van Leeuwen, Wagter, Oxman, 1995).

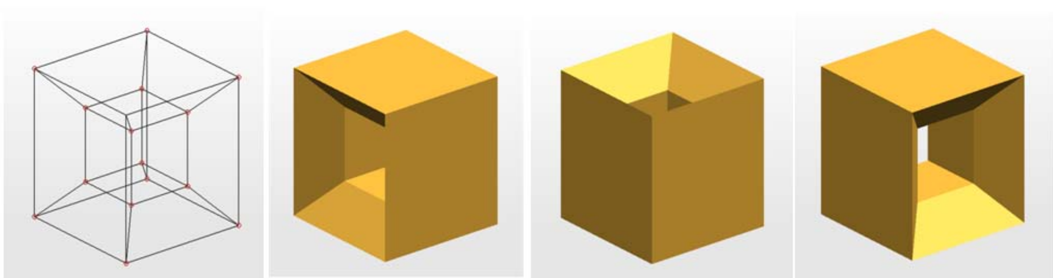


Figure 2.1. Ambiguity in wireframe models eliminated by adding face information.

2.1.1.3 Solid Models

Solid models, which represent the 3D object in complete visualization and unambiguous description, are introduced in the early 1970's. Different from wireframe and surface modeling, in solid modeling mathematically predefined solid primitives, such as blocks, cylinders, cones, wedges, spheres, etc are used instead of lines, arcs, and surfaces. Limited number of predefined primitives of CAD packages can only be limited by the creativity of the designer, seeing that simple primitives can be used to create complex shapes by the help of Boolean operations: *union* (the sum of two primitives), *intersection* (the common mass shared by two primitives), and *difference* (subtracts a primitives from another). The reason why solid models are regarded as the most preferred and indispensable type of model for designing, analyzing, and manufacturing products is the information about the closure and connectivity of shapes are superior to wireframe and surface models. Moreover, solid models can offer information about mass properties such as weight and center of gravity unlike wireframe and solid models.

Different ways of organizing the same geometric and topological data in the form of a data structure, namely representation schemes are developed and used in the solid modeling. Constructive solid geometry (CSG) and boundary representation (B-rep) are the commonly employed schemes. Others can be listed as; parameterized primitive instancing, spatial occupancy enumeration, cell decomposition, sweeping, etc. Modern modeling software may use a combination of these schemes to represent a solid.

(1) Missing information concerning material properties, dimensional and geometric tolerances, surface finish, etc. and (2) the data form, preventing extraction of engineering intent and feature recognition, can be listed as the deficiencies of solid models.

2.1.1.4 Hybrid Models

A hybrid model combines surface model and solid model inside. The advantage of the hybrid model is that, the surfaces and solid faces are able to interact with each other and permit constraints and relationships between the two different entity types. For instance, a solid body can be trimmed using a surface.

2.1.2 Feature Modeling

Today's most CAD systems utilize the previously discussed geometric modeling techniques providing an incomplete definition of the product. Despite their power of representation of product model, due to the lacking high level information they cannot be used alone for downstream applications such as process planning, manufacturing, CNC programming, inspection, etc (Shah, Rogers, 1988). Always there is a need for another system to translate the implicit geometric data into explicit process planning and manufacturing information adding the design intent of the product.

In order to eliminate this need and be able to supply geometric tools with a higher level of information, feature modeling was introduced in 1980's. Lacking information elements in geometric modeling were introduced and named as "features" to create a link between geometric models and downstream applications. This means; features make possible understanding the design intent and manufacturing information besides geometric information by providing engineering attributes such as materials, dimensions, tolerances, surface finish, etc. So the definition given by Shah (1991) can be accepted as a common feature definition: "Features are generic shapes with which engineers associate certain properties or attributes and knowledge useful in reasoning about the product".

2.1.2.1 Classification of Features

Although many attempts to classify features, there is not any commonly accepted one. But the common types for almost each different classification are given below:

- Form features: elements of nominal geometry
- Pattern features: regular pattern of similar entities
- Tolerance features: deviation from nominal form or size or location

- Assembly features: grouping of various feature types to define assembly relations, such as mating conditions, part relative position and orientation, etc.
- Material features: material composition, treatment, condition, etc.
- Compound features: two spate features formed together to form another feature. Such as a counterbore hole is formed using a blind hole and a through hole of smaller size.

2.1.2.2 Feature Mapping

One complexity of dealing with features is that how a part is viewed in terms of features varying from one application to another. For instance, features are viewed as geometry constructs of a product from design aspect while process planning views features as volumes to be removed by machining operations. Therefore, the same part could be viewed as having different features, depending upon the engineering task being undertaken. This implies that feature data need to be transformed from one viewpoint to another when exchanging product data between two dissimilar applications, such as design and manufacturing. This transformation process is referred to as *feature mapping or feature transformation* (Amaitik, 2005; Shah, Mantyla, 1995).

The major difference between feature recognition and feature mapping is in the initial state of the model from which features are derived. In feature recognition, application-specific features must be extracted directly from a geometric model. In feature mapping, application-specific features are recognized from another set of features. In theory, at least, it can be said that feature mapping takes place at a higher level because it can take advantage of features that already exist in the original feature model (Amaitik, 2005).

2.1.2.3 Feature Data Exchange

Output of feature based systems, feature based product data, is the input by downstream applications and may also be used by other systems. This brings the necessity of exchangeability of features between the various systems and applications. Hard work of several years for developing product data exchange format within the STEP program included the feature data exchange and will be discussed in more detail in the next chapter. The feature data exchange standards are likely to have a significant impact on CAD/CAM systems (Shah, Mantyla, 1995).

2.1.3 Feature Creation Techniques

There are several techniques used to create features, either directly or indirectly. The direct creation case named as "design by features" is using the predefined features in design stage and creating geometry from the feature definitions. The indirect case is "feature form geometry" where features are derived from the geometry of a product created by CAD software. "Feature recognition" is another name given to this technique and can be done by a human assist or by a computer algorithm.

2.1.3.1 Feature Recognition

Feature recognition is the process of investigating the topology and geometry of a part, comparing with the definitions among the feature library and matching with the most suitable feature. This method is used for generating feature based models form the geometric models created by CAD systems and to make them usable for downstream applications. But it has some disadvantages. Challenging feature recognition algorithms need to be developed and it is a complex and time consuming process, tough further refinements of the recognition algorithms is necessary. In case of feature interactions, incremental feature validation is required that is the partially recognized part has to be checked whether any features are interacting or not. Furthermore, feature model conversion has to be done to convert the recognized form features into manufacturing features in order to provide them as useful to downstream applications (Fidan, 2004; Han, 1996).

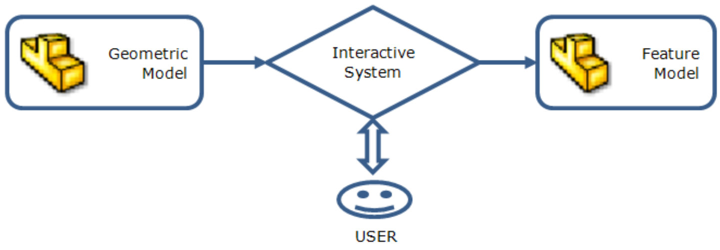


Figure 2.2. Interactive feature recognition

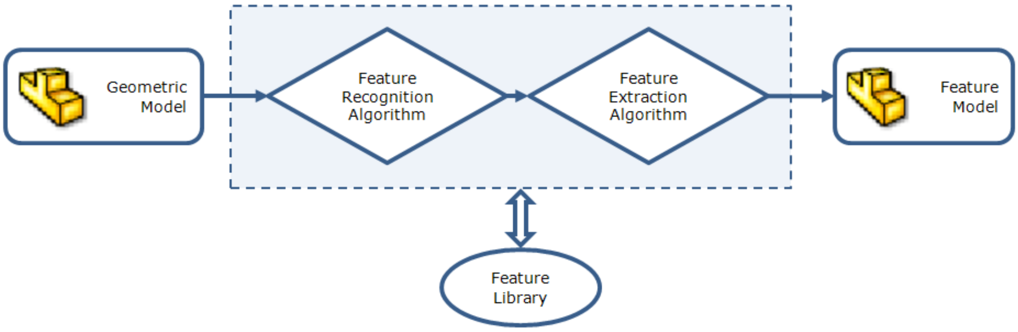


Figure 2.3. Automatic feature recognition

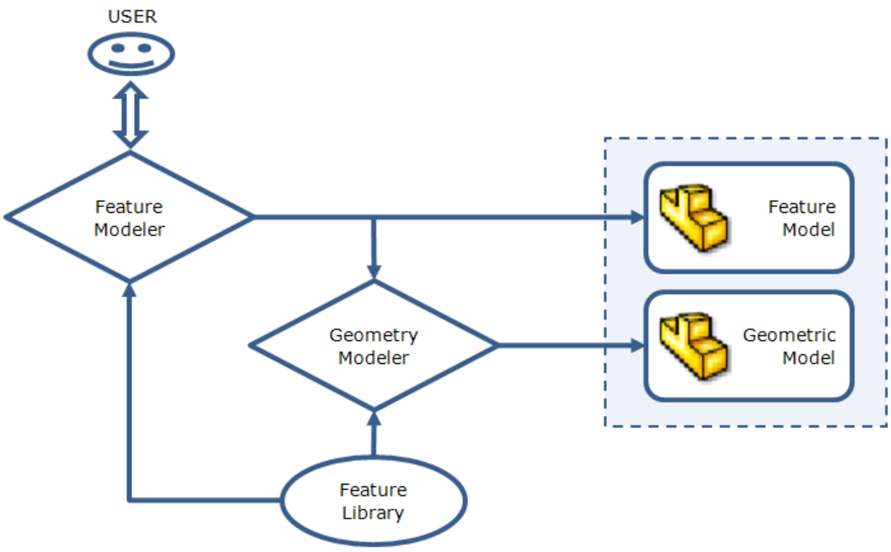


Figure 2.4. Design by features

Feature recognition process can be done manually by selecting proper features for the downstream application using an interactive system, or done automatically by an algorithm developed to pick proper features for the downstream applications from a feature library of pre-defined generic features (Figure 2.2 and Figure 2.3).

2.1.3.2 Design by Features

As the process named product data and its geometric model is created by selecting among predefined features named "feature library". This gives the opportunity of adding the functional design intent and explicit manufacturing information into the product data as well as the geometric modes. This property of design by features makes the high-level communication between design and manufacturing possible without any additional process after the design phase. If there is a complete feature library for product definition the only limit is the creativity of the designer and competence of the feature based design software in using the pre-defined generic features (Amaitik, Kılıç, 2005) (Figure 2.4).

There have been many attempts to define a comprehensive feature library to be able to define all mechanical parts. In the research of Sheu (1998), computer integrated manufacturing system for rotational parts have been established. The same part data is used in all the CAD, CAM and CAPP modules. The basic primitives and manufacturing features are used to define the feature library. Lack of an internationally accepted standard feature library, lead to a variety in feature definitions, but the studies of ISO terminated this problem. Promisingly, their feature definitions library expanding and attributes of existing features are being updated with the arising needs.

One disadvantage is that; the existing CAD systems based on geometric modeling become useless, a new CAD system is required for feature based modeling. But pleasingly, number of design-by-feature systems is increasing and their capabilities are advancing.

2.1.4 Review of Feature Modeling Approaches

Presented feature modeling approaches have their own advantages and limitations. Despite the difficulties for feature recognition are remaining, drawbacks feature based design approach are decreasing day by day. For example some difficulties for feature recognition are;

- Need of sophisticated and model dependent algorithms,
- Difficulty in recognition of compound features, created by Boolean operations.

Examples to the decreasing drawbacks of design by features are:

- A universal feature library is being developed for standardization,
- The number of design-by-feature systems are increasing,
- Design by features allows the designer to create a richer and higher level product data to be used by downstream applications, even faster and more conveniently.

2.2 Product Data Models

Product data model is aimed to satisfy the need of representing and exchanging information about a product generated or gathered during, and used in, design and manufacture of that product. Therefore, the contents of this product model must be able to support the information needs of a large variety of computerized manufacturing applications (i.e., CAPP, part programming, etc.). Popularly used CAD, CAPP and CAM systems used for creating, representing, exchanging and using product design data and their variety lead to the introduction of different product data models such as IGES, SET, DXF, etc. Most of the product data models, focused on representing and exchanging geometric information about the product, have shown a success in transferring data between CAD systems and supplying input for CAPP and CAM systems, but they have failed to supply high-level information for CAPP and CAM systems. As a solution, the International Standards Organization (ISO) first proposed Product Data Exchange Specification (PDES). Although, feature based PDES, having dimension/tolerance information, seemed to establish a direct connection between CAD systems and downstream applications, its complexity and specific terms that are not familiar to a designer, limited its usage. Kim, O'grady and Young (1991) made a research on mapping design-feature taxonomy of rotational parts onto PDES-features in order to make terms familiar to designer. In order to overcome the problems of PDES, also using the experience gained from PDES, ISO introduced a new standard named 'Standard for the Exchange of Product Data', in short 'STEP'. STEP is comprehensive exchange standard that would satisfy the requirements of CAD-CAPP-CAM chain integration (Usher, 1996; PDES, 1998).

In order to present the historical and technological evolution of product data models, nationally or internationally approved and defacto (an unofficial standard that is widely used throughout industry) standards for product data model representation and exchange are investigated.

2.2.1 IGES

- Development of IGES can be accepted as the initiator of attempts to define a formal standard for product data exchange.
- IGES development was based upon the concepts using neutral format and half translators.
- IGES standard was launched in January of 1980.
- It didn't take too much for the CAD vendors to realize the possible profit of utilizing ability of product data exchange using IGES.
- Problems of the initial launch were corrected by the contribution of United States Air Force, Army and Navy, and the National Aeronautics and Space Administration (NASA).
- IGES became an accepted American national standard under ANSI Y14 (American National Standards Institute).
- IGES is continually being updated, requiring compatibility between versions.

- Scope of IGES is limited to store and exchange geometric data of the product.

2.2.2 SET

- SET is the product of the efforts driven by French industry, especially automotive and aerospace industries.
- SET became a French national standard.
- With the introduction of STEP and realization of the importance of a single international data exchange standard, French efforts are directed to development of the STEP.

2.2.3 VDA-FS

- VDA-FS is a national German standard for the exchange of product data to satisfy the need of their automobile industry.
- VDA focused on and good at transferring surface/shell data.
- Efforts of the Germans are now also focused on developing the STEP, for the sake of uniqueness.

2.2.4 DXF

- DXF is an unofficial standard that is widely used throughout industry.
- DXF standard is published and propertied by AutoDesk, a major CAD vendor.
- Early versions of AutoCAD focused on drafting, 2D data exchange, named as 'DWG'.
- Later versions focused on solid 3D models.
- DXF is the most widely used product data model throughout the industry.
- DXF includes 3D geometric data (DWG includes 2D) and lacks in including the scope of product model data that is included in STEP.

2.2.5 STEP

- The development of STEP started in 1984 as a successor of IGES, SET and VDA-FS (ISO TC184 / SC4, 1984).
- “ISO 10303” is the standard reserved for STEP.
- The initial plan was that "STEP shall be based on one single, complete, implementation-independent Product Information Model, which shall be the Master Record of the integrated topical and application information models" (ISO TC184 / SC4, 1988).
- But because of the complexity, the standard had to be broken up into smaller parts that can be developed, balloted and approved separately (ISO TC184 / SC4, 1990).
- In 1994/95 ISO published the initial release of STEP as international standards (IS) with the parts 1, 11, 21, 31, 41, 42, 43, 44, 46, 101, AP 201 and AP 203 (ISO TC184 / SC4, 1994)
- In the second phase the capabilities of STEP got widely extended, primarily for the design of products in the aerospace, automotive, electrical, electronic, and other industries. This phase ended in the year 2002 with the second major release, including the STEP parts AP 202, 209, AP 210, AP 212, AP 214, AP 224, AP 225, AP 227, AP 232 (ISO TC184 / SC4, 1998).
- Basic harmonization between the APs is achieved by introducing the Application Interpreted Constructs and the deficits of APs are eliminated with development of the STEP modular architecture.
- Obviously STEP is still being improved by adding new APs, new additions of already released APs, according to the industrial needs, requirements for advanced product data models and arising paradigms in product life cycle management and e-manufacturing.
- STEP is able to support feature based data representation with its AP224.

2.2.6 Review of Product Data Models

Prior to the development of STEP; all standards developed for product data representation and exchange, failed to supply high-level information for CAPP and CAM systems and only dealt with geometric aspect of design data. However STEP, with its different parts, such as description methods integrated resources, application specific protocols one of which serves for feature based design of mechanical parts, is an international standard developed and supported by a large consortium and exchanging not only geometric design data but also the high-level information for CAPP and CAM systems.

In this work, STEP AP224 features are used for feature based design of rotational parts so STEP, its structure are reviewed for the sake of familiarization to the concepts as a basis for the system framework.

2.3 STEP

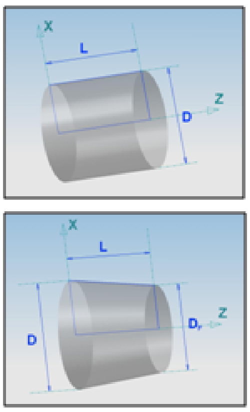
IS0 10303 is an international standard for the computer-interpretable representation and exchange of product data, Standard for the Exchange of Product Model Data; STEP. The objective is to provide a mechanism that is capable of describing product data throughout the life cycle of a product, independent from any particular system. The nature of this description makes it suitable not only for neutral file exchange, but also as a basis for implementing and sharing product databases and archiving (ISO TC184/SC4, 1994).

ISO 10303-Part 1 summarizes the need for developing STEP by saying: "The information generated about a product during its design, manufacture, use, maintenance, and disposal is used for many purposes during that life cycle. The use may involve many computer systems, including some that may be located in different organizations. In order to support such uses, organizations need to 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".

STEP is organized as a series of parts, each published separately. These parts fall into one of the following series: description methods, integrated resources, application protocols (APs), abstract test suites, implementation methods, and conformance testing (ISO TC184/SC4, 1994).

STEP uses a formal specification language, EXPRESS, to specify the product information to be represented. The use of a formal language enables precision and consistency of representation and facilitates development of implementations (ISO TC184/SC4, 1994).

EXPRESS specification of 'Outer Diameter' feature from AP224 is given for illustration in Figure 2.5.




```
ENTITY outer_diameter
 SUBTYPE OF (outer_round);
 diameter : numeric_parameter;
 feature_length : numeric_parameter;
 reduced_size : OPTIONAL taper_select;
END_ENTITY;

TYPE taper_select = SELECT (diameter_taper, angle_taper);
END_TYPE;

ENTITY diameter_taper;
 final_diameter: numeric_parameter;
END_ENTITY;

ENTITY angle_taper;
 angle: numeric_parameter;
END_ENTITY;
```

Figure 2.5. Sample EXPRESS specification

For the exchange of the product information specified by EXPRESS, one of the neutral file approach is employed, either Part 21 or Part 28 of ISO 10303. Product data transfer between two different applications is an implementation requiring (Figure 2.6):

- a sending system translator (pre-processor) to generate neutral data file
- a transport mechanism for sending neutral files to the receiving system
- a receiving system translator (post-processor) to convert neutral data files to an internal format

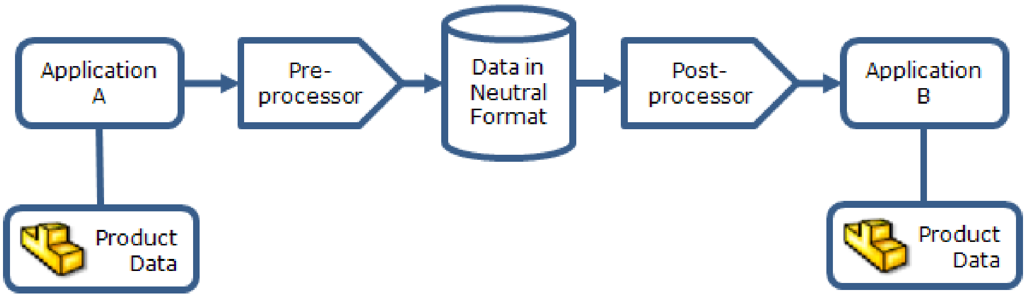


Figure 2.6. Data translation

2.3.1 STEP Structure

STEP is divided into many parts, grouped into

- Environment
  - Parts 1x: Description methods: EXPRESS, EXPRESS-X
  - Parts 2x: Implementation methods: STEP-File, STEP-XML, SDAI
  - Parts 3x: Conformance testing methodology and framework
- Integrated data models
  - The Integrated Resources (IR), consisting of
    - Parts 4x and 5x: Integrated generic resources
    - Parts 1xx: Integrated application resources
    - PLIB ISO 13584-20 *Parts library: Logical model of expressions*
  - Parts 5xx: Application Integrated Constructs (AIC)
  - Parts 1xxx: Application Modules (AM)
- Top parts
  - Parts 2xx: Application Protocols (AP)
  - Parts 3xx: Abstract Test Suites (ATS) for APs
  - Parts 4xx: Implementation modules for APs

In total STEP consists of several hundred parts and every year new parts are added or new revisions of older parts are released. This makes STEP the biggest standard within ISO. Each part has its own scope and introduction.

2.3.2 STEP Application Protocols (APs)

STEP uses application protocols (APs) to specify the representation of product information for a particular application and industry domain. Hence APs are the top parts, most relevant for users of STEP. The APs define the scope, the information to be exchanged, the means of testing and a user guide for implementing the application (Sharma, Gao, 2002).

In this work, AP224 (Mechanical Product Definition for Process Planning using Machining Features) and its feature library is used after refining for the rotational features and for the lacking information in AP224, AP238 (Application Interpreted Model for Computerized Numerical Controllers) is referred. STEP AP224 contains all the information needed to manufacture the required part, including (Fidan, 2004; ISO TC184/SC4, 2000):

- Geometrical and topological entities required to represent manufacturing features implicitly in boundary representation format.
- Explicit representation of manufacturing features.
- Information necessary to identify the dimensional and geometrical tolerances of the manufacturing features.
- Information necessary to define material, hardness, surface finish and other technological data.

STEP AP238 defines the context, scope, and information requirements for numerical controlled machining and associated processes and specifies the integrated resources necessary to satisfy these requirements. STEP AP238 completes the information given in AP224 employing the manufacturing point of view. So these two APs are used together to build a complete feature library for representation of product data (ISO TC184/SC4, 2007).

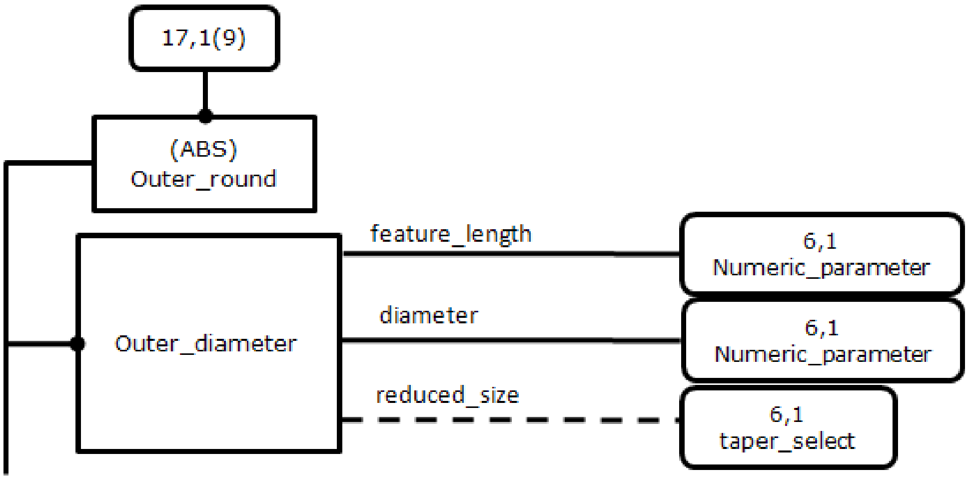


Figure 2.7. Sample EXPRESS-G specification (ISO TC184/SC4, 2000)

EXPRESS specification of the application is either implicitly or explicitly given in the related AP of STEP and the application specific information of product is exchanged using a neutral file approach. For instance AP244 presents the EXPRESS specification of 'Outer-Diameter' is implicitly given in the form of 'EXPRESS-G Diagram' as in Figure 2.7, while AP238 explicitly gives the same specification as in Figure 2.5. The neutral file output of the EXPRESS specification of 'Outer-Diameter' is presented in both (a) Part 28 and (b) Part 21 formats in Figure 2.8.

```
<machining_feature>
    <placement>
        <location x="0" y="0" z="100" />
    </placement>
        <outer_diameter id="SOD2">
            <feature_length length="200" />
            <diameter diameter="100" />
        </outer_diameter>
</machining_feature>
```

(a) STEP Part 28 Output

```
#10=OUTER_DIAMETER(ID_feature,#11,#12,#13,$);
#11=ORIENTATION((x_pos,y_pos,z_pos),(xi_dir,xj_dir,xk
_dir,yi_dir,yj_dir,yk_dir,zi_dir,zj_dir,zk_dir));
#12=NUMERIC_PARAMETER('DIAMETER',unit_length,
Diameter,#14);
#13=NUMERIC_PARAMETER('LENGTH',unit_length,Le
ngth,#15);
```

(b) STEP Part 21 Output

Figure 2.8. Neutral file output of the EXPRESS specification

2.3.3 STEP Implementation

The STEP standard covers the exchange of product model data. A STEP implementation is an application that uses this standard to exchange product information, or makes it possible for other applications to do so. Thus, STEP implementations cover the range from CAD systems, bill of materials systems and so forth, to stand-alone translators, to packages that make it possible to develop the above systems (Loffredo, 1999).

STEP implementations fall into several categories. Translators take data from preexisting systems and convert it into STEP AP defined data. The tool converts non-STEP data into STEP data. Other applications might take STEP data as input, and then perform some function on it, generating more STEP output. An example of such a thing would be an application that takes partial AP information from several sources, like geometry from a CAD system and configuration information from a CM system, and then merges them into a complete AP-203 exchange file. Another category might be an application that takes specific AP data and performs some analysis on it, such as a finite element package or a geometry visualizer. These applications should all work from STEP exchange files and possibly a shared database as well, so it is important to consider how the application will be tied to the EXPRESS information models of the various APs that they will work with. A number of techniques have been used successfully on previous implementations (Loffredo, 1999).

The roadmap for STEP implementation can be summarized as:

- The first task in a STEP implementation is to determine the proper application protocol for your problem area.
- After familiarizing with the correct application protocol, the project requirements should be examined and the tools should be selected that will help to meet those requirements.
- Once the AP and support tools are in place, a simple prototype should be produced to gain experience, and troubleshoot the mechanics of compiling, linking and so forth.

- Afterwards more and more functionality should be introduced and the correctness of the implementation should be verified using conformance test suites.
- At last, implementation should be running, the major elements of a STEP implementation should all be in place and only incremental improvements related to the problem domain should be needed.

2.4 Machining

Machining is one of the most important material removal methods and it is a collection of material-working processes in which power-driven machine tools, such as saws, lathes, milling machines, and drill presses, are used with a sharp cutting tool to mechanically cut the material to achieve the desired geometry.

In this work; turning, related cutting tools and terminology, its automation with CNC lathes and NC-codes are reviewed shortly.

2.4.1 Turning and Cutting Tools

Turning is one of the most common of metal cutting operations. In turning, a workpiece is rotated about its axis as single-point cutting tools are fed into it, shearing away unwanted material and creating the desired part. Turning can occur on both external and internal surfaces to produce an axially-symmetrical contoured part.

Parts ranging from pocket watch components to large diameter marine propeller shafts can be turned on a lathe. The capacity of a lathe is expressed in two dimensions. The maximum part diameter, or "swing," and the maximum part length, or "distance between centers."

Coordinate system of turning machine tool (lathe) is coming from the general definition of ISO: for a machine tool with spindle, z axis is the axis of the spindle and x axis is the axis radial and parallel to the cross slide. In both axis (+) direction is accepted as tool goes away from the workpiece.

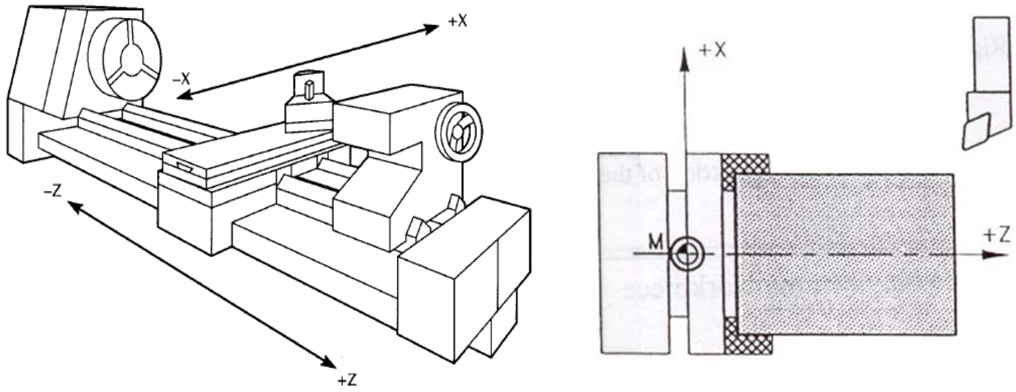


Figure 2.9. Standard lathe coordinate system

The general-purpose engine lathe is the most basic turning machine tool. As with all lathes, the two basic requirements for turning are holding the work while it rotates and holding cutting tools and moving them to the work.

2.4.1.1 Turning Parameters

Regardless of the type of lathe, three key parameters determine productivity and part quality. These parameters are:

- the cutting speed
- the feed rate
- the depth of cut

Cutting Speed (v_c)

The workpiece rotates at a certain number of revolutions (*n*) per minute. This gives a specific cutting speed, v_c (or surface speed), measured in (m/min) at the cutting edge.

$$v_c = \frac{\pi . D . n}{1000} \quad m/min \tag{2.1}$$

Cutting Depth (a_p)

The cutting depth (a_p) is the difference between the un-cut and cut surfaces. The cutting depth is measured in mm and at a right angle (90°) to the feed direction.

Feed (f_n)

The axial (or in face turning the radial) tool movement is called feed, f_n, and is measured in mm/rev. When feeding radially towards the centre of the workpiece, the rpm will increase, until it reaches the rpm limit of the machine spindle. When this limitation is passed, the cutting speed, v_c, will decrease until it reaches 0 m/min at the component centre.

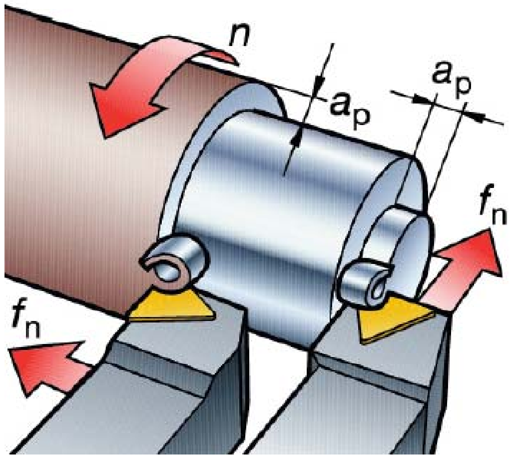


Figure 2.10. Key parameters effecting productivity and quality in turning (SANDVIK Coromant)

Three operation types can be defined according to the depth of cut and feed used during machining:

- Finishing (F)
- Medium (M)
- Roughing (R)

These basic definitions are from “SANDVIK Coromant Machining Technical Guide” with feed and depth of cut values to be a reference for the comparison between operation types. Feed and depth of cut values are determined considering CNC machine tool, material, heat treatment, surface hardness of the workpiece and the proper cutting tool together.

Finishing

Operations at low depths of cut and low feeds:

$$f_n = 0.1 - 0.3 \text{ mm/rev}$$

$$a_p = 0.5 - 2.0 \text{ mm}$$

Medium

Medium to light roughing operations. Wide range of depths of cut and feed rate combinations:

$$f_n = 0.2 - 0.5 \text{ mm/rev}$$

$$a_p = 1.5 - 5.0 \text{ mm}$$

Roughing

Operations for maximum stock removal and/or severe conditions. High depths of cut and feed rate combinations:

$$f_n = 0.5 - 1.5 \text{ mm/rev}$$

$$a_p = 5.0 - 15.0 \text{ mm}$$

2.4.1.2 Cutting Tools

Cutting tools for metal cutting have many shapes, each of which are described by their material or geometry. Every one of these tool shapes has a specific purpose in metal cutting. The primary machining goal is to achieve the most efficient separation of chips from the workpiece. For this reason, the selection of the right cutting tool material and geometry is critical.

Nearly all turning processes use single point cutting tools, this is, tools that cut with only a single edge in contact with the work. Most turning is done with coated indexable carbide inserts, but the tool material may also be high-speed steel, brazed carbide, ceramic, cubic boron nitride, or polycrystalline diamond. 75 percent of turning operations use just a few basic tool geometries. The geometry of an insert includes:

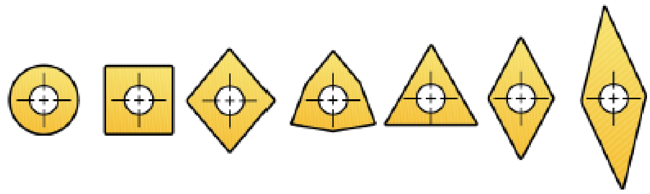
- the insert's basic shape
- its relief or clearance angle
- the insert type
- the insert's inscribed circle or "IC" size
- the insert's nose radius
- the insert's chip breaker design

Insert Shape - Number of Cutting Edges

The number of cutting edges on an insert varies depending on the choice of insert and nose angle. An insert with a negative basic shape normally has twice as many edges compared to a positive insert. In heavy roughing, a single-sided, negative basic-shape insert is recommended for best stability. For other roughing operations, a double-sided insert with twice as many cutting edges is recommended. The round insert has the highest number of cutting edges (Figure 2.11).

Insert Shape – Depth of Cut

The recommended maximum values in the table are intended to provide machining reliability for continuous cuts using a roughing geometry. Deeper cuts, up to the total cutting edge length, *l*, can be made for a shorter period (Figure 2.12).



Basic shape	R	S	C	W	T	D	V
Negative Double sided	*) ∞	8	4	6	6	4	4
Single sided	∞	4	2	3	3	2	-
Positive	∞	4	2	3	3	2	2

Figure 2.11. Insert Shape - Number of Cutting Edges (SANDVIK Coromant)

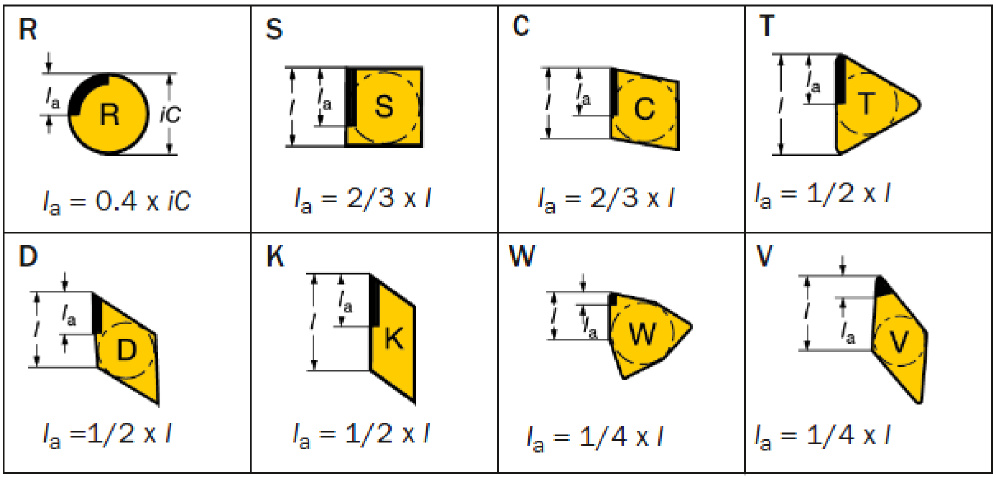


Figure 2.12. Insert Shape – Depth of Cut (SANDVIK Coromant)

<u>Insert Size and Depth of Cut</u>

The depth of cut influences the metal removal rate, the number of necessary cuts, chip breaking, and the power required.

Establish the effective cutting edge length l_a along with the shape of the insert, the entering angle (or complimentary to side cutting-edge angle), κ_r, of the tool holder, and the depth of cut, a_p (Figure 2.13).

$$l_a = \frac{a_p}{\sin{(\kappa_r)}} \tag{2.2}$$

The minimum necessary effective cutting edge length can be determined from Figure 2.12, which shows the relation between the depth of cut, a_p, the entering angle, κ_r. For extra reliability in more demanding operations, a larger and thicker insert should be considered.

When machining against a shoulder, the depth of cut increases dramatically; in such cases, a stronger insert (thicker or larger) should be used to minimize the risk of insert breakage.

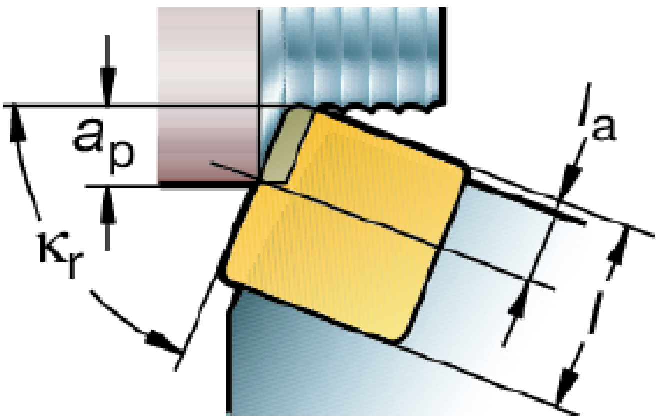


Figure 2.13. Insert Size and Depth of Cut (SANDVIK Coromant)

<u>Nose radius</u>

The nose radius, r_e, on the insert is a key factor in turning operations. Selection of nose radius, influencing the surface finish, chip breaking and insert strength, depends on the:

- Depth of cut, a_p
- Feed, f_n

'Small nose radius' is ideal for small cutting depths, reduces vibration and less insert strength, while 'large nose radius' is preferred for heavy feed rates, large depths of cut, stronger edge and increased radial forces.

When turning with inserts, much of the geometry is built into the tool holder itself rather than the actual insert.

Insert type tool holders for turning consist of a shank, head, insert pocket, and clamping hardware. Tool holders are either right or left handed, or neutral. The size and type of the tool holder are determined by:

- the turning operation
- the feed direction
- the size of cuts
- machine tool design
- the need for accessibility
- the shape of workpiece

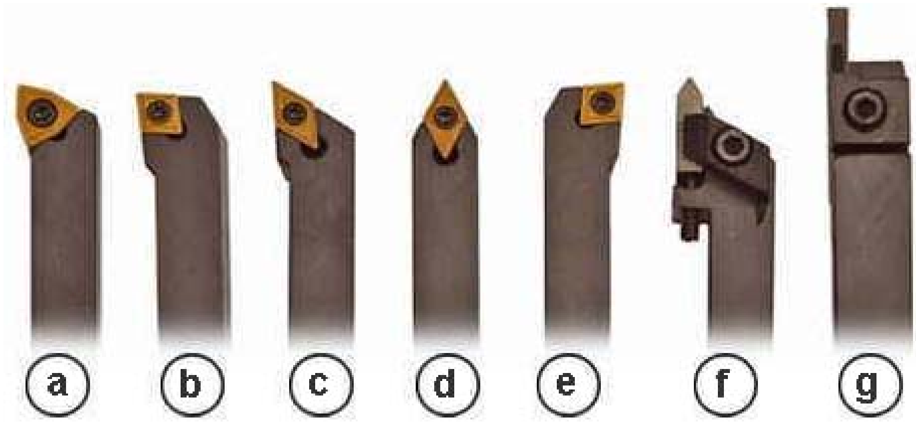


Figure 2.14. External Turning Tools (Cut-Tools)

In Figure 2.14, (a), (b) and (c) are right handed (RHS) external turning tools, (d) is neutral external turning tool, (e) is left handed (LHS) external turning tool, (f) is external threading tool and (g) is external grooving/parting tool.

Figure 2.15. Drilling Tools (ROSHANI PIPE & FITTINGS PVT LTD.)

Figure 2.16. Internal Turning (Boring) Tools (Boring Bars Site)

2.4.2 CNC Turning and NC-Codes

CNC turning is rapidly replacing the conventional turning due to its ease of setting and operation. The part may be designed and the tool paths programmed by the CAD/CAM process, and the resulting file uploaded to the machine, and once set and trialled the machine will continue to turn out parts under the occasional supervision of an operator.

2.4.2.1 CNC Machine Tools

The CNC machine is controlled electronically via a computer menu style interface, the program may be modified and displayed at the machine, along with a simulated view of the process. The setter/operator needs a high level of skill to perform the process, however the knowledge base is broader compared to the older production machines where intimate knowledge of each machine was considered essential. These machines are often set and operated by the same person, where the operator will supervise a small number of machines.

Even with the rapid advancement of information technology, its integration with the manufacturing technology and the usage of advanced internet technology, all the CNC's in a production environment can be controlled by a central automated system called e-manufacturing. CNC's are the last loop of the CAD-CAPP-CAM integration, consequently with the realization of full integration, the product data model including high-level information is intended to be directly used on CNC's. Indeed, CNC's are needed to be programmed by a process planner, either manually or by the aid of an interactive CAPP system, using 'NC-codes' to realize a product design.

2.4.2.2 NC-Codes

NC-code is the common name for the most widely used computer numerical control (CNC) programming language, which has many implementations.

The first implementation of numerical control was developed at the MIT Servomechanisms Laboratory in the early 1950s. The main standardized version used in the United States was settled by the Electronic Industries

Alliance in the early 1960s. A final revision was approved in February 1980 as *RS274D*. In Europe, the standard *ISO 6983* is often used, although in varied states sometimes used other standards, example *DIN 66025* or PN-73M-55256, PN-93/M-55251 in Poland.

Some CNC machine manufacturers attempted to overcome compatibility difficulties by creating own standards but basically employing the commonly used codes and adding the new ones.

In this work, commonly accepted codes by all RS274D, ISO6983 and almost every CNC manufacturer are used especially for turning and here is the summary listing of G and M codes.

Table 2.1. G-Code Listings

G00	Positioning (rapid transverse)
G01	Linear interpolation (feed)
G02	Circular interpolation (CW)
G03	Circular interpolation (CCW)
G04	Dwell
G20	Programming in inches
G21	Programming in mm
G28	Return to home position
G40	Tool radius compensation off
G41	Tool radius compensation left
G42	Tool radius compensation right
G96	Constant surface speed control
G97	Constant surface speed control Cancel
G98	Feed per minute
G99	Feed per revolution

Table 2.2. M-Code Listings

M00	Program stop
M01	Optional program stop
M02	Program stop and rewind
M03	Start of spindle rotation (CW)
M04	Start of spindle rotation (CCW)
M05	Spindle stop
M06	Auto tool change
M10	Chuck open
M11	Chuck close
M38	Door open
M39	Door close
M98	Sub program call
M99	Sub program end

CHAPTER 3

FRAMEWORK OF THE SYSTEM

3.1 General Architecture of the System

As stated in the objective and scope of the thesis, system development includes two main phases: (1) feature based modeler development phase, (2) NC-code generator development phase. In this chapter, these two main phases with every sub steps are discussed in detail, in the order of development and running sequence during usage. The system components, working principles, idea behind the constructs will be covered in this chapter. Furthermore, example procedures among the repetitive stages and sample calculations will be presented in this chapter.

Main system components and system development phases are illustrated in Figure 3.1, and can be summarized as:

Feature Library

The starting point of the system development is building a feature library for a feature based design environment and AP224 features of STEP is selected as a basis for this purpose. Rotational part defining features, from manufacturing view point turning features; are extracted from AP224. For the missing definitions in AP224, the AP238 is used as a complementary application protocol, and refined to build a feature library.

AP224 and AP238 are specified in an object-oriented manner due to the object-oriented STEP specification language, “EXPRESS”, either in the form of EXPRESS Schemas or EXPRESS-G Diagrams. For the sake of coherency, "Dynamic Link Library" (dll) for the features, built up in an object-oriented manner using Visual Basic 6.0 as well as the feature library.

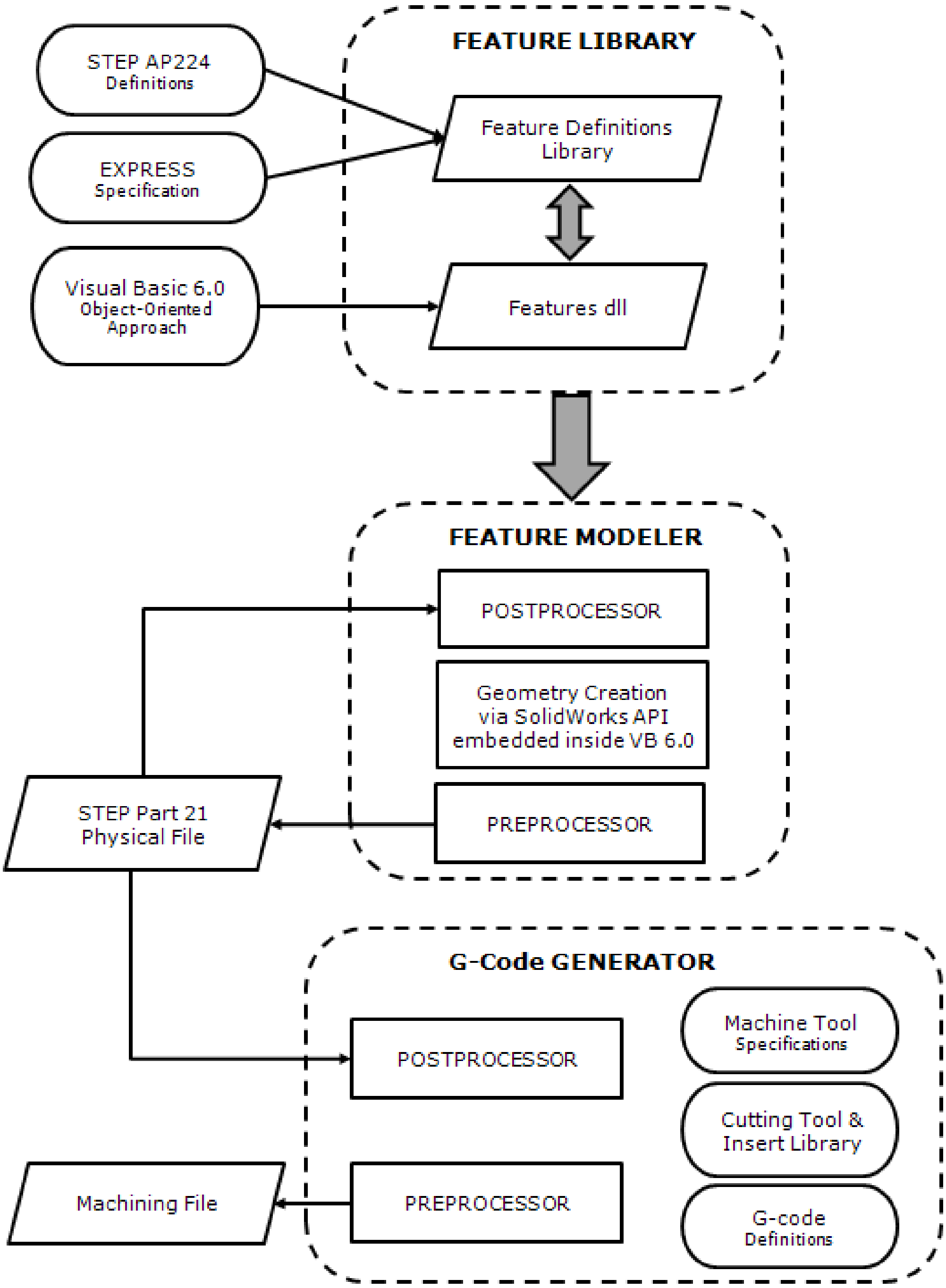


Figure 3.1. General System Architecture

Feature Modeler

Feature modeler is the rotational part design module of the developed system, using the built in feature dll and having a link with SolidWorks to 3D visualization of design geometry. Feature modeler is formed from a series of user forms specially designed for each feature in the library and integrated with error handling algorithms to improve the design capability. In the meantime "UNDO" option is added to feature modeler module to overcome the unexpected errors and synchronic preprocessor is embedded in the feature modeler for the design data storing and exchange purposes.

Feature Modeler Preprocessor

This preprocessor is an algorithm responsible for adding feature definition into the STEP Part 21 physical file, synchronously with creation of feature by the feature modeler. In the same manner, UNDO option of feature modeler synchronously deletes the feature definition from the STEP Part 21 physical file.

STEP Part 21 Physical File

Part 21 file is one of the neutral file system defined by STEP and in this work feature modeler of the system developed is preferred to output design data in Part 21 file system, for being eligible and mostly preferred STEP file system by commercial applications.

Postprocessor

The Part 21 file output by the feature modeler module needed to be input for the NC-code generation module of the developed system. An algorithm is composed to find and extract the embedded feature definitions from the neutral file and regenerate product design data. This postprocessor uses the same feature library for both geometry generation and feature definitions.

NC-code Generator

NC-code generator module of the developed system is responsible for accepting a STEP Part 21 physical file based on the built feature library and generating NC-code for the product design data. NC-code generator accepts the needed machine tool data and list of available cutting tools and inserts with their machining parameters from the process planner and can generate NC-codes for both simulation and machining purposes; this adds versatility to the module.

3.2 Feature Library

Building a feature library is the initial step for the system development, since the aim is to develop a STEP AP224 features based design environment and use the feature based design data directly for a downstream application, NC-code generation.

AP224 and AP238, as a complementary application protocol, are used as a basis for building a feature library. Rotational features specific for turning are extracted and collected in the library with their;

- Geometric data
- Tolerance data
- Properties data

The structure of the feature library is shown in Figure 3.2.

Feature definitions are represented using 'EXPRESS-G Diagrams' in AP224 and 'EXPRESS Schemas' in AP238, therefore basic definitions, data types and structure of the EXPRESS Specification of STEP is given for familiarization.

Extraction of rotational features from AP224 and AP238 and related data are presented for unique rotational features.

The 'feature library' built with the given procedure and used for the developed system is given in 'Appendix D'.

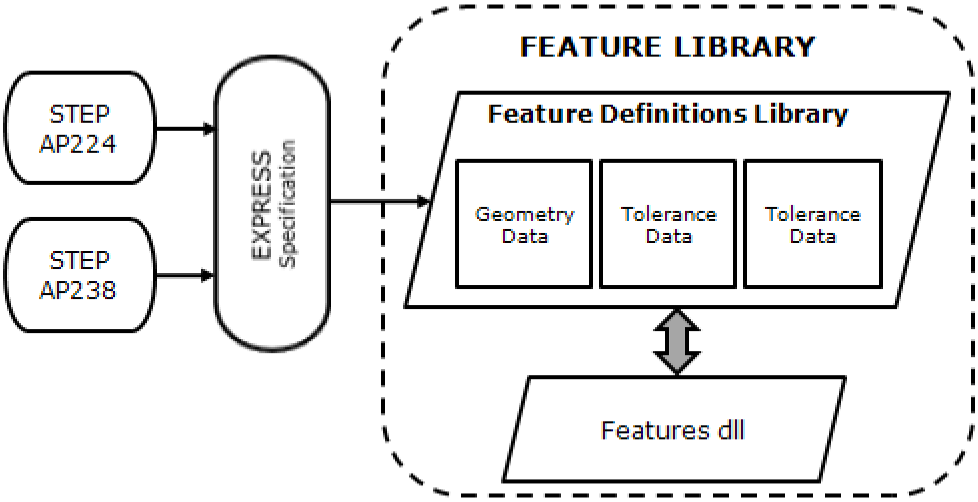
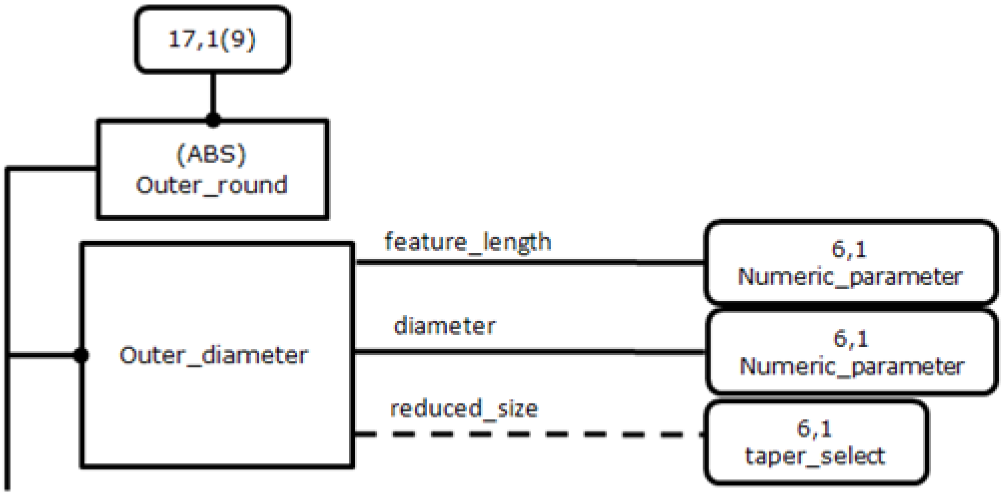


Figure 3.2. Structure of the Feature Library

3.2.1 EXPRESS Specification of STEP

Basics of STEP implementation rely on selection of proper application for the desired task, understanding the selected protocol, finally extracting and using the related entities from the application protocol (AP). The AP documents can be quite long, but most of this is the Application Interpreted Model (AIM). The AIM is an EXPRESS information model that formally describes the application objects in terms of a library of pre-existing definitions, called the generic resources or integrated resources. This highly normalized representation contains the structures as well as the constraints that those structures must obey. The AIM is used as the basis for the implementation and data exchange. An easier way to become familiar with the AIM is to look at the EXPRESS-G diagrams in the AP document. EXPRESS-G is a formal diagrammatic form for the EXPRESS language and can be converted to EXPRESS Specification which is the formal language form that uses a lexical notation and syntax defined by a grammar (Figure 3.3). Oppositely, EXPRESS Specification for a defined data can be converted to EXPRESS-G form of STEP.




```
ENTITY outer_diameter
    SUBTYPE OF (outer_round);
    diameter : numeric_parameter;
    feature_length : numeric_parameter;
    reduced_size : OPTIONAL taper_select;
END_ENTITY;
```

Figure 3.3. Different representation forms of EXPRESS

In order to comprehend the structure of data defined by EXPRESS and extract the required parameters, some definitions and data types of EXPRESS should be known (Amaitik, 2003).

<u>Some facts about EXPRESS</u>

- The EXPRESS language is not case sensitive. As a convention, EXPRESS keywords are often written in uppercase to improve readability.
- EXPRESS is not a programming language. It is a data modeling language.
- EXPRESS is readable to humans and fully computer interpretable.

EXPRESS Schemas

EXPRESS specifications are organized into schemas. An EXPRESS schema is a name space of named data types. Data types may be simple types such as strings and integers or entity types, representing more complex collections of attributes (properties). Schemas can be related together to form models. The following example is taken from 'The EXPRESS Language Reference Manual' (ISO TC184 / SC4, 1997).

```
SCHEMA Family;

        ENTITY Person
          ABSTRACT SUPERTYPE OF (ONEOF (Male, Female));
            name: STRING;
            mother: OPTIONAL Female;
            father: OPTIONAL Male;
        END_ENTITY;

        ENTITY Female
          SUBTYPE OF (Person);
        END_ENTITY;

        ENTITY Male
          SUBTYPE of (Person);
        END_ENTITY;

END_SCHEMA;
```

EXPRESS-G diagram of the 'Family' schema is given in Figure 3.4.

Entities

An entity is analogous to an object in an object oriented environment or feature in a feature based application. Each entity has a name and a set of attributes. Each attribute has a name and a data type.

```
ENTITY Entityname;
   a1: data_type;
   a2: data_type;
   ...
   aN: data_type;
END_ENTITY;
```

For example a point entity might be defined as follows:

ENTITY Point;
x: REAL;
y: REAL;
END_ENTITY;

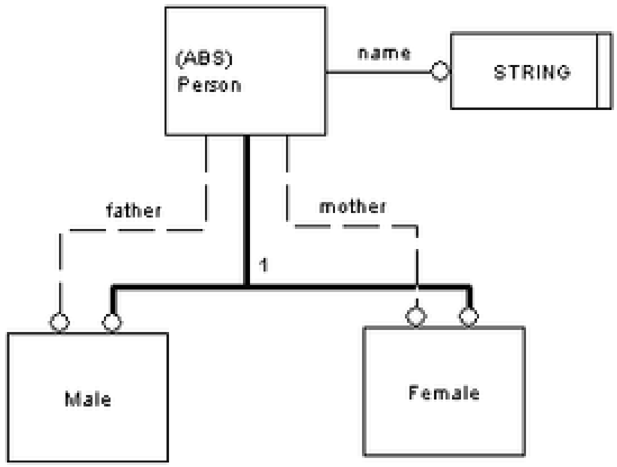


Figure 3.4. EXPRESS-G Diagram for the 'Family' SCHEMA (ISO TC184 / SC4, 1997).

Data Types

The data type of an attribute can be either a simple type, an aggregate type, a defined type, an enumeration type, an entity type, or a select type.

- *Simple Data Type* is one of implicitly defined primitive data types, such as integer, real, Boolean, logical, string, and binary.
- *Aggregate Data Type* is a container like data type that holds multiple elements of the same type. The EXPRESS aggregate types are:
  - *Bag:* An unordered collection. Null values are not allowed. Duplicate values are allowed.
  - *List:* An ordered collection. Null values are not allowed. Duplicate values are allowed.

- o *Set:* An unordered collection. Neither null values nor duplicate values are allowed.
- o *Array:* An ordered collection of fixed size. Both null values and duplicate values are allowed.

- *Entity Data Type:* Any entity declared in a schema can be used to specify the data type of an attribute. Using an entity as an attribute's data type establishes a relationship between the two entities. For example, using the Entity "Point" shown earlier, we could define a "Line" entity as follows:

```
ENTITY Line;
    enda: Point;
    endb: Point;
END_ENTITY;
```

- *Select Data Type* allows an attribute to be of one of several possible types. Items in the selection type's list are restricted to be either entity data types, or defined data types. A select type must be defined within a TYPE block. Example:

```
TYPE physical_connection_type = SELECT
      (nail, screw, glue);
END_TYPE;
...
attachment_method : physical_connection_type;
```

  In this example, the value of an *attachment_method* attribute can be either a *nail* , a *screw* , or *glue* . It is assumed that nail, screw, and glue are either entity types or defined types.

Inheritance

Subtypes in EXPRESS allow new types to be derived from existing types. The derived types are "almost like" other existing types, with some incremental changes. They inherit attributes and functionality from their supertypes. Subtypes can define additional attributes and functionality, thereby extending or restricting the existing data types.

The EXPRESS language supports several types of inheritance relationships. The following example shows one type of inheritance relationship supported by EXPRESS:

```
ENTITY Point3D
   SUBTYPE OF (Point);
   z : REAL;
END_ENTITY;
```

The entity Point3D will have three attributes: x and y which are "inherited" from Point , and z, which is declared locally.

Reading from EXPRESS-G Diagrams

- Square blocks are entities, or objects, or features.
- Bold solid lines connect an entity to the sub entities.
- Oval blocks are sub entities or data types belonging to parameters.
- Parameter names are just above the lines connecting entities to the data types.
- Solid lines are ‘mandatory’ parameters.
- Dashed lines are ‘optional’ parameters.
- Square blocks with dashed lines are specific to ‘Select Data Types’.

No example is given for reading from EXPRESS-G Diagrams, since the next section covers this subject.

3.2.2 Extraction of Features from EXPRESS Specification of APs

AP224 presents the feature definitions basically in EXPRESS-G form and AP238 in EXPRESS Specification form. Thus both form of the STEP EXPRESS is used for feature extraction. In this work for the sake of using single representation, all the definitions extracted from AP224 in EXPRESS-G form are converted to EXPRESS Specification form and cross checked from AP238 and missing definitions are completed using AP238.

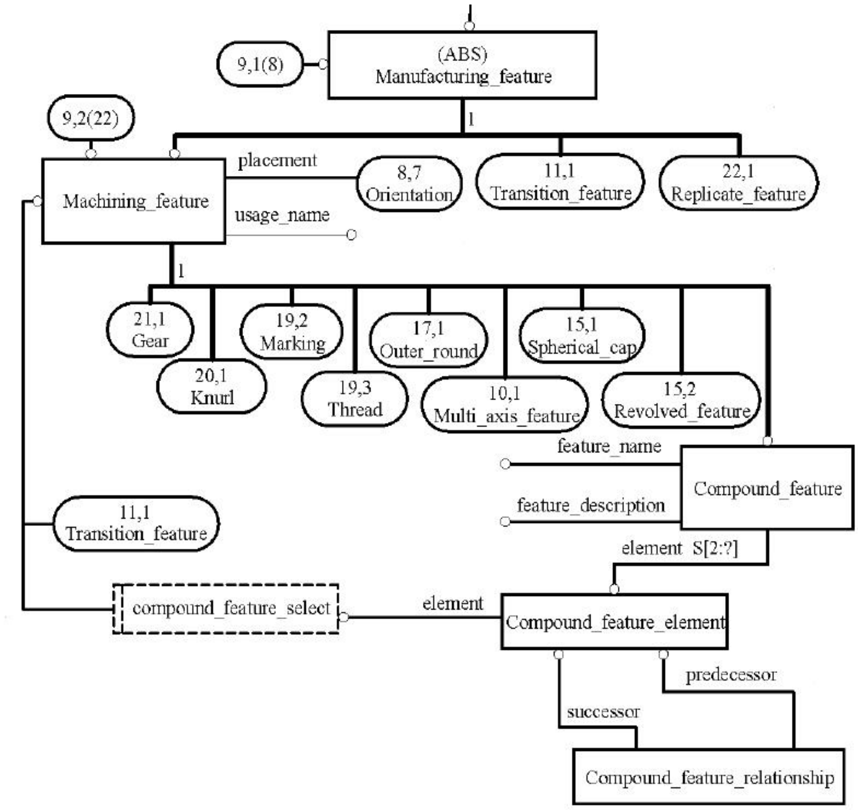


Figure 3.5. EXPRESS-G for '*Manufacturing_feature*' from AP224 (ISO TC184/SC4, 2000)

First of all, turning specific features which are planned to use in this work are determined and extracted from EXPRESS-G Diagram of AP224 for '*Manufacturing_feature*' (Figure 3.5). Those are '*Outer_round*', '*Spherical_cap*', '*Revolved_feature*' and '*Transition_feature*'. Although, '*Thread*' and '*Knurl*' are rotational features they are excluded from this work, since simulator and machine tool planned to be used are incapable of dealing with them.

'*Compound_feature*' is also picked up for the library since counterbore hole and countersunk hole features are encapsulated in this definition.

EXPRESS Specification form can be obtained from the EXPRESS-G form of the STEP representation as:

```
ENTITY manufacturing_feature
   ABSTRACT SUPERTYPE OF (ONEOF(machining_feature,
   transition_feature, replicate_feature));
END_ENTITY;

ENTITY machining_feature
   ABSTRACT SUPERTYPE OF (ONEOF(gear, knurl, marking,thread,
   outer_round, multi_axis_feature, spherical_cap, revolved_feature,
   compound_feature))
   SUBTYPE OF (manufacturing_feature);
   usage_name: STRING;
   placement: orientation;
END_ENTITY;
```

Outer_round

'*Outer_diameter*' and '*Outer_diameter_to_shoulder*' features are defined as sub entities of '*Outer_round*' as seen from the EXPRESS-G Diagram, '*Outer_round*' entity is connected to '*Outer_diameter*' and '*Outer_diameter_to_shoulder*' by bold solid line (Figure 3.6).

'*Outer_round*' entity is abstract supertype of either '*Outer_diameter*' or '*Outer_diameter_to_shoulder*' and subtype of '*Machining_feature*'.

```
ENTITY outer_round
   ABSTRACT SUPERTYPE OF (ONEOF (outer_diameter,
   outer_diameter_to_shoulder))
   SUBTYPE OF (machining_feature);
END_ENTITY;
```

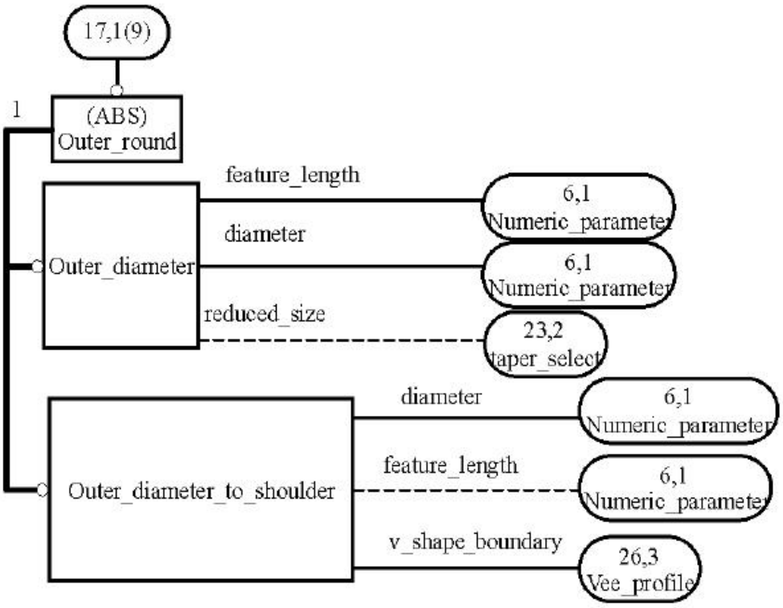


Figure 3.6. EXPRESS-G for '*Outer_round*' from AP224 (ISO TC184/SC4, 2000)

Outer_diameter

'*Outer_diameter*' is a sub entity of '*Outer_round*' as stated and the '*Outer_diameter*' entity is composed of three parameters as seen in Figure 3.6:

- Mandatory (solid line stands for mandatory parameters) '*feature_length*' defined as '*numeric_parameter*'.
- Mandatory '*diameter*' defined as '*numeric_parameter*'.
- Optional (dashed line stands for optional parameters) '*taper_select*' which is select data type. Used if there is a reduction of size in diameter of the feature.

'*taper_select*' prefers three options according to AP224; (1) '*Directed_taper*', (2) '*Angle_taper*', (3) '*Diameter_taper*'. But, AP238 refined and reduced these options to two as; (1) '*Angle_taper*', (2) '*Diameter_taper*' which are defined as '*numeric_parameter*'.

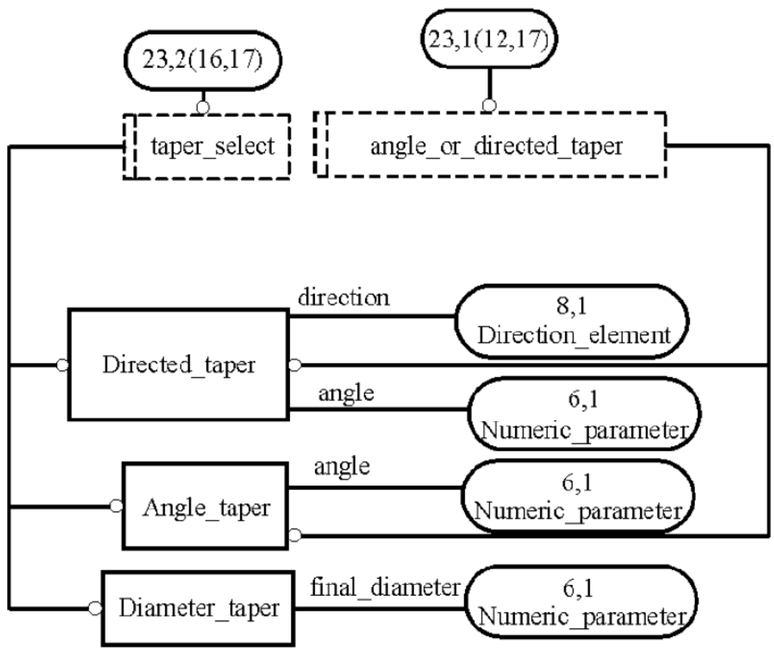


Figure 3.7. EXPRESS-G for '*taper_select*' from AP224 (ISO TC184/SC4, 2000)

Taking all these into account and cross checking from AP238 the EXPRESS Specification for '*Outer_diameter*' forms as:

```
ENTITY outer_diameter
    SUBTYPE OF (outer_round);
    diameter : numeric_parameter;
    feature_length : numeric_parameter;
    reduced_size : OPTIONAL taper_select;
END_ENTITY;

TYPE taper_select = SELECT (diameter_taper, angle_taper);
END_TYPE;

ENTITY diameter_taper;
    final_diameter: numeric_parameter;
END_ENTITY;
```

ENTITY angle_taper;
angle: numeric_parameter;
END_ENTITY;

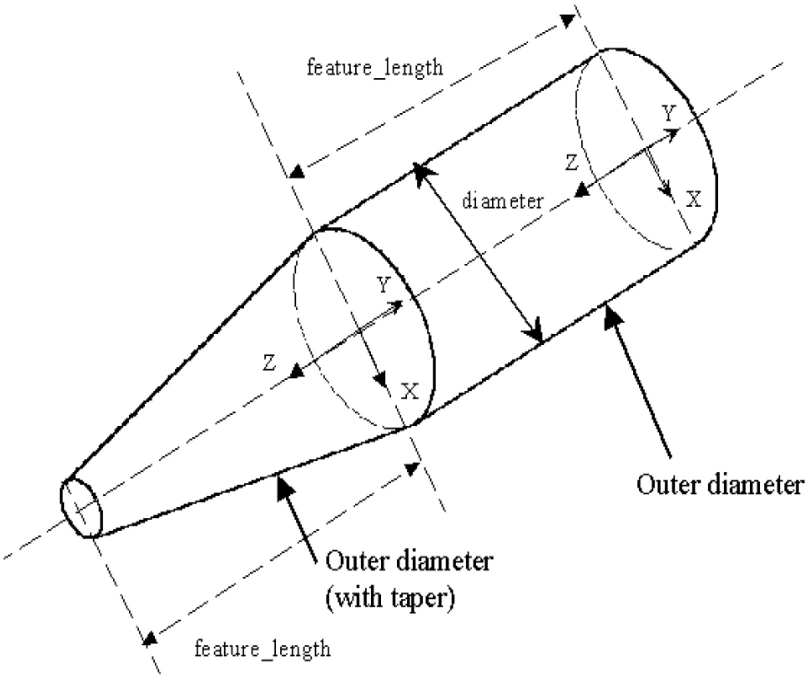


Figure 3.8. Figure 79 of AP224 (ISO TC184/SC4, 2000)

Thus the parameters to define and create the geometry of the '*Outer_diameter*' feature are determined, together with the visual aid of 'Figure 79 of AP224' as seen in Figure 3.8 as:

- Diameter
- Length
- Final diameter <u>or</u> taper angle

Outer_diameter_to_shoulder

'*Outer_diameter_to_shoulder*' is also a sub entity of '*Outer_round*' like the '*Outer_diameter*' and '*Outer_diameter_to_shoulder*' entity is composed of three parameters as seen in Figure 3.6 and additional five parameters due to '*vee_shape_boundary*' as seen in Figure 3.9:

- Mandatory '*diameter*' defined as '*numeric_parameter*'.
- Optional '*feature_length*' defined as '*numeric_parameter*'.
- Mandatory '*vee_shape_boundary*' defined as '*vee_profile*' entity type.
    - Optional '*profile_radius*' defined as '*numeric_parameter*'.
    - Mandatory '*profile_angle*' defined as '*numeric_parameter*'.
    - Mandatory '*tilt_angle*' defined as '*numeric_parameter*'.
    - Optional '*profile_radius*' defined as '*numeric_parameter*'.
    - Optional '*profile_radius*' defined as '*numeric_parameter*'.

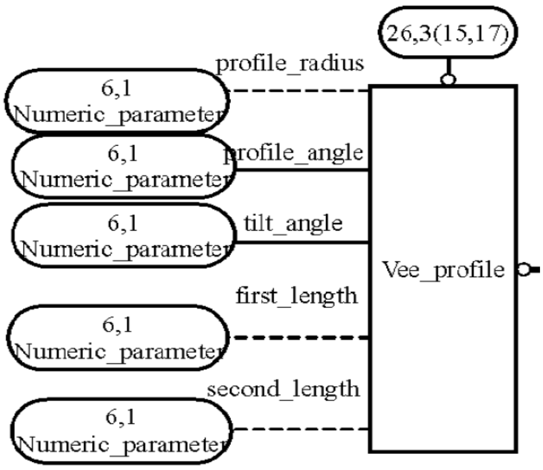


Figure 3.9. EXPRESS-G for '*vee_profile*' from AP224 (ISO TC184/SC4, 2000)

The EXPRESS Specification for '*Outer_diameter_to_shoulder*' forms as:

```
ENTITY outer_diameter_to_shoulder
   SUBTYPE OF (outer_round);
   diameter : numeric_parameter;
   feature_length : OPTIONAL numeric_parameter;
   v_shape_boundary : vee_profile;
END_ENTITY;

ENTITY vee_profile;
   SUBTYPE OF (open_profile);
   profile_radius: OPTIONAL numeric_parameter;
   profile_angle : numeric_parameter;
   tilt_angle : numeric_parameter;
   first_length: OPTIONAL numeric_parameter;
   second_length: OPTIONAL numeric_parameter;
END_ENTITY;
```

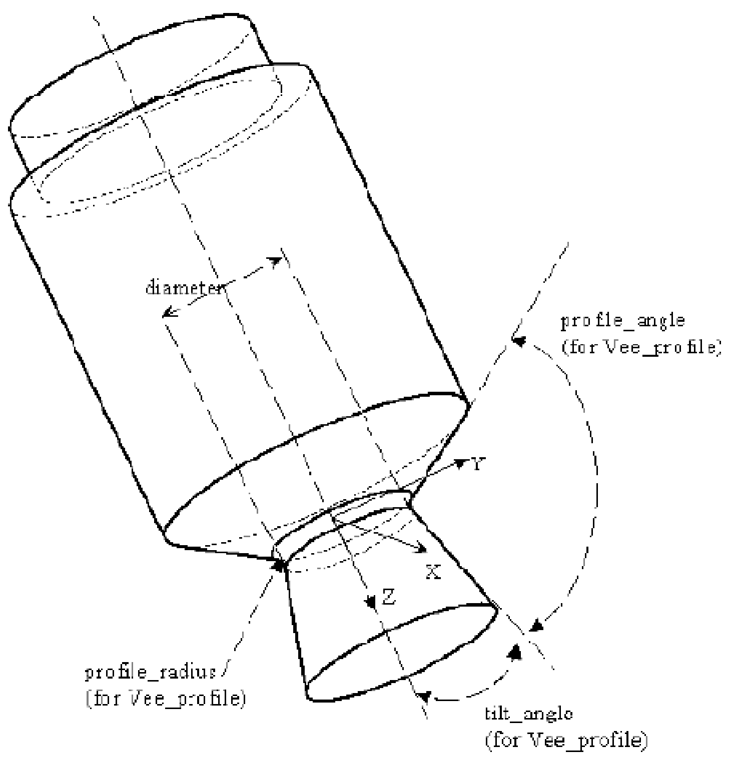


Figure 3.10. Figure 80 of AP224 (ISO TC184/SC4, 2000)

Thus the parameters to define and create the geometry of the '*Outer_diameter_to_shoulder*' feature are determined, together with the visual aid of 'Figure 80 of AP224' as seen in Figure 3.10 as:

- Diameter
- Length
- Tilt angle
- Profile angle
- Profile radius

Creation of geometry using these parameters, together with the analytic calculations, will be discussed in 3.3 Feature Modeler section.

3.2.3 Tolerance Data

Tolerance data is one of the high level information defined by AP224 additional to the geometry definition. Every parameter of a feature defined in AP224 assigned as numeric parameter, involves the tolerance data. Since AP224 defines the numeric parameter data type as;

```
ENTITY property_parameter;
   SUPERTYPE OF (descriptive_parameter, numeric_parameter)
   parameter_name: OPTIONAL STRING;
END_ENTITY;

ENTITY numeric_parameter;
   SUPERTYPE OF (numeric_parameter_with_tolerance)
   SUBTYPE OF (property_parameter);
   parameter_unit: STRING;
   parameter_value: REAL;
END_ENTITY;

ENTITY numeric_parameter_with_tolerance;
   SUBTYPE OF (property_parameter);
   implicit_tolerance: tolerance_definition_select;
END_ENTITY;
```

```
TYPE tolerance_definition_select = SELECT(tolerance_range,
plus_minus_value, limits_and_fits);
END_TYPE;

ENTITY tolerance_range;
    upper_range: REAL;
    lower_range: REAL;
    significant_digits: INTEGER;
END_ENTITY;

ENTITY plus_minus_value;
    upper_limit: REAL;
    lower_limit: REAL;
    significant_digits: INTEGER;
END_ENTITY;

ENTITY limits_and_fits;
    deviation: REAL;
    grade: REAL;
    its_fitting_type: OPTIONAL fitting_type;
END_ENTITY;

TYPE fitting_type = ENUMERATION OF (shaft,hole);
END_TYPE;
```

This definition makes possible assigning tolerance data to feature by selecting among the three options:

- Tolerance range
- Plus minus value
- Limits and fits

3.2.4 Creation of Dynamic Link Library

After extracting rotational features from AP224 with their geometric definitions enriched with tolerance data, the major step in using these definitions as basis for a feature modeler in VB 6.0 is to classify the features, as individual objects and create a hierarchy between them and the common parameter definitions. The usual and common way in object oriented structures is to create a dynamic link library (dll) to copy the object oriented information model of STEP AP224 into VB 6.0 using classes and objects.

Every entity defined in STEP AP224 is created as a class object and the interrelation between class objects are associated as they are in the AP224. So each parameter is stored in the belonging hierarchical level and these classes in VB, used as variables assigned to separate feature forms, involved in geometry calculations and carried information to the neutral STEP Part 21 file via preprocessor.

Created classes are combined in dynamic link library (dll), compiled using VB 6.0 to be used as a package in both feature modeler and NC-code generator and named as 'TurnDESIGN.dll'.

3.3 Feature Modeler

Constitution of feature modeler is the second objective for the system development, after building a feature library as the basis of system. The feature modeler is responsible for:

- Enabling user friendly and interactive environment for rotational part design by picking up from the predefined available features.
    - The interactive forms designed for input of required data for feature creation are designed to be simple, but functional.
    - Positioning of features are automated, where possible, to avoid user from unnecessary calculations during design and to expedite the design procedure.
    - Feature creation order is restricted to help designing producible parts.
        - First external features
        - Second internal features and grooves
        - Last transition features
    - Error detection algorithms are used, and user is guided through the design process via error messages.
    - Furthermore, an 'UNDO' option enabling unlimited stepping back action stands for the cases where error detection algorithm is inadequate.

- Creating visual 3D representation of the part geometry, using SolidWorks as a linked application automated via SolidWorks API.
- Generating neutral file for design data representation by the embedded preprocessor, in the form of STEP Part 21.

3.3.1 Feature Modeler Architecture

General Architecture of the system is presented in Figure 3.11.

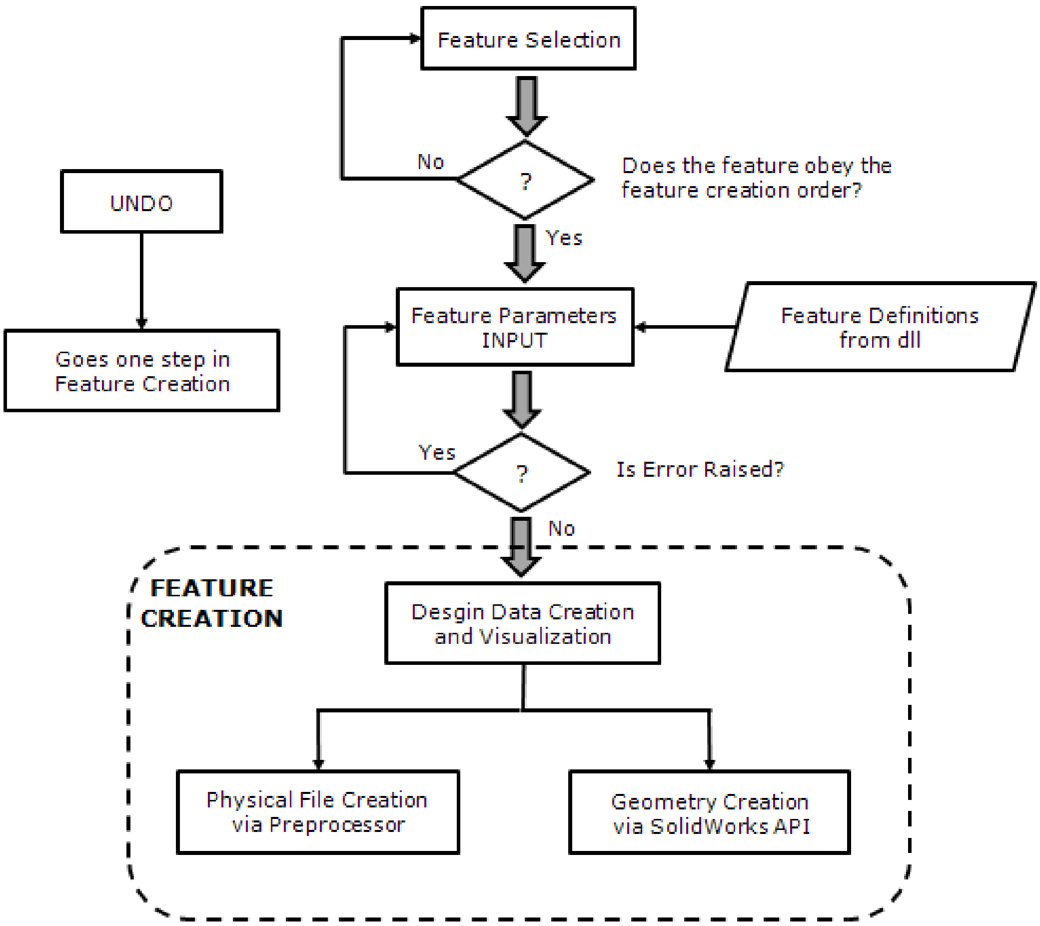


Figure 3.11. Feature Modeler Architecture

3.3.2 Feature Selection and Feature Parameters Input

As stated in the introduction to feature modeler, the order of feature creation is restricted with some simple rules for the sake of designing producible parts. So the first step in the modeling algorithm is to check the existing state of the modeling stage. For example if there is no external feature created, obviously trying to create a hole or a groove is useless, so creation of these features as the first feature is not allowed.

Creation order of features is given as; firstly external features, then internal features and grooves, last transition features. Taking this order into account, creation of a hole or groove is not allowed anymore if any transition feature is created.

Input of parameters to constrain the geometry and other properties according to the definitions of STEP AP is an important phase in modeling. For this reason, usage of object oriented structure of feature definitions and employing proper feature parameters, requires special care to eliminate ambiguity. So, simple but functional interactive forms, guiding user throughout the design process are designed for data input. To avoid unnecessary number of forms, similar or derived features are combined into one form, as in the case for '*revolved_round*' and '*spherical_cap*': '*spherical_cap*' is special case of '*revolved_round*' with sweep_angle 90° and sweep_radius is equal to radius at placement (see Table D.1 Feature Library).

The feature parameters are presented in Table D.1 Feature Library of Appendix D Feature Library.

3.3.3 Feature Creation

Feature creation includes two separate operations as seen in Figure 3.11; (1) Geometry creation, (2) Physical file creation. In this section mainly, geometry creation is discussed, in the basis of parameters belonging to the features and sample calculations are presented for the visualization via SolidWorks application. Physical file creation is discussed in section 3.4 'Feature Modeler Preprocessor', with the additional discussion about STEP Part 21 physical file.

In the light of the fact that feature library consists of rotational features extracted from AP224, the simple and commonly accepted method in CAD; revolution of a section 360° about an axis, can be used for generation of 3D model in SolidWorks application (Figure 3.12).

For demonstration of geometry creation technique, sample calculations of obtaining the sketch to revolve for feature generation are presented for the features used in 3.2.2 Extraction of Features from EXPRESS Specification of APs. Also, complex calculations for generating transition features are presented.

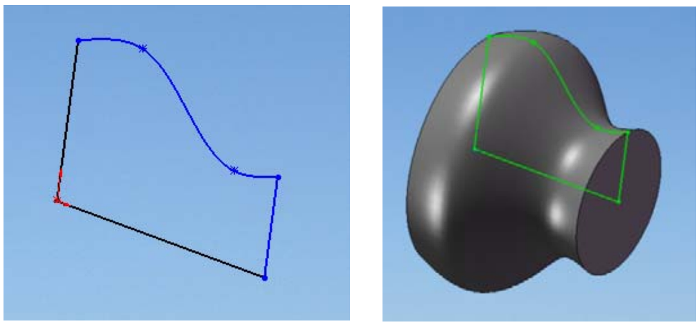


Figure 3.12. Revolution of a section 360° about an axis

<u>Outer diameter</u>

As stated in 3.2.2 Extraction of Features from EXPRESS Specification of APs, due to the optional parameter 'taper_select' in EXPRESS Specification of '*outer_diameter*' feature, it is possible to create three different variations:

- Straight outer diameter
- Tapered outer diameter with decreasing diameter
- Tapered outer diameter with increasing diameter

Optional '*taper_select*' parameter gives opportunity to select among:

- final diameter
- taper angle

for specifying taper amount. So for a tapered outer diameter feature, either with decreasing or increasing diameter, definition can be completed by specifying one of final diameter or taper angle, additional to the parameters of straight outer diameter; diameter and length. Please note that when using 'taper angle' as the third parameter, sign notation for the angle is very important in calculations:

- for decreasing taper: α is <u>negative</u> and the value is between 0° and 90°.
- for increasing taper: α is <u>positive</u> and the value is between 0° and 90°.

In any case for '*outer_diameter*' four lines formed by four points are required to form the sketch needed for revolution (see 0 on page 60). Being aware of the fact that, first point 'P_1' is on the z-axis (meaning P_{1x}=0) and P_{1z}= z_pos (z_pos is a public variable used for positioning the feature on z-axis); insertion points can be calculated.

Additional to the two parameters; diameter (D) and length (L), common for each variation: specifying final diameter (D_F), either directly or calculating using taper angle (α), enables calculation of points.

$$D_F = D + 2.sin(\alpha) \tag{3.1}$$

Then;

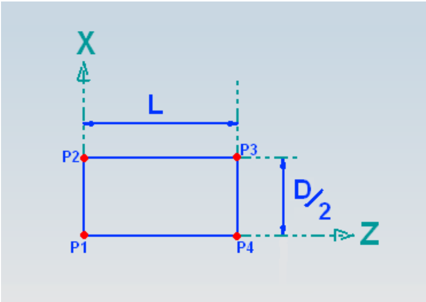
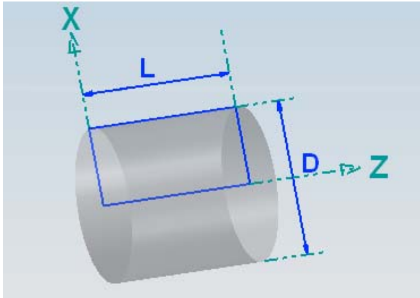
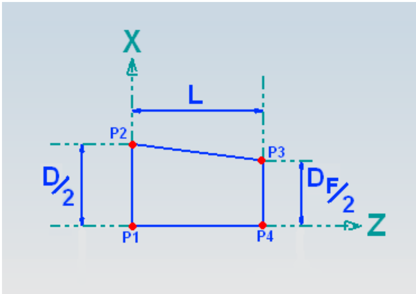
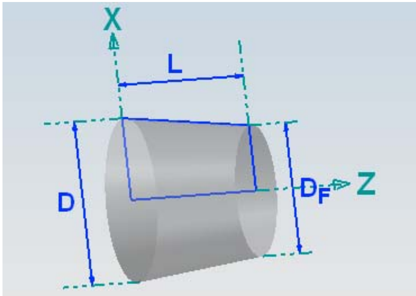
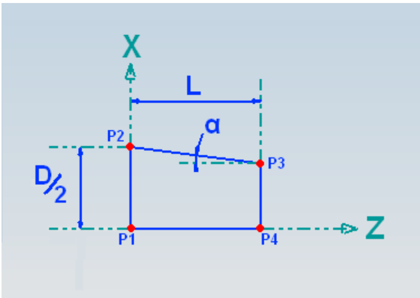
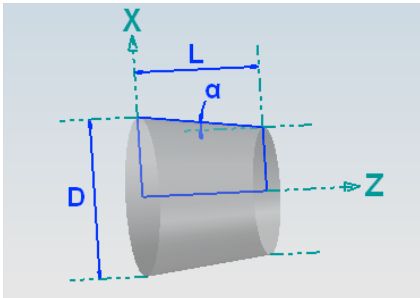
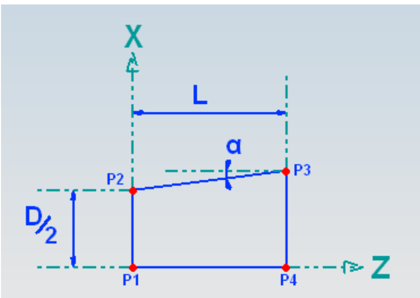
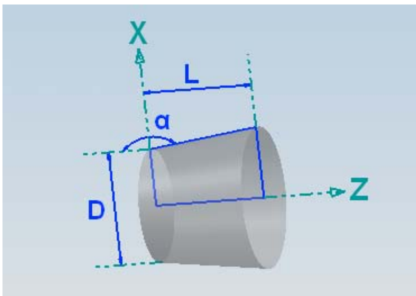
$$P_{1x} = 0 \quad ; \quad P_{1z} = z\_pos \tag{3.2}$$

$$P_{2x} = {}^{D}/_{2} \quad ; \quad P_{2z} = z\_pos \tag{3.3}$$

$$P_{3x} = {}^{D_F}/_{2} \quad ; \quad P_{3z} = z_{pos} + L \tag{3.4}$$

$$P_{4x} = 0 \quad ; \quad P_{4z} = z_{pos} + L \tag{3.5}$$

Table 3.1. Sketch and Revolution for '*Outer_diameter*'

Sketch	Revolution
	
	
	
	

Outer_diameter_to_shoulder

Five lines formed by five points are required to form the sketch needed for revolution (see Table 3.2 on page 63). Being aware of the fact that, first point 'P_1' is on the z-axis (meaning P_{1x}=0) and P_{1z}= z_pos (z_pos is a public variable used for positioning the feature on z-axis); insertion points can be calculated.

Parameters, extracted from EXPRESS Specification of AP224 are: radius at placement (R), length (L), first length (L_1), second length (L_2), tilt angle (α), profile angle (β) and profile radius (r).

However, the two parameters; L_1 and L_2 are not easy to use for the user so these two parameters are preferred to be replaced by diameter (D) and final diameter (D_F) only in the user forms to serve for the user friendly design. D and D_F can be easily converted to L_1 and L_2 by simple equations (Figure 3.13).

$$L_1 = \frac{D - R}{2.tan(\pi - \alpha - \beta)} \quad ; \quad L_2 = \frac{D_F - R}{2.tan(\beta)} \tag{3.6}$$

where;

$$L = L_1 + L_2 \tag{3.7}$$

Noting that, L and D_F used alternatives for each other, since using two of them at the same time results ambiguity.

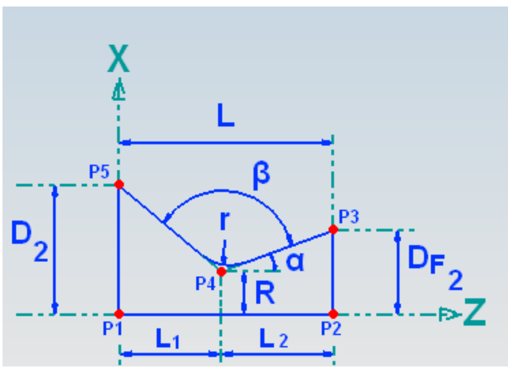
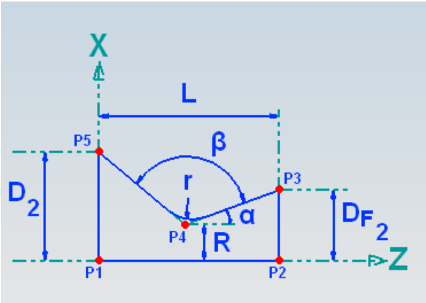
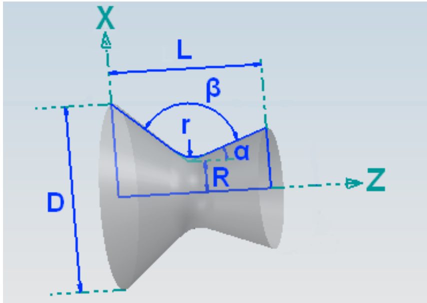


Figure 3.13. Variables associated to '*Outer_diameter_to_shoulder*'

Table 3.2. Sketch and Revolution for '*Outer_diameter_to_shoulder*'

Sketch	Revolution
	

If D_F is given, L can be found using;

$$L = \left(\frac{D - R}{2.\,tan(\pi - \alpha - \beta)}\right) + \left(\frac{D_F - R}{2.\,tan(\beta)}\right) \qquad (3.8)$$

If L is given, D_F can be found using;

$$D_F = R + 2\left(L - \frac{D - R}{2.\,tan(\pi - \alpha - \beta)}\right).\tan(\beta) \qquad (3.9)$$

Then;

$$P_{1x} = 0 \quad ; \quad P_{1z} = z\_pos \qquad (3.10)$$

$$P_{2x} = 0 \quad ; \quad P_{2z} = z_{pos} + L \qquad (3.11)$$

$$P_{3x} = {}^{D_F}/_{2} \quad ; \quad P_{3z} = z_{pos} + L \qquad (3.12)$$

$$P_{4x} = R \quad ; \quad P_{4z} = z_{pos} + L_1 \qquad (3.13)$$

$$P_{5x} = {}^{D}/_{2} \quad ; \quad P_{5z} = z\_pos \qquad (3.14)$$

Transition Features

'*edge round*', '*fillet*' and '*chamfer*' are the transition features.

'*edge round*' and '*fillet*' shares the same parameters; (1) first feature, (2) second feature and (3) radius: but they are different from each other in creation method. Round is a subtraction from the existing geometry and belonging arc piece is convex, while fillet is addition to the existing geometry and belonging arc piece is concave.

'*chamfer*' may be a subtraction of union operation depending on the parameters of features on which chamfer is attached. Parameters for chamfer are; (1) first feature, (2) second feature, (3) first offset, (4) second offset or chamfer angle.

For any transition feature, the feature modeler decides whether the transition feature is a subtraction or union operation and edge round or fillet.

Sketch geometry for transition features is dependent to the parameters of features on which transition features are attached. This makes the calculations a bit complicated, long and sometimes requires employing numerical methods.

As stated the calculations of transition features depends on the parameters and characteristics of the features on which a transition feature is attached. So features are classified according to their similar characteristics and a variety of combinations are evaluated for transition features and summarized as a table (see 0 on page 66 and Table 3.6 on page 72).

Edge round and fillet

Two lines and an arc formed by four points are required to form the sketch needed for revolution (see 0 on page 65). The parameters for '*edge round*' and '*fillet*' are first feature, second feature and radius (r), but only 'r' is used in the calculations.

Table 3.3. Sketch and Revolution for '*edge_round*' and '*fillet*'

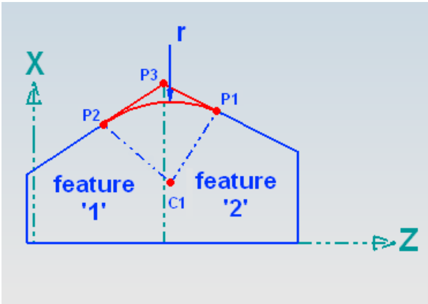
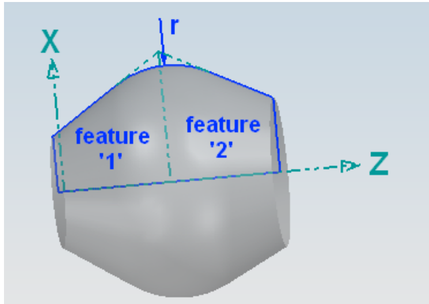
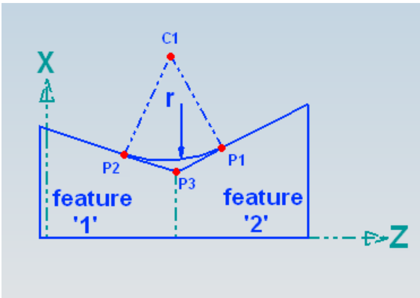
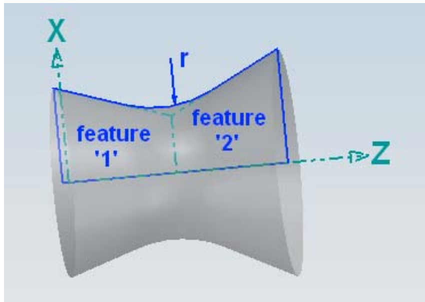
Sketch	Revolution
	
	

Table 3.4. Round and Fillet Summary

2nd feat \ 1st feat			TYPE 1 (OD; ODT; ODSHLD; REVFLT)	TYPE 2 (REVRND; CAP)
EDGE 1			**CASE p1** Df=D(i+1), z_pos_end=z_pos(i+1), $\alpha_{i1}=\pi/2$ make feature second feature ROUND	**CASE p2** Df=D(i+1), z_pos_end=z_pos(i+1), $\alpha_{i1}=\pi/2$ OR very near to $\pi/2$; make feature second feature ROUND
EDGE 2			**CASE p1** $\alpha_{i2}=-\pi/2$ ROUND	**CASE p3** $\alpha_{i2}=-\pi/2$ OR very near to $-\pi/2$ IF feat1 CAP NO EDGE2 ROUND
TO FEATURE	TYPE 1	Df > D (i+1)	**CASE p1** $\alpha_{i1}=-\pi/2$ FILLET	**CASE p1** $\alpha_{i1}=-\pi/2$ FILLET
TO FEATURE	TYPE 1	Df = D (i+1)	**CASE p1** IF feat1 OD $\alpha_{i1}=0$ IF feat 2 OD $\alpha_{i2}=0$ IF $\alpha_{i1}=\alpha_{i2}$ NO RADIUS	**CASE p3** IF feat 2 OD $\alpha_{i2}=0$ IF $\alpha_{i1}=\alpha_{i2}$ NO RADIUS
TO FEATURE	TYPE 1	Df < D (i+1)	**CASE p1** $\alpha_{i2}=\pi/2$ FILLET	**CASE p3** $\alpha_{i2}=\pi/2$ OR very near to $\pi/2$ FILLET
TO FEATURE	TYPE 2	Df > D (i+1)	**CASE p2** $\alpha_{i1}=-\pi/2$ FILLET	**CASE p2** $\alpha_{i1}=-\pi/2$ FILLET
TO FEATURE	TYPE 2	Df = D (i+1)	**CASE p2** IF feat1 OD $\alpha_{i1}=0$ NO RADIUS	**CASE p4**
TO FEATURE	TYPE 2	Df < D (i+1)	**CASE p1** $\alpha_{i2}=\pi/2$ FILLET	**CASE p3** $\alpha_{i2}=\pi/2$ OR very near to $\pi/2$ FILLET

For the following calculations, parameters other than 'r' belongs to the first feature, where a parameters belonging to the second feature is needed the parameters are identified by subscript notation as '1' and '2'.

CASE p1:

$$P_{2x} = \frac{D_F}{2} - r.\left[\cos(\alpha_{i1}) - \frac{\cos\left(\frac{\alpha_{i2} + \alpha_{i1}}{2}\right)}{\cos\left(\frac{\alpha_{i2} - \alpha_{i1}}{2}\right)}\right].sgn(\alpha_{i2} - \alpha_{i1}) \tag{3.15}$$

$$P_{2z} = z\_pos\_end + r.\left[\sin(\alpha_{i1}) - \frac{\sin\left(\frac{\alpha_{i2} + \alpha_{i1}}{2}\right)}{\cos\left(\frac{\alpha_{i2} - \alpha_{i1}}{2}\right)}\right].sgn(\alpha_{i2} - \alpha_{i1}) \tag{3.16}$$

(3.17)

$$P_{1x=}P_{2x} + r.\left(cos(\alpha_{i1}) - cos(\alpha_{i2})\right).sgn(\alpha_{i2} - \alpha_{i1}) \tag{3.18}$$

(3.19)

$$P_{1z=}P_{2z} - r.\left(sin(\alpha_{i1}) - sin(\alpha_{i2})\right).sgn(\alpha_{i2} - \alpha_{i1}) \tag{3.20}$$

(3.21)

$$C_{1x=}P_{1x} + r.cos(\alpha_{i2}).sgn(\alpha_{i2} - \alpha_{i1}) \tag{3.22}$$

$$C_{1z=}P_{1z} - r.sin(\alpha_{i2}).sgn(\alpha_{i2} - \alpha_{i1}) \tag{3.23}$$

$$P_{3x} = {}^{D_F}/_{2} \qquad ; \qquad P_{3z} = z\_pos\_end \tag{3.24}$$

P_{3x} and P_{3x} are same for all cases so calculation is not repeated from this point forward.

CASE p2:

$$A = \left[\frac{-r.\sqrt{\left[1+\left(tan(\alpha_{i1})\right)^2\right]}.sgn(\alpha_{i1}) + R}{R - r.sgn(\alpha_{i1})} \right] \tag{3.25}$$

$$a = 1 + (tan(\alpha i1\ ))^2 \tag{3.26}$$

$$b = 2.A.tan(\alpha i1\ ) \tag{3.27}$$

$$c = A^2 - 1 \tag{3.28}$$

$$\Delta = b^2 - 4.a.c \tag{3.29}$$

$$\beta_1 = asin\left(\frac{-b-\sqrt{\Delta}}{2.a}\right) \tag{3.30}$$

$$\beta_2 = asin\left(\frac{-b+\sqrt{\Delta}}{2.a}\right) \tag{3.31}$$

$$\beta = if[(\alpha_{i1}) > 0, if[(\beta_1 > 0), \beta_1, \beta_2], if(\beta_1 < |\alpha_{i1}|, \beta_1, \beta_2)] \tag{3.32}$$

$$P_{1x=}\frac{D_F}{2} - R.\left(1 - cos(\beta)\right) \tag{3.33}$$

$$P_{1z} = z\_pos\_end + R.sin(\beta) \tag{3.34}$$

$$P_{2x=}P_{1x} - r.\left(cos(\beta) - cos(\alpha_{i1})\right).sgn(\alpha_{i1}) \tag{3.35}$$

$$P_{2z=}P_{1z} - r.\left(sin(\beta) - sin(\alpha_{i1})\right).sgn(\alpha_{i1}) \tag{3.36}$$

$$C_{1x=}P_{1x} - r.cos(\beta).sgn(\alpha_{i1}) \tag{3.37}$$

CASE p3:

$$z\_pos\_end = R.sin(\alpha_{i1}) \tag{3.38}$$

$$A = \left[\frac{r.\sqrt{\left[1+\left(tan(\alpha_{i2})\right)^2\right]}.sgn(\alpha_{i1}+\alpha_{i2}) + R.\left(cos(\alpha_{i1}) - tan(\alpha_{i2}).sin(\alpha_{i1})\right)}{R + r.sgn(\alpha_{i1}-\alpha_{i2})}\right] \tag{3.39}$$

$$a = 1 + (tan(\alpha i1\ )\ )^2 \tag{3.40}$$

$$b = 2.A.tan(\alpha i1\ ) \tag{3.41}$$

$$c = A^2 - 1 \tag{3.42}$$

$$\Delta = b^2 - 4.a.c \tag{3.43}$$

$$\beta_1 = asin\left(\frac{-b-\sqrt{\Delta}}{2.a}\right) \tag{3.44}$$

$$\beta_2 = asin\left(\frac{-b+\sqrt{\Delta}}{2.a}\right) \tag{3.45}$$

$$\beta = if[(\alpha_{i1}) > 0, if[(\beta_1 > 0), \beta_1, \beta_2], if(\beta_1 < |\alpha_{i1}|, \beta_1, \beta_2)] \tag{3.46}$$

$$C_{1x=}\frac{D_F}{2} - R.cos(\alpha_{i1}) + \left(R + r.sgn(\alpha_{i1}+\alpha_{i2}).cos(\beta)\right) \tag{3.47}$$

$$C_{1z=}\ z\_pos\_end - R.sin(\alpha_{i1}) + \left(R + r.sgn(\alpha_{i1}+\alpha_{i2}).sin(\beta)\right) \tag{3.48}$$

$$P_{2x=}\frac{D_F}{2} - R.\left(cos(\alpha_{i1}) - cos(\beta)\right) \tag{3.49}$$

$$P_{2z=}\ z\_pos\_end - R.\left(sin(\alpha_{i1}) - sin(\beta)\right) \tag{3.50}$$

$$P_{1x=}C_{1x-}\ r.cos(\alpha_{i2}).sgn(\alpha_{i1}+\alpha_{i2}) \tag{3.51}$$

$$P_{1z=}C_{1z+}\ r.sin(\alpha_{i2}).sgn(\alpha_{i1}+\alpha_{i2}) \tag{3.52}$$

CASE p4:

$$z\_pos\_end = R_{i1}.sin(\alpha_{i1}) \tag{3.53}$$

$$D_F = 2.\left(\frac{D}{2} - R_{i1}cos(\alpha_{i1})\right) \tag{3.54}$$

$$\gamma = atan\left(\frac{R_{i1}.cos(\alpha_{i1}) - R_{i2}}{R_{i1}.sin(\alpha_{i1})}\right) \tag{3.55}$$

$$A = \sqrt{\left(R_{i2} - R_{i1}.cos(\alpha_{i1})\right)^2 + \left(-R_{i1}.sin(\alpha_{i1})\right)^2} \tag{3.56}$$

$$\theta = acos\left[\frac{A^2 + (R_{i2} + r)^2 - (R_{i1} + r)^2}{2.A.(R_{i2} + r)}\right] \tag{3.57}$$

$$\tau = acos\left[\frac{A^2 + (R_{i1} + r)^2 - (R_{i2} + r)^2}{2.A.(R_{i1} + r)}\right] \tag{3.58}$$

$$\beta_{i1=}\frac{\pi}{2} - (\tau + \gamma) \tag{3.59}$$

$$\beta_{i2=}(\theta - \gamma) - \frac{\pi}{2} \tag{3.60}$$

$$P_{1x=}\frac{D_F}{2} - R_{i2}.\left(1 - cos(\beta_{i2})\right) \tag{3.61}$$

$$P_{2x=}\frac{D_F}{2} + R_{i1}.\left(cos(\beta_{i1}) - cos(\alpha_{i1})\right) \tag{3.62}$$

$$C_{1x=}P_{1x+}\ r.cos(\beta_{i2}) \tag{3.63}$$

$$P_{2z=}\ z\_pos\_end{+}R_{i2}.sin(\beta_{i1}) \tag{3.64}$$

$$P_{2z=}\ z\_pos\_end + R_{i1}.\left(sin(\beta_{i1}) - sin(\alpha_{i1})\right) \tag{3.65}$$

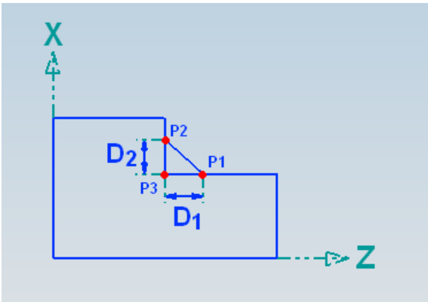
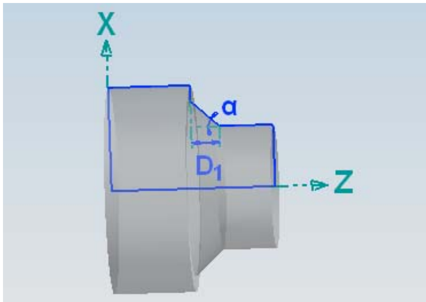
$$C_{1z} = P_{1z} + r.sin(\beta_{i2}) \tag{3.66}$$

Chamfer

Three lines formed by three points are required to form the sketch needed for revolution (see Table 3.5 on page 71). The parameters for '*chamfer*' are first feature, second feature, first offset (D_1), second offset (D_2) and chamfer angle (α). 'D_2' and 'α' are alternatives to each other. 'D_1' , 'D_2' and 'α' are used in the calculations.

For the following calculations, parameters other than 'r' belongs to the first feature, where a parameters belonging to the second feature is needed the parameters are identified by subscript notation as '1' and '2'. Only CASE c1 will be presented as sample calculation for chamfer.

Table 3.5. Sketch and Revolution for '*chamfer*'

Sketch	Revolution
	

Only case c1 of Table 3.6 is presented as sample calculation.

CASE c1:

If α is given as the second parameter first D_2 is calculated:

$$\alpha = sgn(\alpha_{i2} - \alpha_{i1}).\alpha \tag{3.67}$$

$$\beta = \alpha_{i2} - \alpha_{i1} - \alpha \tag{3.68}$$

$$D_2 = \frac{D_1.sgn(\alpha_{i2} - \alpha_{i1})\left(tan(\beta - \alpha_{i2}).cos(\alpha_{i1}) + sin(\alpha_{i1})\right)}{sgn(\alpha).\left(tan(\beta - \alpha_{i2}).cos(\alpha_{i2}) + sin(\alpha_{i2})\right)} \tag{3.69}$$

Table 3.6. Chamfer Summary

1st feat / 2nd feat			**TYPE 1** (OD; ODT; ODSHLD; REVFLT)	**TYPE 2** (REVRND; CAP)
EDGE 1			**CASE c1** Df=D(i+1), z_pos_end=z_pos(i+1), $\alpha_{i1}=\pi/2$ make feature second feature	**CASE c2** Df=D(i+1), z_pos_end=z_pos(i+1), $\alpha_{i1}=\pi/2$ OR very near to $\pi/2$; make feature second feature
EDGE 2			**CASE c1** $\alpha_{i2}=-\pi/2$	**CASE c3** $\alpha_{i2}=-\pi/2$ OR very near to $-\pi/2$ IF feat1 CAP NO EDGE2
TO FEATURE	**TYPE 1**	**Df > D (i+1)**	**CASE c1** $\alpha_{i1}=-\pi/2$	**CASE c1** $\alpha_{i1}=-\pi/2$
		Df = D (i+1)	**CASE c1** IF feat1 OD $\alpha_{i1}=0$ IF feat 2 OD $\alpha_{i2}=0$ IF $\alpha_{i1}=\alpha_{i2}$ NO CHAMFER	**CASE c3** IF feat 2 OD $\alpha_{i2}=0$ IF $\alpha_{i1}=\alpha_{i2}$ NO CHAMFER
		Df < D (i+1)	**CASE c1** $\alpha_{i2}=\pi/2$	**CASE c3** $\alpha_{i2}=\pi/2$ OR very near to $\pi/2$
	TYPE 2	**Df > D (i+1)**	**CASE c2** $\alpha_{i1}=-\pi/2$	**CASE c2** $\alpha_{i1}=-\pi/2$
		Df = D (i+1)	**CASE c2** IF feat1 OD $\alpha_{i1}=0$ NO CHAMFER	**CASE c4**
		Df < D (i+1)	**CASE c1** $\alpha_{i2}=\pi/2$	**CASE c3** $\alpha_{i2}=\pi/2$ OR very near to $\pi/2$

$$P_{1x=}\frac{D_F}{2}+D_2.sin(\alpha_{i2}) \quad ; \quad P_{1z=}\ z\_pos\_end+\ D_2.cos(\alpha_{i2}) \tag{3.70}$$

$$P_{2x=}\frac{D_F}{2}-D_1.sin(\alpha_{i1}) \quad ; \quad P_{2z=}\ z\_pos\_end-\ D_1.cos(\alpha_{i1}) \tag{3.71}$$

$$P_{3x=}0.5.D_F \quad ; \quad P_{3z=}\ z\_pos\_end \tag{3.72}$$

If D_2 is given as the second parameter the calculations presented can be directly done, and α can be calculated:

$$\beta = atan\left(\frac{-D_1.sin(\alpha_{i1})-D_2.sin(\alpha_{i2})}{D_1.cos(\alpha_{i1})+D_2.cos(\alpha_{i2})}\right)+\alpha_{i2} \tag{3.73}$$

$$\alpha = \alpha_{i2}-\alpha_{i1}-\beta \tag{3.74}$$

3.3.4 Error Handling and 'UNDO' Option

Error handling capability of a system is very crucial, besides being powerful, user friendly and having an interactive environment. Error handling in practice requires; not allowing improper and faulty data input and warning user by error messages. For the sake of uninterrupted design process, feature modeler module is enhanced with a series of error handling algorithms and error messages. Some cases are presented to give an idea about error handling capability of the developed system and visual display of error messages are illustrated in Appendix C.

Possible errors during design stage and handling methods are:

- Many parameters such as diameter, length, width must be <u>positive</u> to create a feature: if zero or negative number is input by the user, the system does not accept the input and warn the user by an explanatory error message.
- Taper angle for a decreasing diameter should be <u>negative</u>: if zero or positive number is input by the user, the system does not accept the input and warn the user by an explanatory error message. This is vice versa for an increasing diameter.

- For the real numbers input as feature parameters, VB 6.0 and STEP Part 21 physical file accept separation of decimal part using a dot '.' Instead of a comma ',': if a real number not obeying this rule is input by the user, the system automatically corrects the number format without any error message, interrupting the user.
- Even the set of parameters input to define the feature are proper individually, in sketch plane the resulting geometry may be erroneous: if a set of numbers resulting erroneous geometry is input by the user, the system does not accept the input and warn the user by an explanatory error message.
- If a parameter or feature definition data is not input by the user, the system does not continue modeling and warn the user by an explanatory error message. For example, tolerance data must be input for the specified parameters of a feature.
- The radius value for '*edge_round*' and '*fillet*' type transition features has a maximum value determined by the parameters of features on which transition features are attached: if a radius value higher than the allowed maximum is input by the user, the system does not accept the input and warn the user by an explanatory error message. Furthermore, the system evaluates the maximum radius value the for the transition feature and displays on the user form, just after selection of the first and second features to create a transition, to prevent user from consuming time with trial and error.

Listed cases are most but not all of the cases for error handling. Although almost all of the error cases are handled, to give the user opportunity to trial and error or previewing, an UNDO option is available for going back in design step by step. However coding for the UNDO option is long, the idea behind is quite simple: The last sketch and its revolution are deleted in SolidWorks application and the lines inserted to the physical file representing the last feature are erased.

3.4 Feature Modeler Preprocessor

The preprocessor responsible for creating STEP physical file, is embedded to the feature modeler and generates the feature representing segment simultaneously with the feature creation.

In order to present the nature of preprocessing, structure of the STEP physical file will be described first. Then, representation of design data in STEP Part 21 format will be illustrated for the features used as example up to this point, for the sake of integrity and completeness.

3.4.1 STEP Part 21 Physical File

This Part of ISO 10303-21 specifies an exchange structure format using a clear text encoding for product data of which the schema is specified in the EXPRESS language (ISO 10303-11). The file format is suitable for the transfer of product data among computer systems. The mapping from the EXPRESS language to the syntax of the exchange structure is specified. Any EXPRESS schema can be mapped onto the exchange structure syntax (Amaitik, 2003).

General Structure

The exchange structure shall be a sequential file using a clear text encoding. The exchange structure shall consist of two sections: the Header section and the Data section. The Header section provides data relating to the exchange structure itself. The Data section provides the data to be transferred. The exchange structure shall begin with a special token "ISO-10303-21;" followed by the Header section and the Data section. Immediately after the Data section, the exchange structure shall be terminated by a special token "END-ISO-10303-21;".

The header section shall appear exactly once in the exchange structure and shall be the first section in the exchange structure.

The data section is a collection of instances of the entities specified in the schemas identified in the Header section. This section shall appear exactly once in the exchange structure.

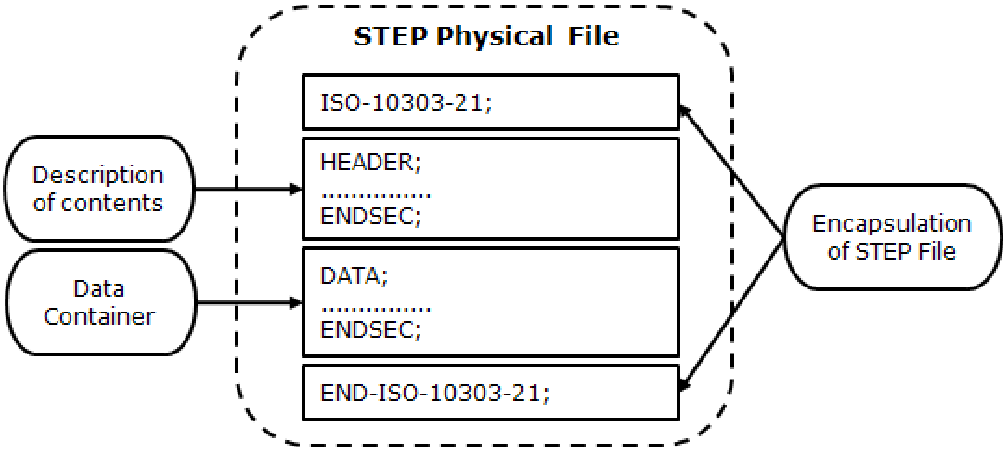


Figure 3.14. STEP Physical File Structure

Data Section

The Data section contains the product data to be transferred by the Exchange structure. The Data section contains instances of entities that correspond to the EXPRESS schemas governing the exchange structure as specified in the Header section. The data section shall begin with the special token "DATA;" and shall be followed by entity instances. The data section shall be terminated with the special token "ENDSEC;".

Mapping from EXPRESS to the Physical File

All the definitions and parameters for features extracted from AP224 using EXPRESS Specifications are mapped to the physical file in a series of rules defined in ISO-10303-11, to represent the product data.

The following guidelines are used when mapping from EXPRESS to exchange structure:

- The attributes will be mapped in the order defined in the Entity.
- Inherited attributes are mapped prior to the attributes of the actual entity.
- Repeated inheritance of attributes does not create multiple mapping.

- Attributes without values are mapped as '$'.
- INVERSE and DRIVE attributes are not mapped.

An example of mapping an entity definition in EXPRESS to Physical File data section as an instance, containing various data types is given to guide the implementation.

Entity definition in EXPRESS:

```
ENTITY widget;
    i1: INTEGER; -----------> A
    i2: INTEGER; -----------> B
    s1: STRING(3); -----------> C
    s2: STRING; -----------> D
    l : LOGICAL; -----------> E
    b : BOOLEAN; -----------> F
    r1: REAL(4); -----------> G
    r2: REAL; -----------> H
END_ENTITY;
```

Sample entity instance in Data section:

```
#2=WIDGET(99, 99999, 'ABC', 'ABCDEFG', .T., .F., 9.000, 1.2345);
          ^    ^       ^       ^         ^   ^    ^      ^
          |    |       |       |         |   |    |      |
          A    B       C       D         E   F    G      H
```

A: i1 has a value of 99 in this entity instance.

B: i2 has a value of 99999 in this entity instance.

C: s1 has a value of 'ABC' in this entity instance. This value falls within the range (3 characters) specified for this attribute.

D: s2 has a value of 'ABCDEFG' in this entity instance.

E: l has a value of TRUE in this entity instance.

F: b has a value of FALSE in this entity instance.

G: r1 has a value of 9.000 in this entity instance. This value falls within the precision (4 decimal digits in the fractional part of a number) specified for this attribute.

H: r2 has a value of 1.2345 in this entity instance.

3.4.2 Physical File ‘Data’ Instances

Instances of the data sections in physical file for features are created using the mapping techniques are presented.

Outer diameter

For straight outer diameter:

```
#10=OUTER_DIAMETER(ID_feature,#11,#12,#13,$);
#11=ORIENTATION((x_pos,y_pos,z_pos),(xi_dir,xj_dir,xk_dir,yi_dir,yj_di
r,yk_dir,zi_dir,zj_dir,zk_dir));
#12=NUMERIC_PARAMETER(‘DIAMETER’,unit_length,D,#14);
#13=NUMERIC_PARAMETER(‘LENGTH’,unit_length,Length,#15);
#14=TOLERANCE_RANGE(upper_range,lower_range,significant_digits);
#15=PLUS_MINUS_VALUE(upper_limit,lower_limit,significant_digits);
```

For tapered outer diameter:

```
#10=OUTER_DIAMETER(ID_feature,#11,#12,#13,#14);
#11=ORIENTATION((x_pos,y_pos,z_pos),(xi_dir,xj_dir,xk_dir,yi_dir,yj_di
r,yk_dir,zi_dir,zj_dir,zk_dir));
#12=NUMERIC_PARAMETER(‘DIAMETER’,unit_length,D,#15);
#13=NUMERIC_PARAMETER(‘LENGTH’,unit_length,L,#16);
#14=DIAMETER_TAPER(#17);  (*diameter_taper*)
#15=TOLERANCE_RANGE(upper_range,lower_range,significant_digits);
#16=PLUS_MINUS_VALUE(upper_limit,lower_limit,significant_digits);
#17=NUMERIC_PARAMETER(‘FINAL DIAMETER’,unit_length,Df,#18);
#19=TOLERANCE_RANGE(upper_range,lower_range,significant_digits);
```

Outer diameter to shoulder

```
#10=OUTER_DIAMETER_TO_SHOULDER(ID_feature,#11,#12,#13,#14);
#11=ORIENTATION((x_pos,y_pos,z_pos),(xi_dir,xj_dir,xk_dir,yi_dir,yj_di
r,yk_dir,zi_dir,zj_dir,zk_dir));
#12=NUMERIC_PARAMETER(‘DIAMETER’,unit_length,D,#15);
#13=NUMERIC_PARAMETER(‘LENGTH’,unit_length,L,#16);
#14=VEE_PROFILE($,#17,#18,#19,#20,#21);
#15=TOLERANCE_RANGE(upper_range,lower_range,significant_digits);
#16=TOLERANCE_RANGE(upper_range,lower_range,significant_digits);
#17=NUMERIC_PARAMETER(‘PROFILE RADIUS’,unit_length,r,#22);
#18=NUMERIC_PARAMETER(‘PROFILE ANGLE’,unit_angle,α,#23);
#19=NUMERIC_PARAMETER(‘TILT ANGLE’,unit_angle,β,#24);
#20=NUMERIC_PARAMETER(‘FIRST LENGTH’,unit_length,L1,#25);
#21=NUMERIC_PARAMETER(‘SECOND LENGTH’,unit_length,L2,#26);
#22=PLUS_MINUS_VALUE(upper_limit,lower_limit,significant_digits);
#23=TOLERANCE_RANGE(upper_range,lower_range,significant_digits);
#24=TOLERANCE_RANGE(upper_range,lower_range,significant_digits);
```

```
#25=PLUS_MINUS_VALUE(upper_limit,lower_limit,significant_digits);
#26=TOLERANCE_RANGE(upper_range,lower_range,significant_digits);
```

3.5 Postprocessor

Postprocessor is used for both visualization of a product design representation in STEP Part 21 format in feature modeler module and input for NC-code generation module for machining file creation. This postprocessor uses the same feature library for both geometry generation and feature definitions.

Postprocessor is responsible for sequentially reading and decoding feature defining data instances of a STEP Part 21 physical file, then creating 3D instance of the product design by using the feature definitions of dll. Architecture of postprocessor is shown in Figure 3.15.

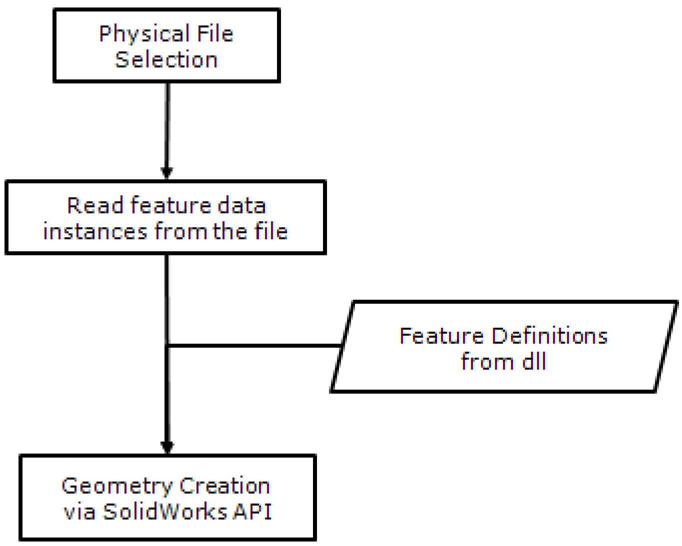


Figure 3.15. Postprocessor Architecture

3.6 NC-Code Generator

The last and main objective of this work is to obtain NC-code from the STEP AP224 features based Part 21 file. The NC-code generator module of the developed system achieves this. Imported product data using the postprocessor is processed with the guide of the embedded turning technology libraries, such as machine tool library, cutting tools and inserts library which are available for editing by the user. General architecture of NC-code generator module is illustrated in Figure 3.16.

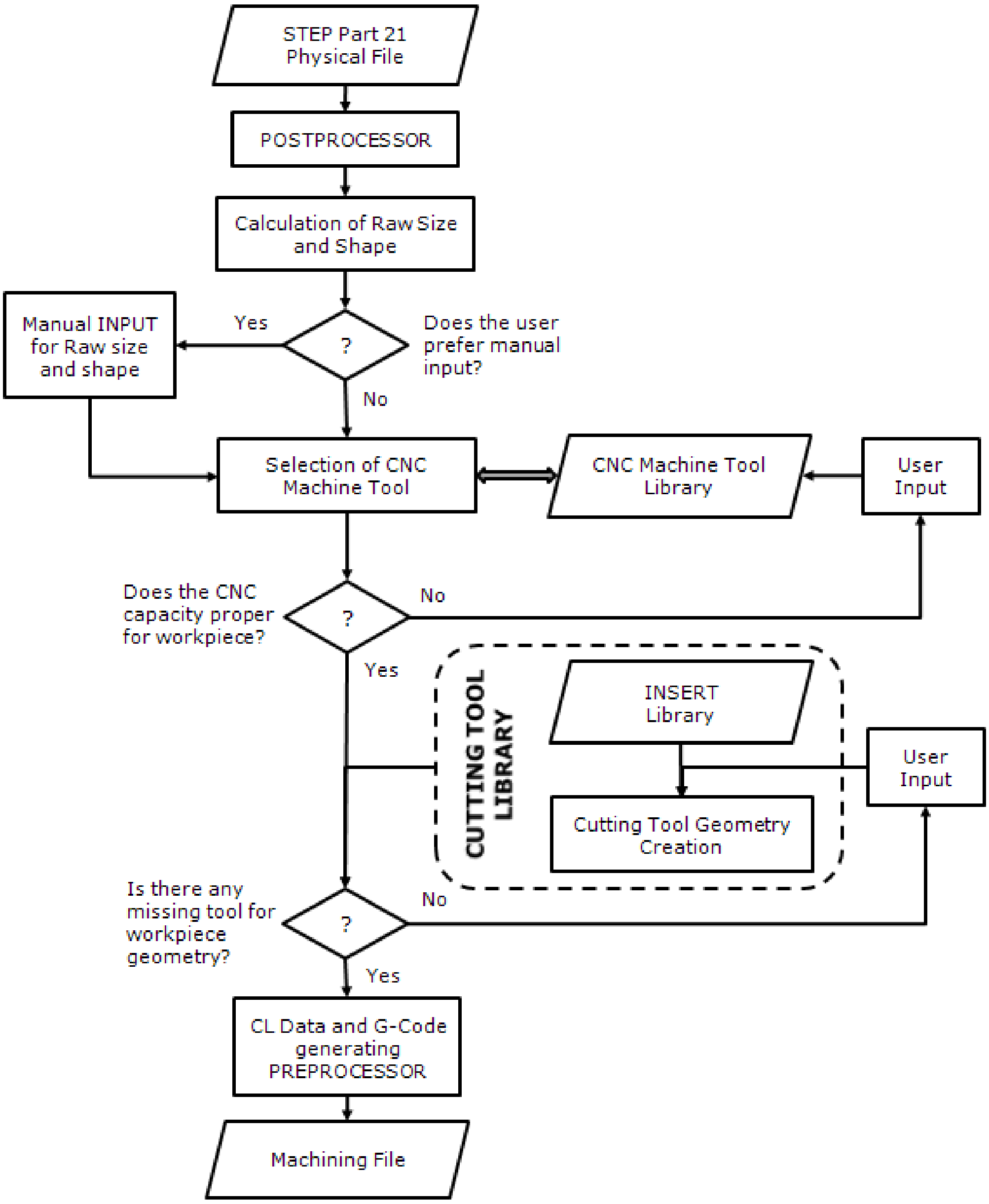


Figure 3.16. NC-code Generator Architecture

In this section, brief description about raw size and shape calculation for the product data (workpiece), setting CNC specifications for CNC machine tool library, technical data in cutting tools and insert library and creation of NC-code will be given and sample calculations will be presented where applicable.

3.6.2 Raw Size and Shape Calculation

Just after importing the design data machining module determines the needed raw shape and stock size for the product. Since two basic shapes, bar and hollow, are mostly preferred as raw material for turning applications the system selects one simply considering if the is a through hole on the product.

Sizing of the raw stock is done simply adding a roughing excess to the product in diameter direction and selecting the next oversize from the raw database of the system.

While calculating the raw stock length, the following are added to the product length:

- Roughing excess for facing
- Parting off excess
- Tool clearance from the chuck

3.6.3 CNC Specification

The system helps the user in foreseeing the producibility of the product with the machine tool available by inclusion of CNC machine tool library. The user is free to select any machine tool from this library or enter a new machine tool with its specifications. The data stored in this library are:

- CNC model
- CNC swing (chuck capacity for maximum workpiece diameter)
- Maximum workpiece length
- Rotational speed range
- Tool capacity of turret

3.6.4 Cutting Tool Library

This library is built to simulate the tooling capacity of the CNC turret. User is free to add a new tool to the library besides the preset ones. The parameters of the tool are used as a basis for NC-code generation, for instance a feed rate or spindle speed specified for a cutting tool, directly appears on the NC-code if the tool is employed for an operation. Furthermore the user is guided through data input by informative and cautionary error messages in the light of the machining technology information embedded inside the module.

The library combines the tool holder and inserts data to form a cutting tool and generates the cutting tools geometry for simulation purposes.

The data stored in this library are:

- Tool name
- Tool type (RHS, LHS, profiling, boring, drilling, etc.)
- Operation type (finishing, medium or roughing)
- Depth of cut (a_p)
- Feed rate (f)
- Spindle speed
- Side Cutting Edge Angle (κ_r)
- Tool length
- Tool width
- Attached insert type (square, triangular, rhombic 35°, etc.)
- Nose angle (ε)
- Cutting edge length (l)
- Effective cutting edge length (l_a)
- Nose radius

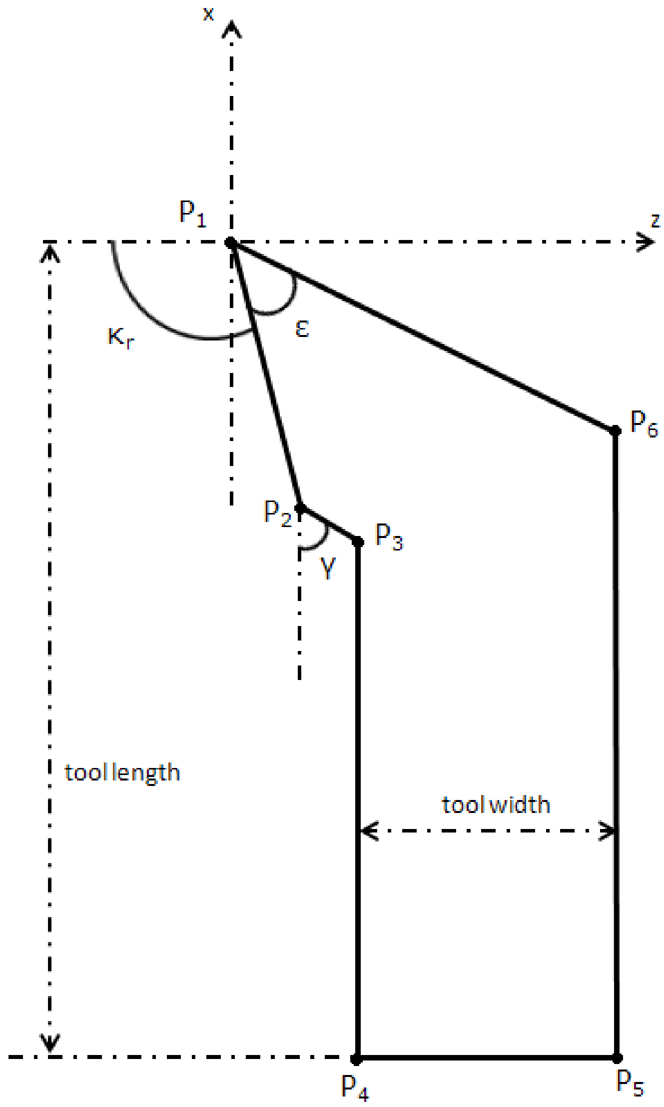


Figure 3.17. RHS External Turning Tool Geometry

Calculating Tool Geometry for External Turning Tool

$$\gamma = \varepsilon + \kappa_r - {}^{\pi}/_{2} \tag{3.75}$$

$$P_{1x} = 0 \tag{3.76}$$

$$P_{1z} = 0 \tag{3.77}$$

$$P_{2x} = P_{1x} - 2.l.cos\left(\kappa_r - {}^{\pi}/_{2}\right) \quad (3.78)$$

$$P_{2z} = P_{1z} - 2.l.\sin\left(\kappa_r - {}^{\pi}/_{2}\right).sgn(RHS) \quad (3.79)$$

$$P_{3x} = P_{2x} - l_a.cos(\gamma) \quad (3.80)$$

$$P_{3z} = P_{2z} - l_a.cos(\gamma).sgn(RHS) \quad (3.81)$$

$$P_{4x} = -Tool\ length \quad (3.82)$$

$$P_{4z} = P_{3z} \quad (3.83)$$

$$P_{5x} = P_{4x} \quad (3.84)$$

$$P_{5z} = P_{4z} + Tool\ width.sgn(RHS) \quad (3.85)$$

$$P_{6x} = sgn(RHS).(P_{1z} - P_{5z})/tan(\gamma) \quad (3.86)$$

$$P_{6z} = P_{5z} \quad (3.87)$$

Note that presented sample calculation is for RHS external turning tool with 'sgn(RHS)=1' and the same calculations are used for a LHS external turning tool with 'sgn(RHS)=-1'

3.6.5 NC-Code Generation

The main objective of this work is accomplished by this part of the system inside the NC-code generator module. After collecting all the data together, product design data, CNC specification and available cutting tool data; the order of turning process is determined and calculations of cutter location and stock removal cycles are held, finally the machining file is generated via a synchronous preprocessor.

Calculations to generate NC-code output for a facing process, external turning and grooving are presented here, for demonstration.

3.6.5.1 Calculations for Facing

The tool path for facing is shown in Figure 3.18 with the parameters used in calculations.

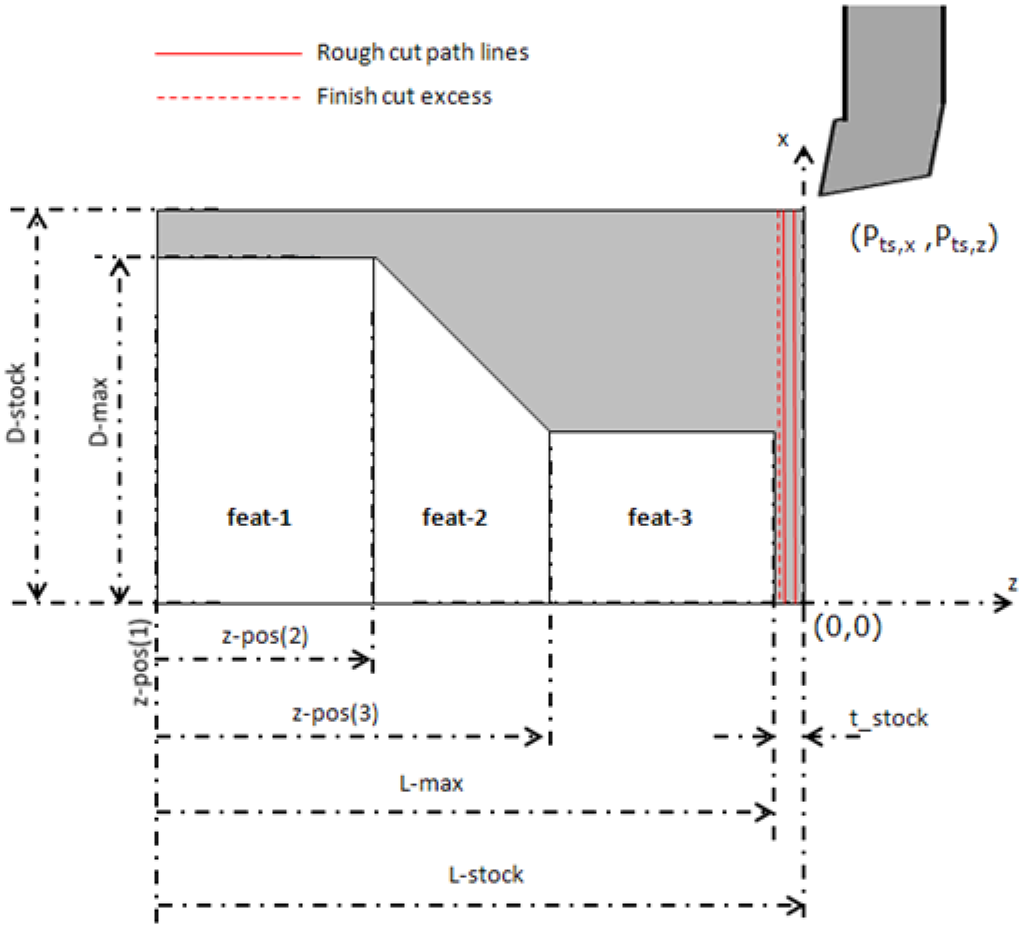


Figure 3.18. Facing Path Lines

$$t\_stock = L\_stock - L\_max \tag{3.88}$$

$$number\_of\_steps = \frac{t\_stock - a_{p-finish}}{a_{p-rough}} \tag{3.89}$$

$$number\_of\_steps = Integer(number\_of\_steps) \quad (3.90)$$

Round up number of steps to the nearest integer:

$$a_{p-rough} = \frac{t\_stock - a_{p-finish}}{number\ of\ steps} \quad (3.91)$$

Noting the tool start position is ($P_{ts,x}$, $P_{ts,z}$), the NC-code for facing is composed step by step:

- Call the cutting tool change procedure. VB code available for reference in Appendix A.5
- Calculate tool position:

$$turn\_current\_position_x = D\_stock + safe\_offset\_x \quad (3.92)$$

$$turn\_current\_position_z = L\_stock \quad (3.93)$$

- For ‘number_of_step’ times the following code repeats to complete roughing:
  - o Calculate new tool position:

$$turn\_current\_position_z = turn\_current\_position_z - a_{p\_rough} \quad (3.94)$$

  - o The tool rapidly moves to the new position:

    Nxx G00 X(turn_ current _position$_x$) Z(turn_ current _position$_z$)

  - o The tool moves and cuts to the center in the x-direction:

    Nxx G01 X-1 Z(turn_ current_position$_z$) F feed(turn.face_rgh_tl_no)

- The last cycle of roughing is same but the new z-position changes to leave material excess for just for finishing.

$$turn\_current\_position_z = L\_max + a_{p\_finish} \quad (3.95)$$

- Call the cutting tool change procedure for finishing operation, the tool goes to the starting position during tool change:

  Nxx G00 X(Pts,x) Z(Pts,z)

- Calculate new tool position:

$$turn\_current\_position_z = L\_max \tag{3.96}$$

- The tool rapidly moves to the new position:

  Nxx G00 X(turn_current_position$_x$) Z(turn_current_position$_z$)

- The tool moves and cuts to the center in the x-direction:

  Nxx G01 X-1 Z(turn_ current_position$_z$) F feed(turn.face_fin_tl_no)

The last step completes the facing operation; the next step will be obviously going back to starting position and change the tool for the new operation.

3.6.5.2 Calculations for External Turning

The tool path for roughing down to D_{max} is shown in Figure 3.19 with the parameters used in calculations.

$$t\_stock = \frac{(D_{stock} - D_{max})}{2} \tag{3.97}$$

$$number\_of\_steps = \frac{t\_stock - a_{p-finish}}{a_{p-rough}} \tag{3.98}$$

$$number\_of\_steps = Integer(number\_of\_steps) \tag{3.99}$$

Round up number of steps to the nearest integer:

$$a_{p-rough} = \frac{t\_stock - a_{p-finish}}{number\ of\ steps} \tag{3.100}$$

Noting the tool start position is ($P_{ts,x}$, $P_{ts,z}$), the NC-code for external turning is composed step by step:

- Call the cutting tool change procedure
- Calculate tool position:

$$turn\_current\_position_x = D_{stock}/2 \tag{3.101}$$

$$turn\_current\_position_z = L_{max} + safe\_offset\_z \tag{3.102}$$

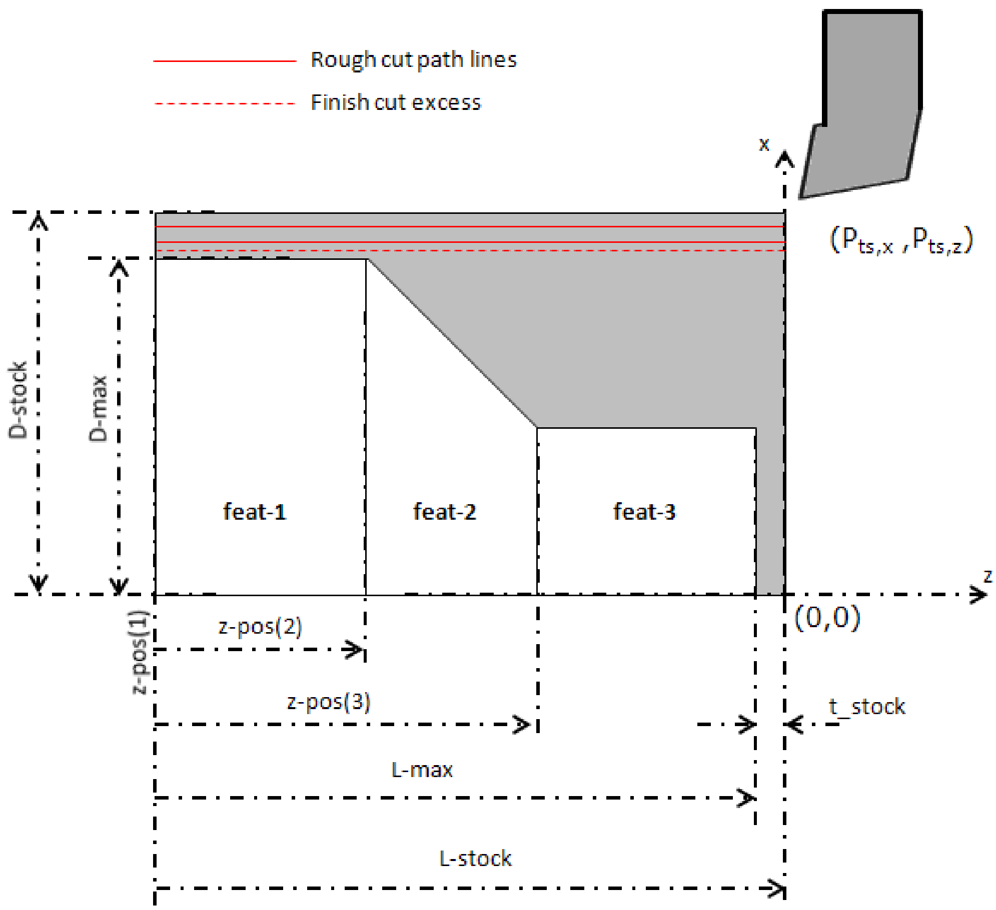


Figure 3.19. External turning roughing down to D_{max}

- For 'number_of_step' times the following code repeats to complete roughing:
  - Calculate new tool position:

$$turn\_current\_position_x = turn\_current\_position_x - a_{p\_rough} \quad (3.103)$$

  - The tool rapidly moves to the new position:

    Nxx G00 X(turn_ current _position$_x$) Z(turn_ current _position$_z$)

  - The tool moves and cuts up to the chuck clearance in the z-direction:

    Nxx G01 X-1 Z(turn_ current_position$_z$) F feed(turn.ext_rgh_tl_no)

- The last cycle of roughing down to D_{max} is same but the new z-position changes to leave material excess for just for finishing.

$$turn\_current\_position_x = D_{max}/2 + a_{p\_finish} \tag{3.104}$$

The external roughing operation does the same x-position calculations but the z-position should be determined considering the outer profile of the part. So the z-position calculations will be presented from this point on. The external turning path lines are presented in Figure 3.20.

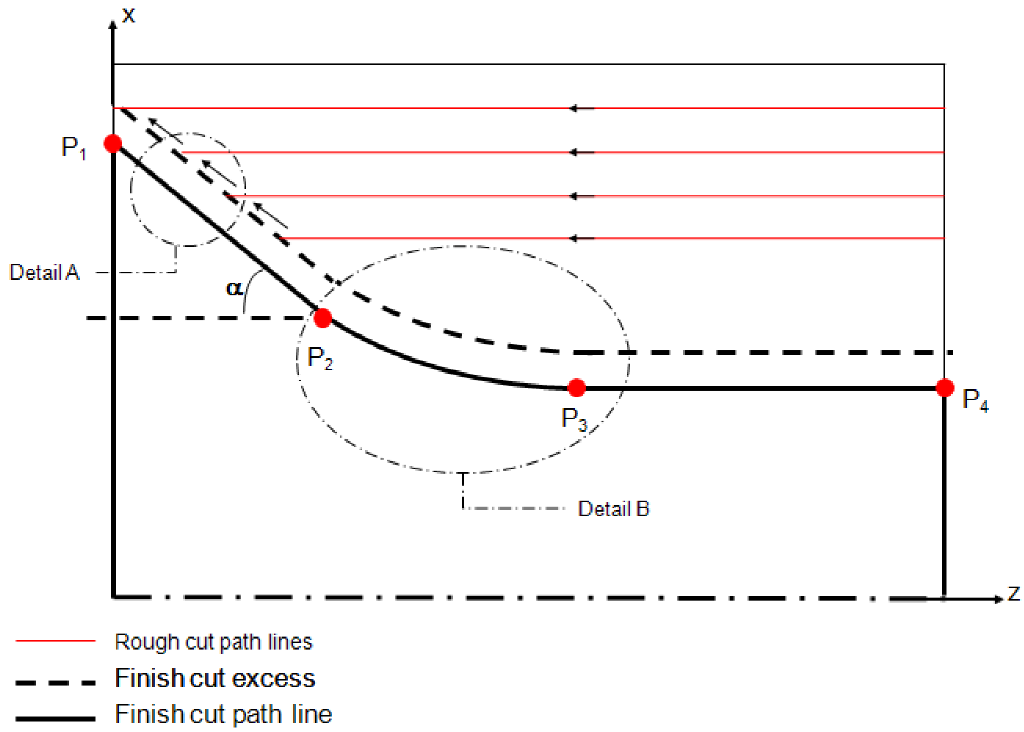


Figure 3.20. External Turning Path lines

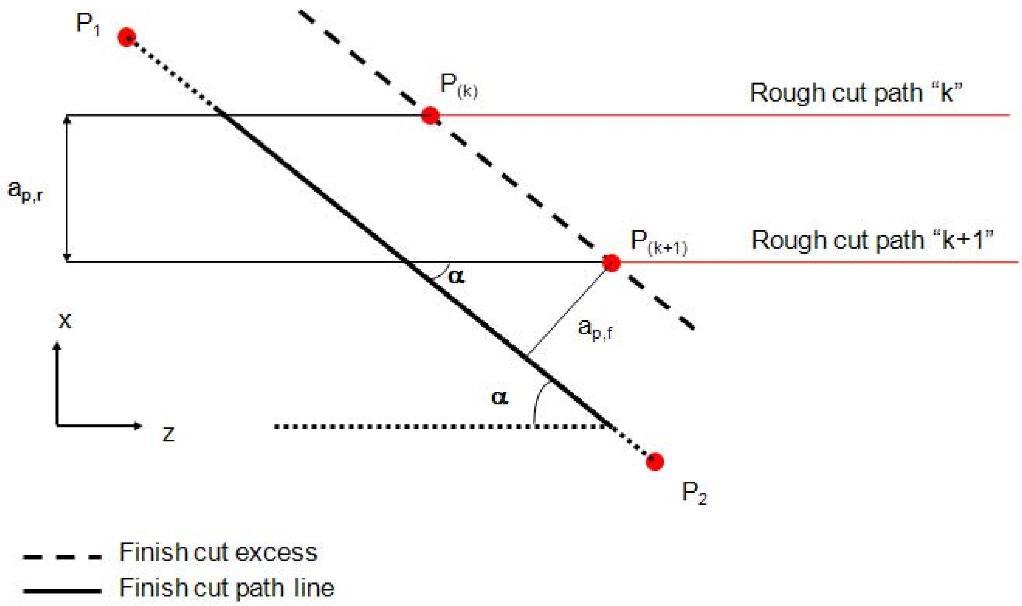


Figure 3.21. Detail A of Figure 3.20

$$\alpha = \tan^{-1}\left(\frac{P_{2,x} - P_{1,x}}{P_{2,z} - P_{1,z}}\right) \tag{3.105}$$

$$P_{(k+1),x} = P_{(k),x} - a_{p,r} \tag{3.106}$$

$$P_{(k+1),z} = \frac{P_{(k+1),x} - P_{1,x}}{\tan\alpha} + P_{1,z} - \frac{a_{p,f}}{\sin\alpha} \tag{3.107}$$

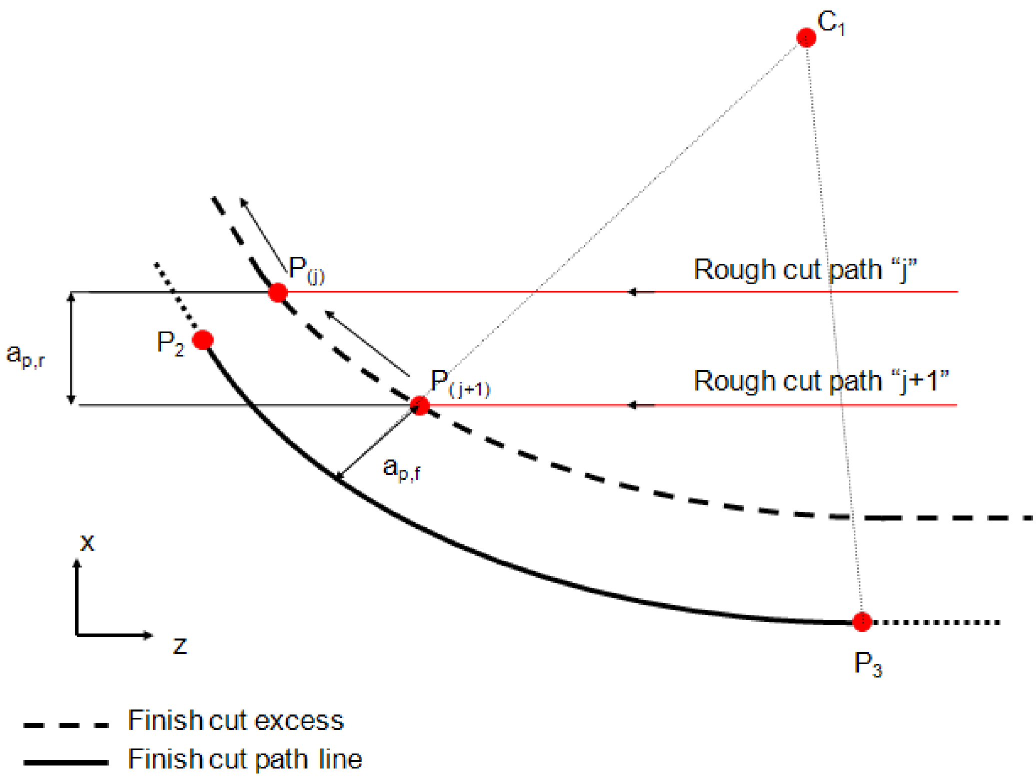


Figure 3.22. Detail B of Figure 3.20

$$r_{(j+1)} = |P_{(j+1)}C_1| \quad (3.108)$$

$$r_{(j+1)} = \sqrt{(P_{2,x} - C_{1,x})^2 + (P_{2,z} - C_{1,z})^2} - a_{p,f} \quad (3.109)$$

$$P_{(j+1),x} = P_{(j),x} - a_{p,r} \quad (3.110)$$

$$P_{(j+1),z} = \sqrt{r_{(j+1)}{}^2 - (P_{(j+1)xz} - C_{1,x})^2} + C_{1,z} \quad (3.111)$$

After the roughing cycle finishes the cutting tool is set to external finishing tool and the cutting tool traces the points from 1 to n up to the chuck clearance.

3.6.5.3 Calculations for Grooving

The x-position and z-position calculations for roughing cycle and finishing are almost same with external turning but the path lines for grooving are given for representation in Figure 3.23.

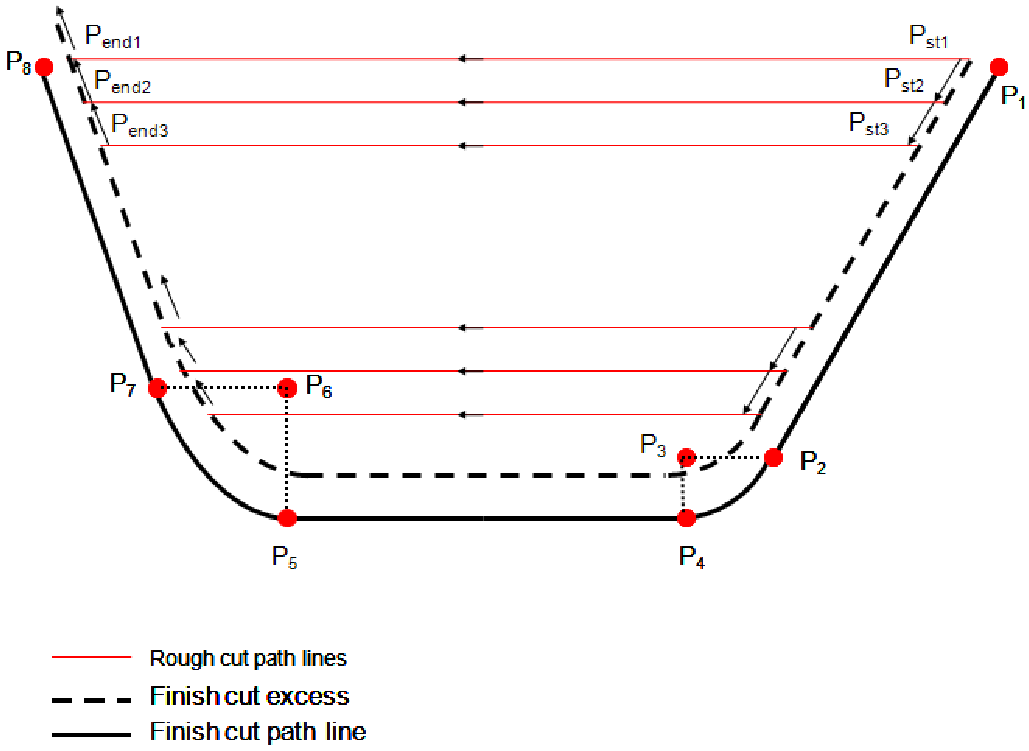


Figure 3.23. Grooving Path Lines

CHAPTER 4

THE DEVELOPED SOFTWARE

4.1 Welcoming Screen

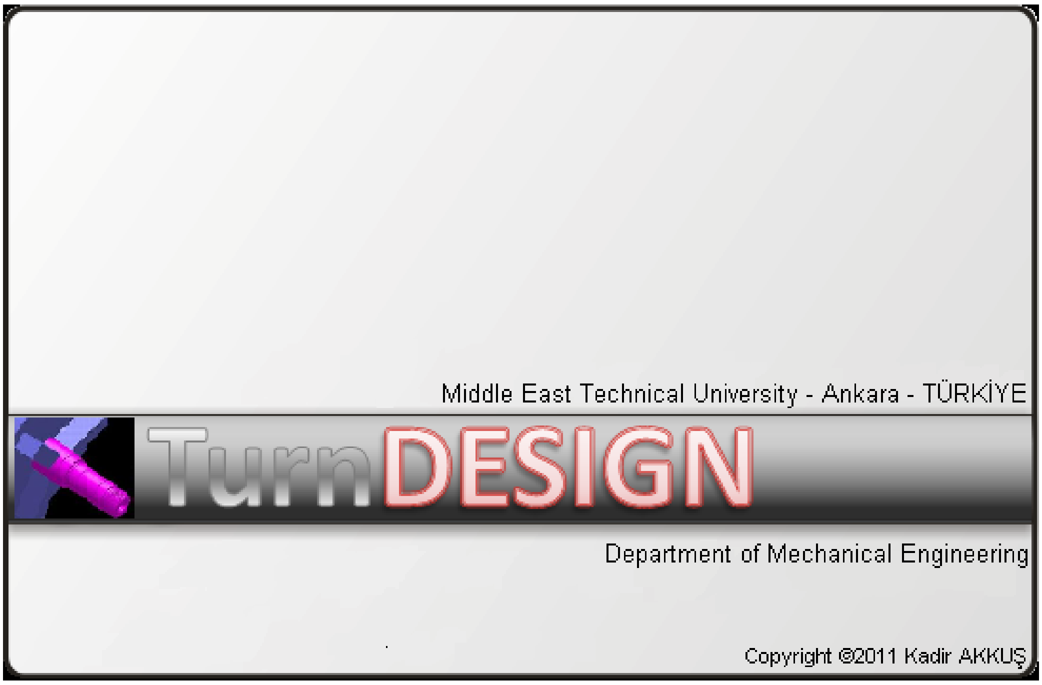


Figure 4.1. Welcoming Screen

A mouse click is required to remove the welcoming screen and the software starts with the modeling module, displaying information about preset preferences.

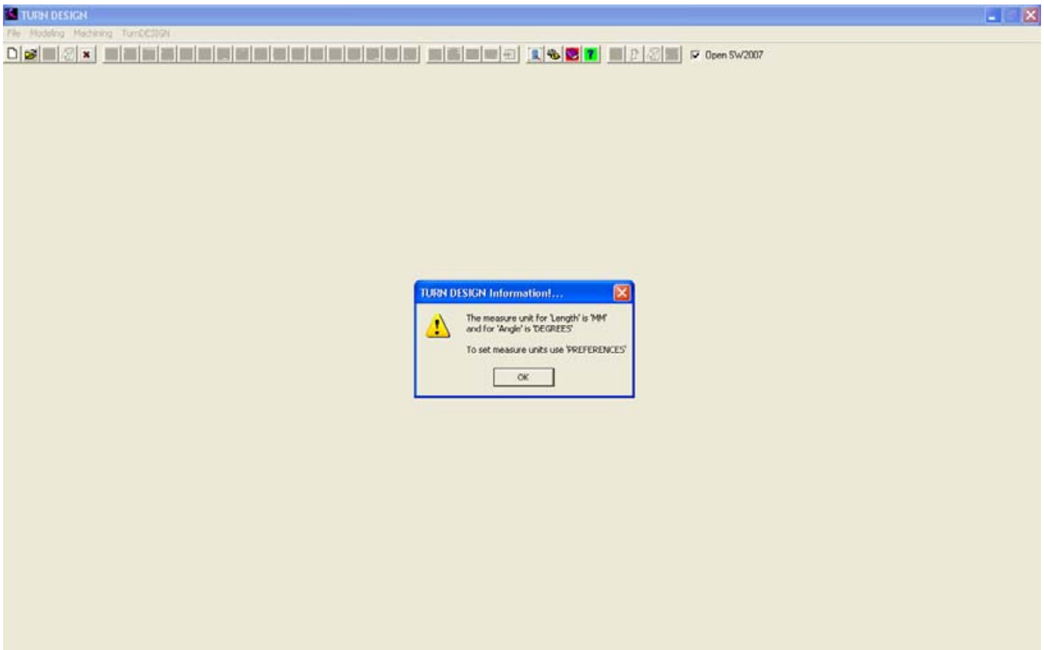


Figure 4.2. Starting Screen

A mouse click is required to remove the welcoming screen and the software starts with displaying information about preset preferences.

4.2 Modeling Module

Modeling module is equipped with buttons enabling linking to SolidWorks, creating features, 'UNDO', setting preferences, importing a STEP physical file for visualization, and viewing physical file content.

4.2.1 Creating Outer Diameter

To create an outer diameter, the user clicks on the 'Outer Diameter' button, types the length and diameter for the feature and clicks on the 'Tolerance' button to input tolerance data for the numeric parameters.

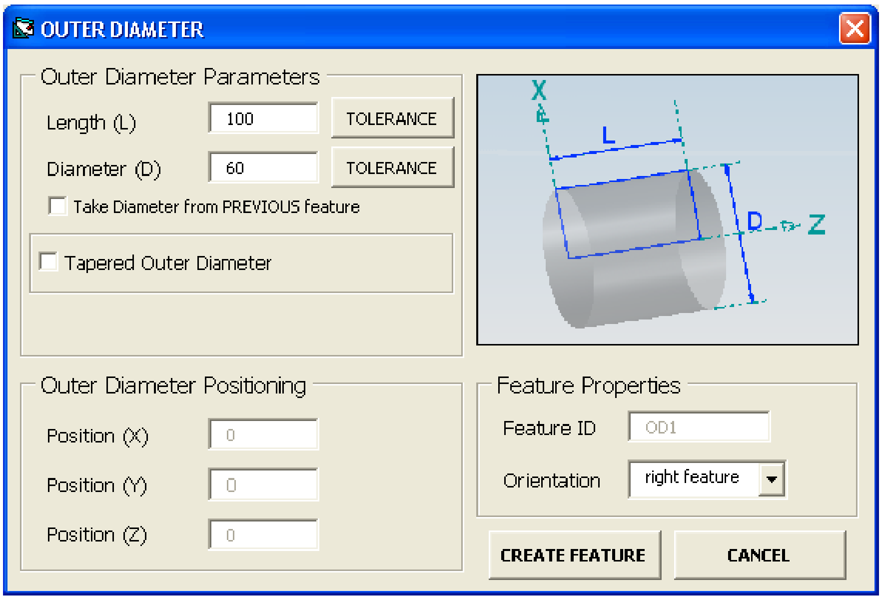


Figure 4.3. Outer Diameter Creation

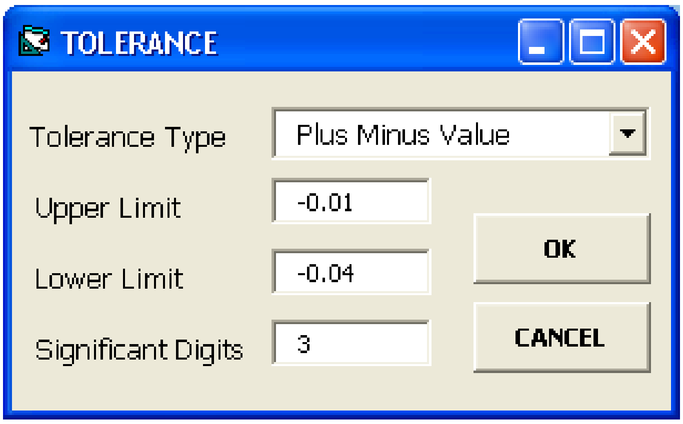


Figure 4.4. Input of Tolerance Data

Feature ID and positioning data is generated automatically. After inputting all parameters with tolerances, the user clicks 'create feature' button.

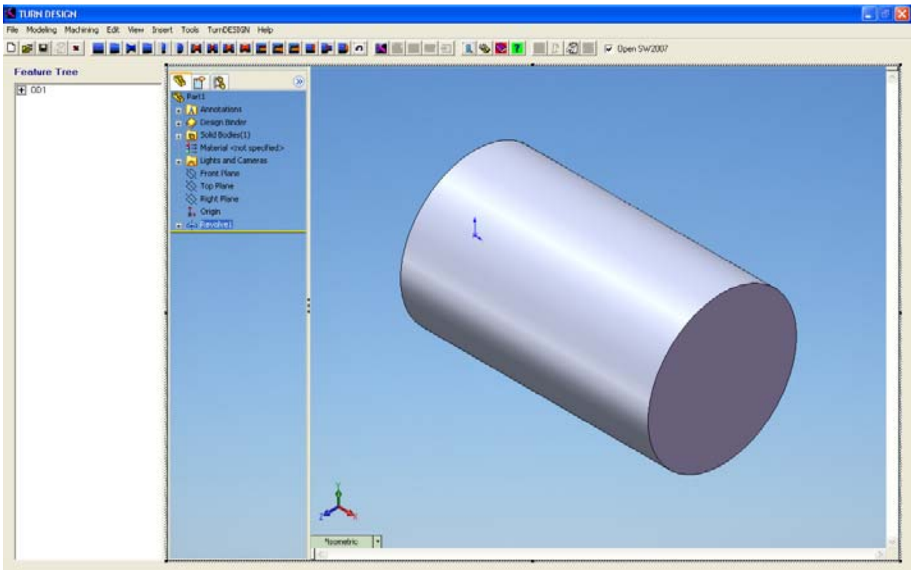


Figure 4.5. Display of Created Feature via SolidWorks

4.2.2 Creating Tapered Outer Diameter

To create a tapered outer diameter, the user clicks on the 'Outer Diameter' button, types the length and diameter for the feature and clicks on the 'Tolerance' button to input tolerance data for the numeric parameters. Different from outer diameter creation, the user clicks on the 'Tapered Outer Diameter' checkbox.

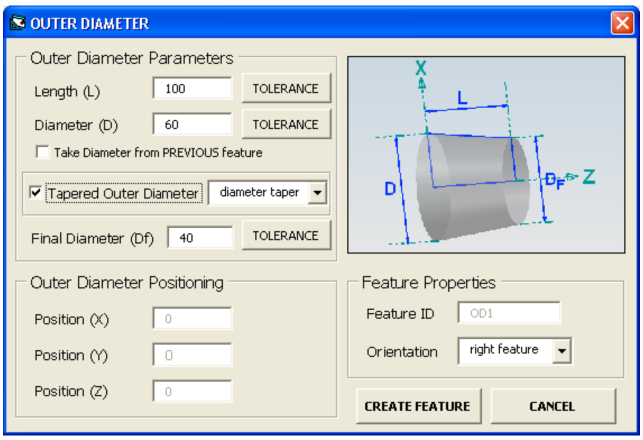


Figure 4.6. Tapered Outer Diameter Creation (with final diameter)

There are three options as mentioned before:

- With specifying final diameter (Figure 4.6)
- With specifying negative angle for decreasing taper (Figure 4.7)
- With specifying positive angle for increasing taper (Figure 4.8)

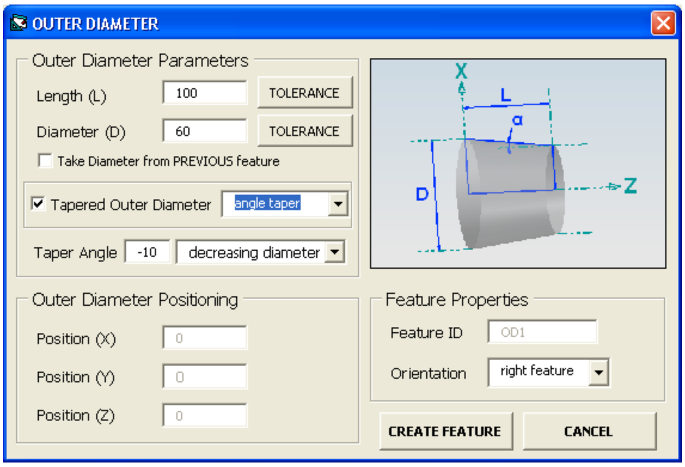


Figure 4.7. Tapered Outer Diameter Creation (with decreasing taper angle)

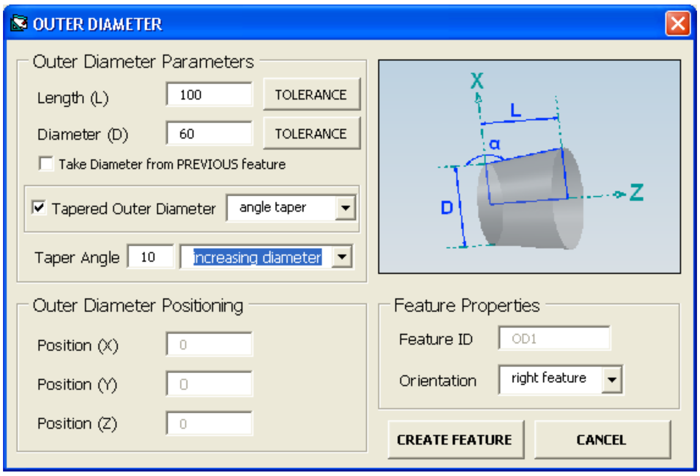


Figure 4.8. Tapered Outer Diameter Creation (with increasing taper angle)

In each case, the user clicks the 'Tolerance' button and specifies the tolerance for the third parameter determining the taper amount.

4.2.3 Other Features

Creation of all other features is straightforwardly similar. The user inputs the desired parameters with tolerances where tolerance button is available. Then the user clicks the 'Create Feature' button.

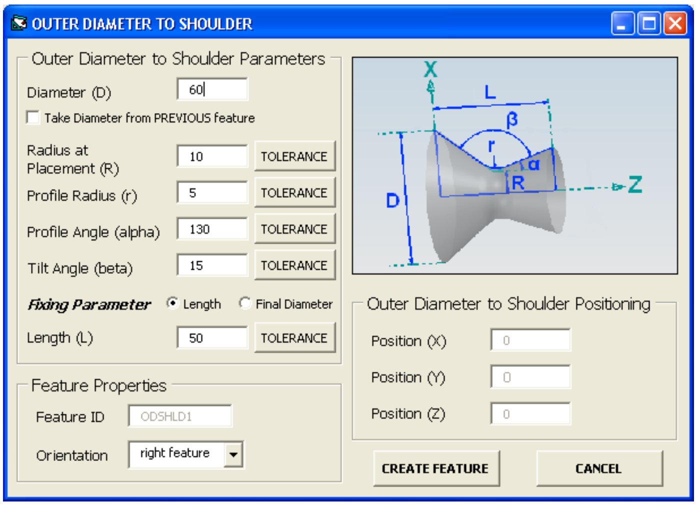


Figure 4.9. Outer Diameter to Shoulder Creation

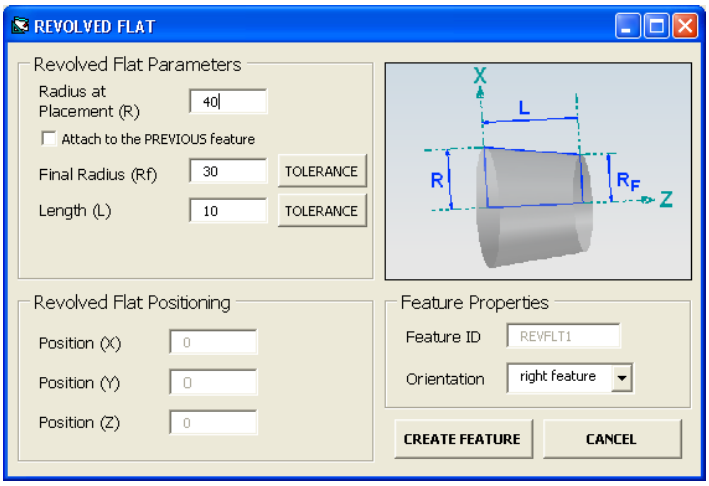


Figure 4.10. Revolved Flat Creation

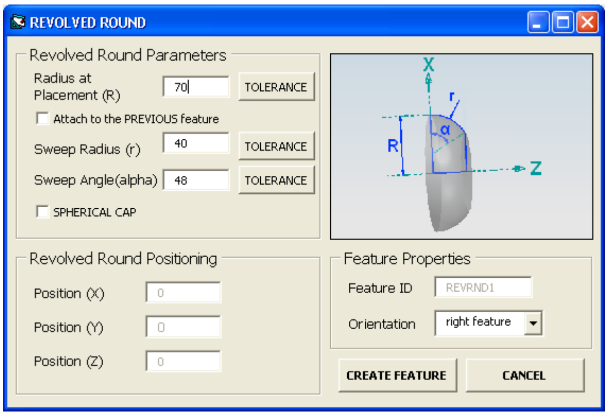


Figure 4.11. Revolved Round Creation

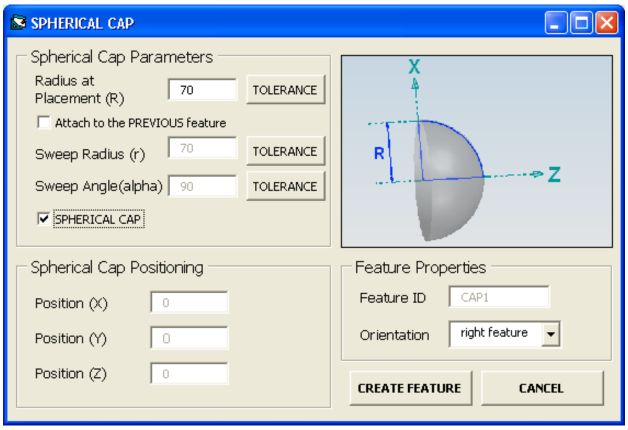


Figure 4.12. Spherical Cap Creation

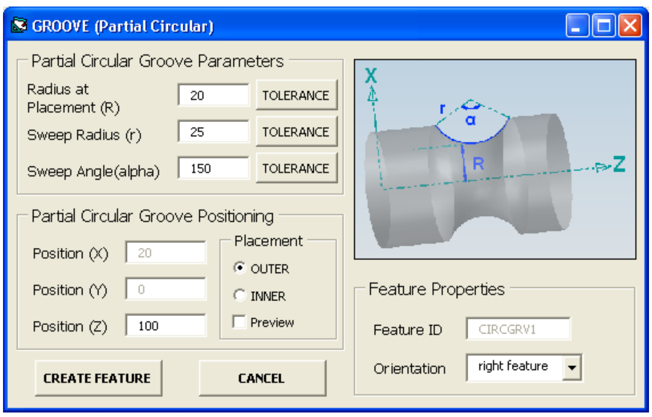


Figure 4.13. Partial Circular Groove Creation

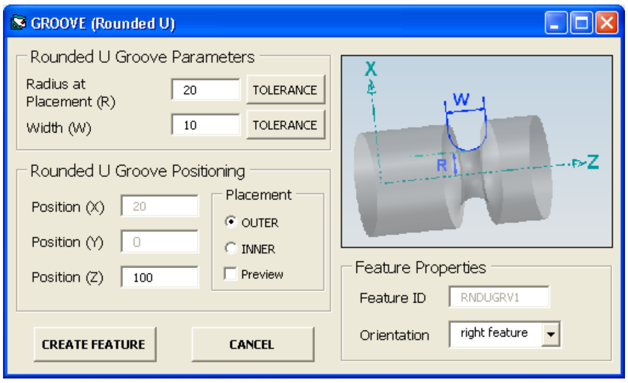


Figure 4.14. Rounded U Groove Creation

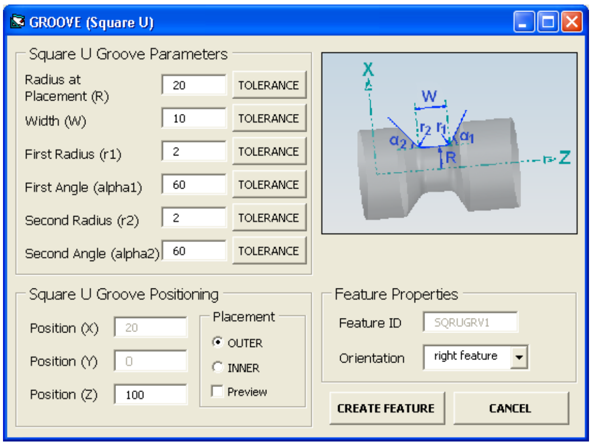


Figure 4.15. Square U Groove Creation

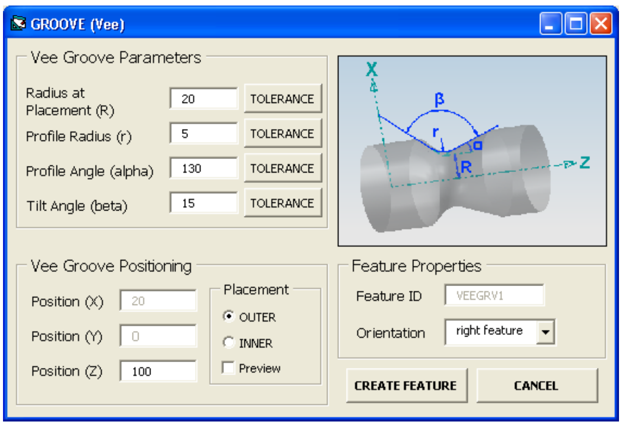


Figure 4.16. Vee Groove Creation

4.2.4 Transition Features

In transition, different from the other features, the user selects the features between those a transition wanted to be created.

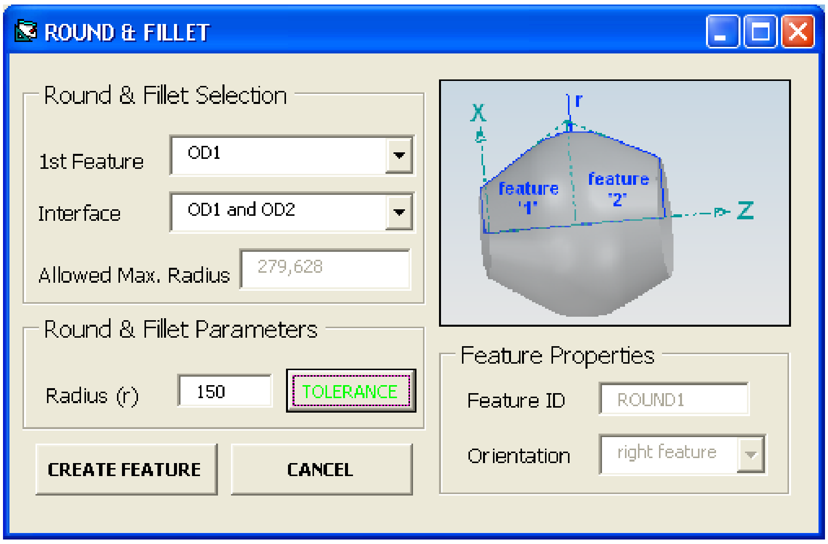


Figure 4.17. Round Transition Creation

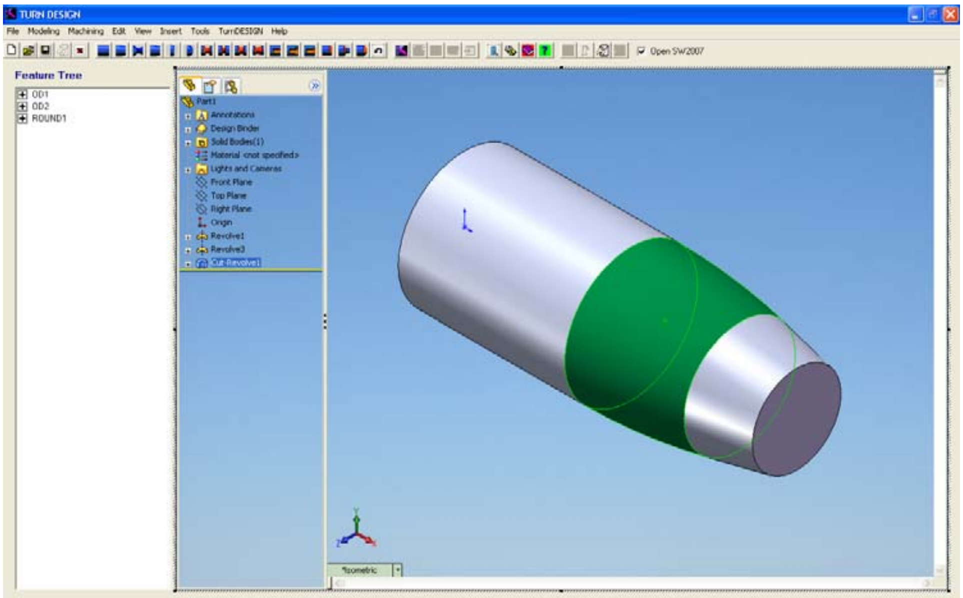


Figure 4.18. Display of Transition Feature via SolidWorks

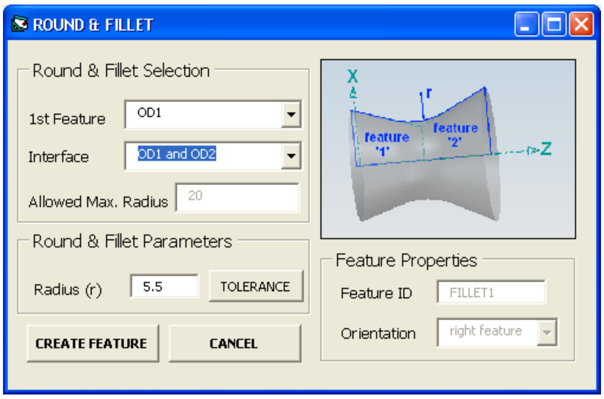


Figure 4.19. Fillet Transition Creation

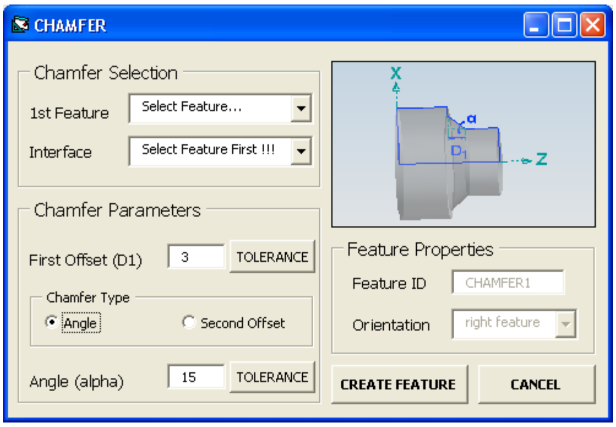


Figure 4.20. Chamfer Transition Creation (with angle)

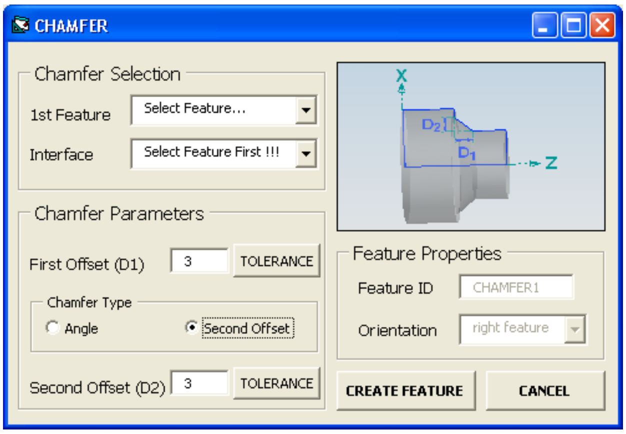


Figure 4.21. Chamfer Transition Creation (with second offset)

4.3 Machining Module

To start generating NC-code 'Machining Module' should be selected from the menu tab. The general layout of the machining module is seen in Figure 4.22. On the screen preferences concerning machining are displayed and selection summary is listed to enable tracking of the data creation progress.

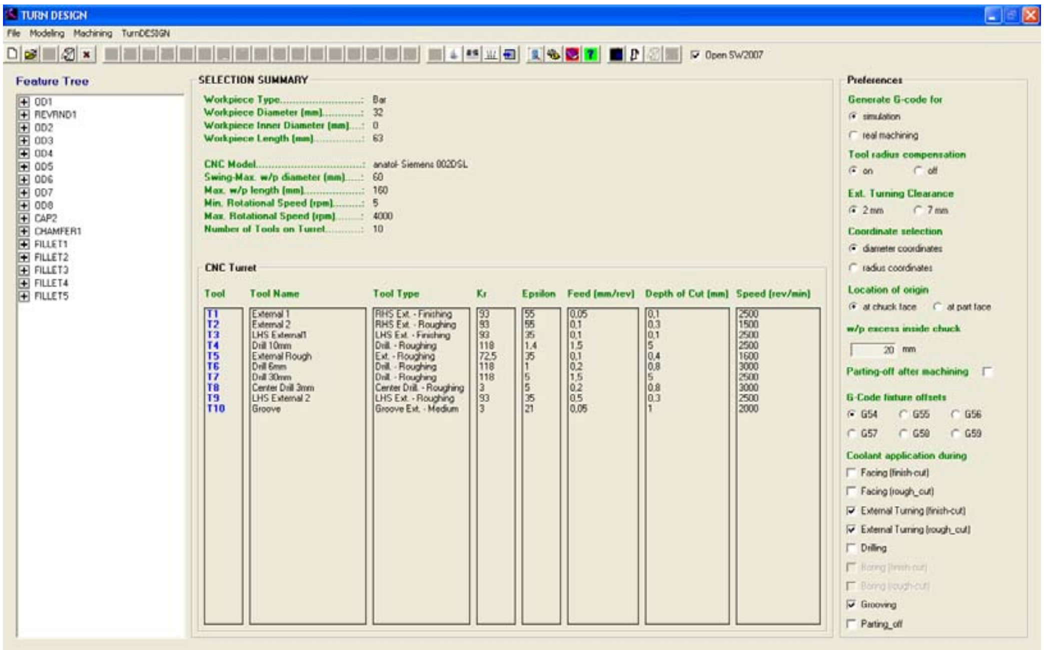


Figure 4.22. Machining Module Screen

The user clicks 'Start' to start data input to machining module and the stock size and shape recommended for machining the product from are calculated and displayed on the 'Selection Summary' frame automatically. If the user wishes to manipulate the raw size and shape, simply clicks on the 'Raw Stock Dimensions' button and sets the new raw size and shape.

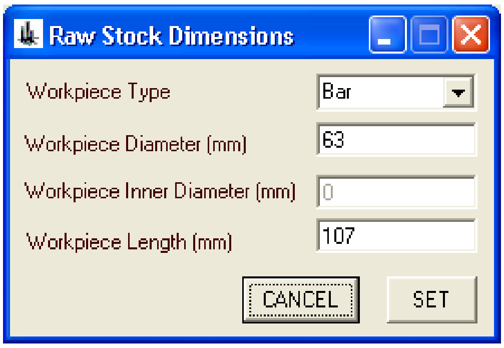


Figure 4.23. Setting Raw Material & Dimensions Manually

The user must set the 'CNC Specifications' and 'Tool Library' either selecting from the available ones or by adding new ones.

4.3.1 CNC Lathe Specifications

CNC Specifications must be selected by clicking 'CNC Specification' button and set by adding to/deleting/editing the data in the library.

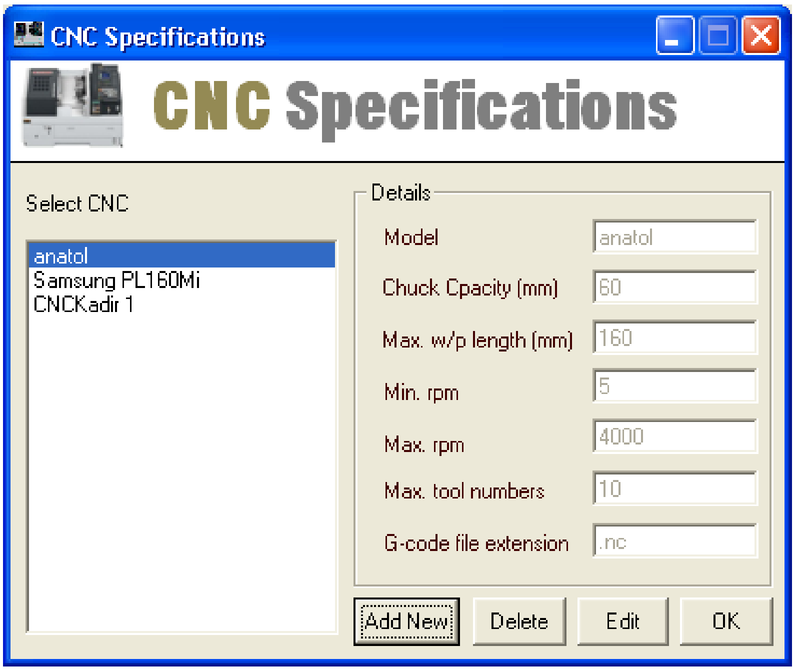


Figure 4.24. Setting CNC Lathe Specifications

4.3.2 Managing Tool Library

To generate effective and productive NC-codes, the user manages the tool library to simulate the tools in the hand and attached to the lathe turret. By adding new, editing the existing ones or deleting the useless ones, the user creates a unique library, since the library isn't a run time library. The library is stored for anytime usage.

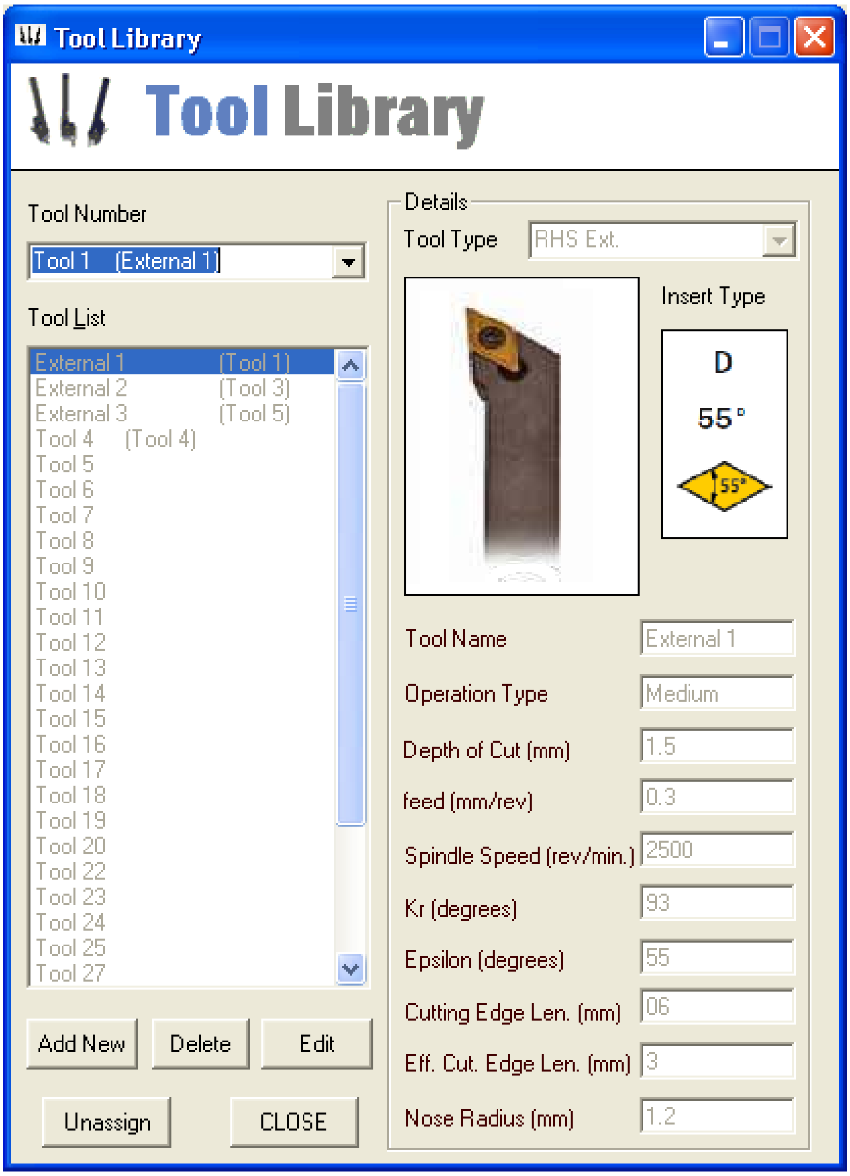


Figure 4.25. Setting Tool & Insert Library

4.3.3 NC-Code Generation

If the data is set carefully to suit the machining needs of the product, the user simply clicks on the 'Generate NC-Code' button and obtains the machining file. The user should be careful in selecting the purpose of NC-code generation before clicking this button; for simulation purpose or for real machining. Simulation with an erroneous machining file does not create problem but machining with an erroneous machining file may harm the machine tool and cutting tools.

4.4 Capabilities and Limitations of Software

This software is developed to present basically the usage of STEP AP224 features for the design of 2-axis rotational parts and generation of NC-code employing these feature definitions as the input data.

The modeling module of the software can model following features refined from STEP AP224:

- Outer diameter (straight or tapered)
- Outer diameter to shoulder
- Revolved flat and revolved round; also the cap which is the specialized version of revolved round
- Grooves (square U, rounded U, partial circular, vee)
- Hole and its combinations such as counterbore hole
- And transition features

Modeling module has some limitations regarding:

- The integration level of the VB and SW2007 via API.
- The lacking mathematical constants and functions in the VB and the user defined approximate solutions via functions and procedures to this problem.

The machining module of the software is capable of generating NC-code for the input data in STEP AP224 format. But it has some drawbacks and limitations due to the following issues:

- Simple NC-codes which are commonly accepted by CNC machine tools are used. This led to complex calculations and iterations inside the code. Combined with the approximate solutions to the lacking mathematical constants and functions in the VB generated codes may miss the design intend.
- The idea behind the tool library developed is being compatible with the simulation tool (CNC Simulator - Free CNC-Simulator by Bulldog Digital Technologies) used for the need of verification of the NC-code generated. Tool library capabilities are limited for rounded tool options.
- Reserved NC-codes for the CNC-Simulator is also in basic level.
- For internal operations only drilling is realized, the machining module should be studied on for boring operations.

Besides all, the software is capable of both designing and generating NC-codes for basic 2-axis turning parts and open to development for further need like boring.

CHAPTER 5

SAMPLE APPLICATIONS

5.1 First Sample

5.1.1 Product Geometry

A simple part is selected for demonstration as seen in Figure 5.1 and dimensions are given in Figure 5.2. In this part, external turning including RHS and LHS directions, center drilling and drilling capabilities of the machining module is presented. During the product design in feature modeler the following features are used:

- Straight outer diameter (D=29mm, L=36mm)
- Straight outer diameter (D=14mm, L=10mm)
- Tapered outer diameter (D=14mm, L=10mm, D_F=10mm)
- Straight outer diameter (D=10mm, L=10mm)
- Round Hole with conical bottom (D=6mm, L=10mm, α=118°)
- Fillet between OD1 and OD2 (r=6mm)
- Square U groove (W=27.5mm, R_p=10mm, α_1=90°, α_2=90°, r_1=0mm, r_2=0mm, z_pos=18.75mm)

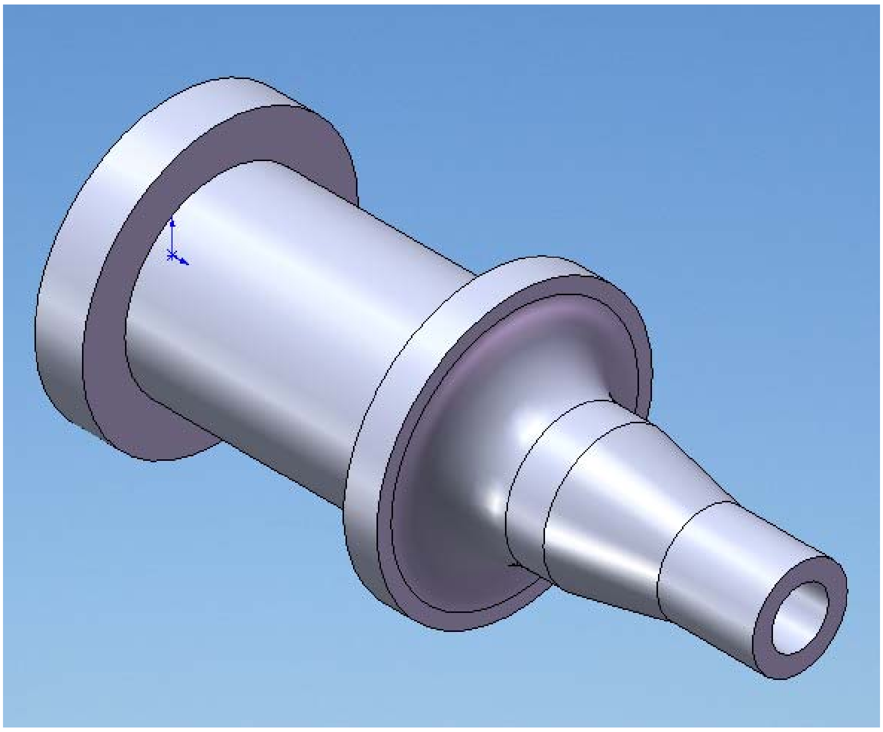


Figure 5.1. Geometry of the First Sample

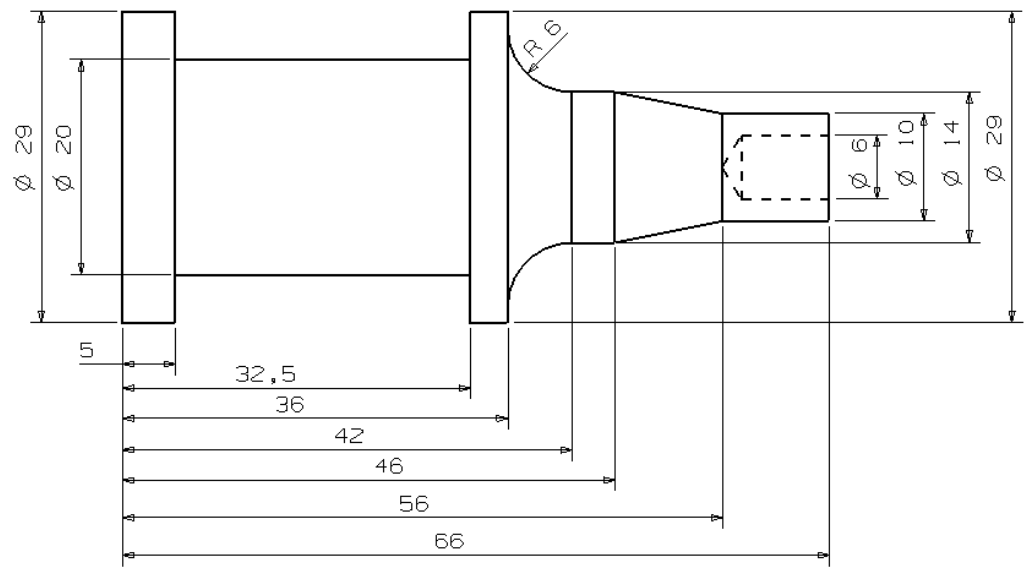


Figure 5.2. Dimensions of the First Sample

5.1.2 STEP Part 21 Physical File Output

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ISO-10303-21;
HEADER;
FILE_DESCRIPTION(('ISO-10303-224,FEATURE BASED ROTATIONAL PART
CREATED BY',
'TurnDESIGN','COPYRIGHT','KADiR AKKUŞ'),'1');
FILE_NAME(('thesis_sample_1.part21','2011-2-9T13:47:33',
('KADİR AKKUŞ','METU-ME','ANKARA TURKİYE'),('METU-ME','ANKARA
TURKİYE'),
'TurnDESIGN VERSION1','TurnDESIGN','APPROVED BY S.ENGİN KILIÇ');
FILE_SCHEMA(('ROTATIONAL PART', 'PART FOR 2-AXIS TURNING'));
ENDSEC;
DATA;
#1=OUTER_DIAMETER('OD1',#2,#3,#4,$);
#2=ORIENTATION((0,0,0),(1,0,0,0,1,0,0,0,1));
#3=NUMERIC_PARAMETER('OD1 DIAMETER',MM,29,#5);
#4=NUMERIC_PARAMETER('OD1 LENGTH',MM,36,#6);
#5=TOLERANCE_RANGE(0.01,0.01,3);
#6=TOLERANCE_RANGE(0.01,0.01,3);
#7=OUTER_DIAMETER('OD2',#8,#9,#10,$);
#8=ORIENTATION((0,0,36),(1,0,0,0,1,0,0,0,1));
#9=NUMERIC_PARAMETER('OD2 DIAMETER',MM,14,#11);
#10=NUMERIC_PARAMETER('OD2 LENGTH',MM,10,#12);
#11=TOLERANCE_RANGE(0.01,0.01,3);
#12=TOLERANCE_RANGE(0.01,0.01,3);
#13=OUTER_DIAMETER('OD3',#14,#15,#16,#17);
```

```
#14=ORIENTATION((0,0,46),(1,0,0,0,1,0,0,0,1));
#15=NUMERIC_PARAMETER('OD3 DIAMETER',MM,14,#18);
#16=NUMERIC_PARAMETER('OD3 LENGTH',MM,10,#19);
#17=DIAMETER_TAPER(#20);
#18=TOLERANCE_RANGE(0.01,0.01,3);
#19=TOLERANCE_RANGE(0.01,0.01,3);
#20=NUMERIC_PARAMETER('OD3 FINAL DIAMETER',MM,10,#21);
#21=TOLERANCE_RANGE(0.01,0.01,3);
#22=OUTER_DIAMETER('OD4',#23,#24,#25,$);
#23=ORIENTATION((0,0,56),(1,0,0,0,1,0,0,0,1));
#24=NUMERIC_PARAMETER('OD4 DIAMETER',MM,10,#26);
#25=NUMERIC_PARAMETER('OD4 LENGTH',MM,10,#27);
#26=TOLERANCE_RANGE(0.01,0.01,3);
#27=TOLERANCE_RANGE(0.01,0.01,3);
#28=ROUND_HOLE('RNDHOL1',#29,$,#30,#31,$,#32);
#29=ORIENTATION((0,0,66),(1,0,0,0,1,0,0,0,1));
#30=LINEAR_PATH(#33,#34);
#31=NUMERIC_PARAMETER('RNDHOL1 DIAMETER',MM,6,#35);
#32=CONICAL_HOLE_BOTTOM(#37);
#33=NUMERIC_PARAMETER('RNDHOL1 DISTANCE',MM,10,#36);
#34=DIRECTION(0,0,-1);
#35=TOLERANCE_RANGE(0.01,0.01,3);
#36=TOLERANCE_RANGE(0.01,0.01,3);
#37=NUMERIC_PARAMETER('RNDHOL1 CONICAL BOTTOM TIP
ANGLE',MM,118,#38);
#38=TOLERANCE_RANGE(0.01,0.01,3);
```

```
#39=NUMERIC_PARAMETER('RNDHOL1 CONICAL BOTTOM TIP
RADIUS',MM,1,#40);
#40=TOLERANCE_RANGE(0.01,0.01,3);
#41=CONSTANT_RADIUS_FILLET('FILLET1','OD1','OD2',#42,$,$);
#42=NUMERIC_PARAMETER('FILLET1 RADIUS',MM,6,#43);
#43=TOLERANCE_RANGE(0.01,0.01,3);
#44=GROOVE('SQRUGRV1',#45,#46,#47,#48);
#45=ORIENTATION((10,0,18.75),(1,0,0,0,1,0,0,0,1));
#46=DIRECTION(-1,0,0);
#47=NUMERIC_PARAMETER('SQRUGRV1 GROOVE RADIUS',MM,10,#49);
#48=SQUARE_U_PROFILE($,#50,#51,#52,#53,#54);
#49=TOLERANCE_RANGE(0.01,0.01,3);
#50=NUMERIC_PARAMETER('SQRUGRV1 WIDTH',MM,27.5,#55);
#51=NUMERIC_PARAMETER('SQRUGRV1 FIRST RADIUS',MM,0,#56);
#52=NUMERIC_PARAMETER('SQRUGRV1 FIRST
ANGLE',DEGREES,90,#57);
#53=NUMERIC_PARAMETER('SQRUGRV1 SECOND RADIUS',MM,0,#58);
#54=NUMERIC_PARAMETER('SQRUGRV1 SECOND
ANGLE',DEGREES,90,#59);
#55=TOLERANCE_RANGE(0.01,0.01,3);
#56=TOLERANCE_RANGE(0.01,0.01,3);
#57=TOLERANCE_RANGE(0.01,0.01,3);
#58=TOLERANCE_RANGE(0.01,0.01,3);
#59=TOLERANCE_RANGE(0.01,0.01,3);
ENDSEC;
END-ISO-10303-21;
```

5.1.3 CNC Simulator Output

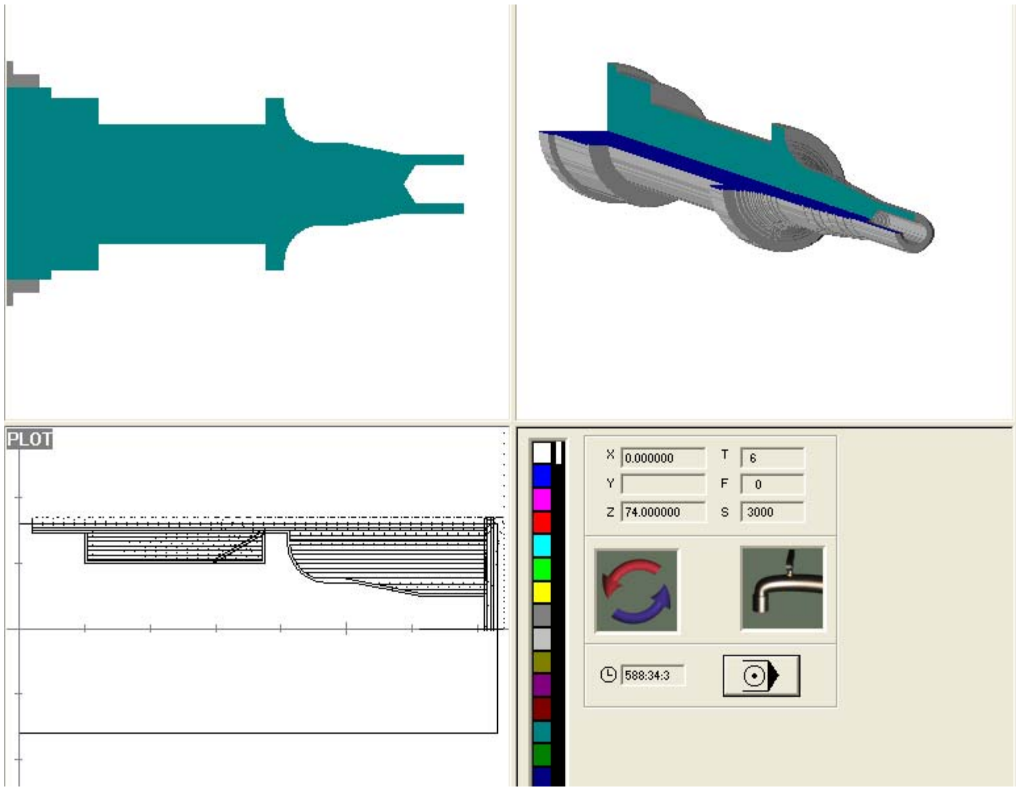

Figure 5.3. CNC Simulator Output for the First Sample

5.1.4 Tools Used for Machining

CNC Turret

Tool	Tool Name	Tool Type	Kr °	Epsilon °	Feed (mm/rev)	Depth of Cut (mm)	Speed (rev/min)
T1	External 2	RHS Ext. - Roughing	93	55	0,1	0,7	1500
T2	External 1	RHS Ext. - Finishing	93	55	0,05	0,4	2500
T3	LHS External 2	LHS Ext. - Roughing	93	55	0,5	1	2500
T4	LHS External1	LHS Ext. - Finishing	93	55	0,05	0,5	2500
T5	Center Drill 3mm	Center Drill. - Roughing	3	5	0,2	0,8	3000
T6	Drill 6mm	Drill. - Roughing	118	1	0,2	0,8	3000
T7	not set	-	-	-	-	-	-
T8	not set	-	-	-	-	-	-
T9	not set	-	-	-	-	-	-
T10	not set	-	-	-	-	-	-

Figure 5.4. Tool List for the First Sample

5.1.5 Produced Part

Figure 5.5. Photo of the First Sample after production

5.2 Second Sample

5.2.1 Product Geometry

A chess piece part composed of external features is selected for demonstration as seen in Figure 5.6 and dimensions are given in Figure 5.7. In this part, more complicated external turning including RHS and LHS directions, and multiple sqru groove handling capabilities of the machining module is presented. During the product design in feature modeler the following features are used:

- Straight outer diameter (D=28mm, L=3mm)
- Revolved round (R_p=14mm, α=34°, r=4mm)
- Straight outer diameter (D=26.6mm, L=22mm)
- Straight outer diameter (D=21mm, L=30mm)
- Straight outer diameter (D=19mm, L=2.1mm)
- Spherical cap (R=5mm)

- Square U groove (W=12mm, R_p=10mm, α_1=14.013°, α_2=90°, r_1=20mm, r_2=3.2mm, z_pos=11.2mm)
- Square U groove (W=11mm, R_p=5mm, α_1=36.867°, α_2=38.15°, r_1=5mm, r_2=3mm, z_pos=45.7mm)
- Fillet between OD2 and OD3 (r=2mm)
- Chamfer on edge2 of OD4 (D_1=1mm, α=45°)

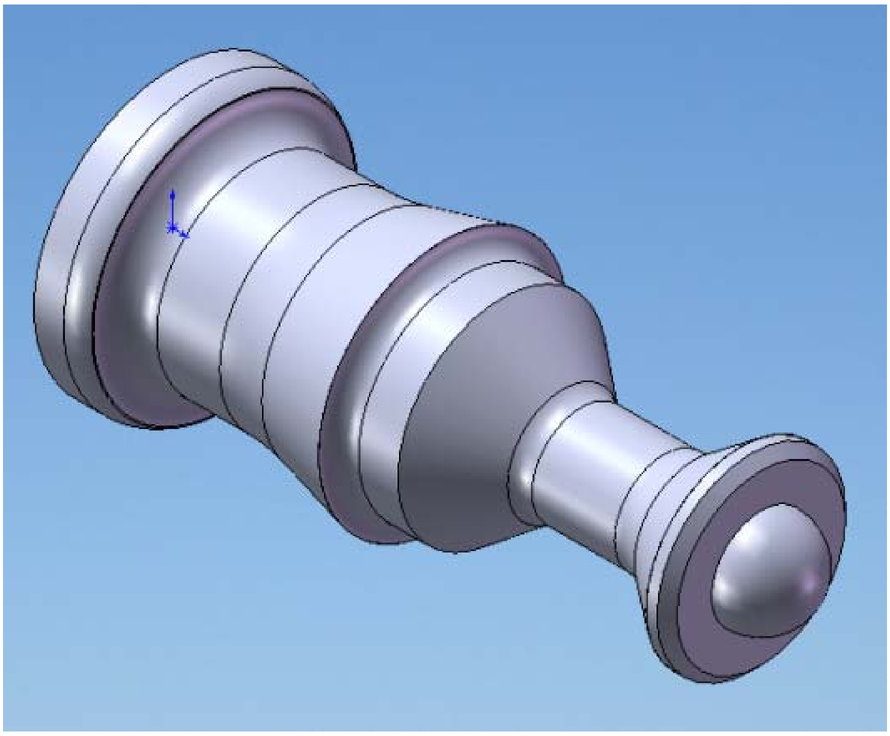

Figure 5.6. Geometry of the Second Sample

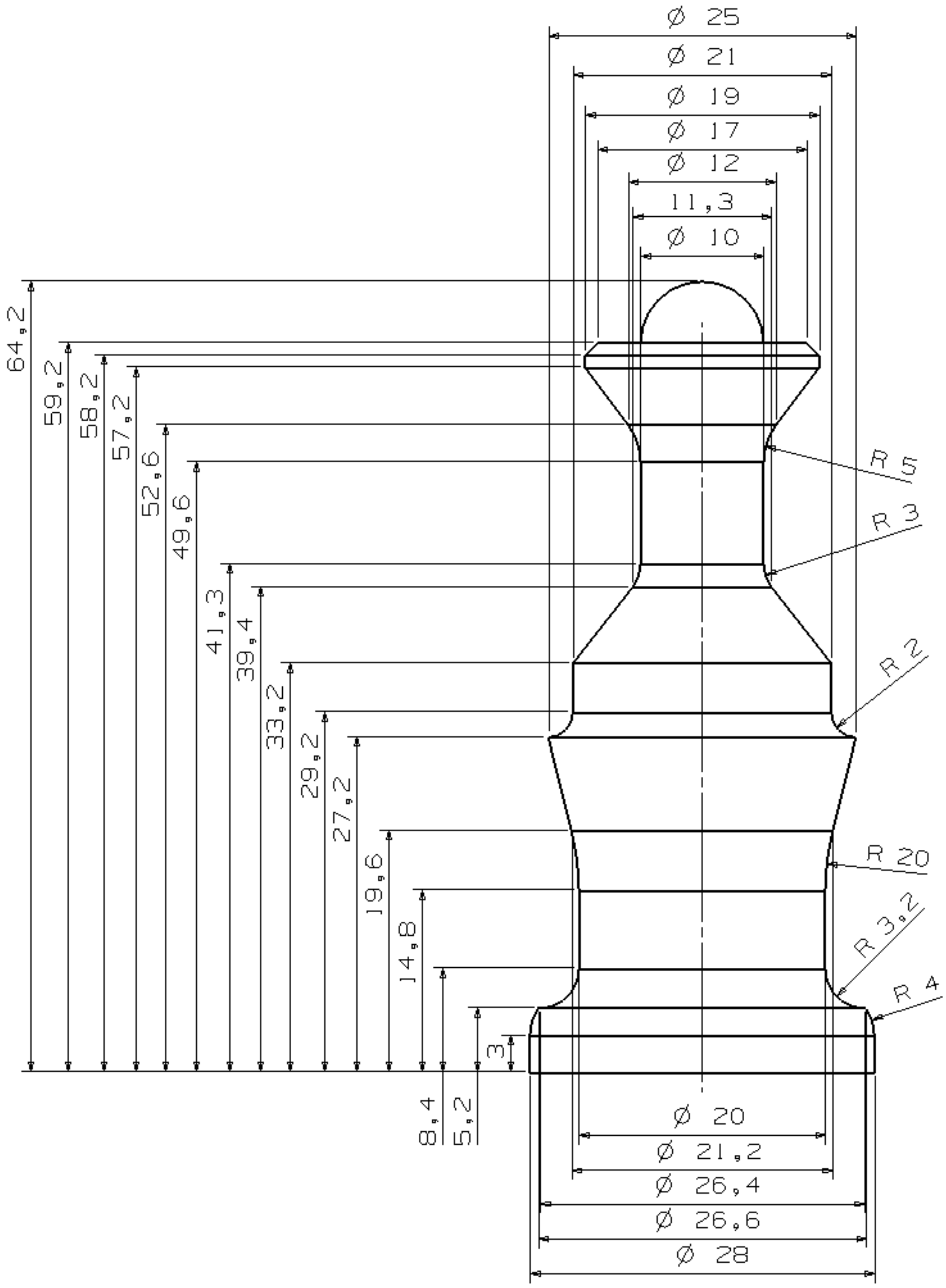

Figure 5.7. Dimensions of the Second Sample

5.2.2 STEP Part 21 Physical File Output

ISO-10303-21;

HEADER;

FILE_DESCRIPTION(('ISO-10303-224,FEATURE BASED ROTATIONAL PART CREATED BY',

'TurnDESIGN','COPYRIGHT','KADiR AKKUŞ'),'1');

FILE_NAME(('thesis_sample_2.part21','2011-5-7T18:29:23',

('KADİR AKKUŞ','METU-ME','ANKARA TURKİYE'),('METU-ME','ANKARA TURKİYE'),

'TurnDESIGN VERSION1','TurnDESIGN','APPROVED BY S.ENGİN KILIÇ');

FILE_SCHEMA(('ROTATIONAL PART', 'PART FOR 2-AXIS TURNING'));

ENDSEC;

DATA;

#1=OUTER_DIAMETER('OD1',#2,#3,#4,$);

#2=ORIENTATION((0,0,0),(1,0,0,0,1,0,0,0,1));

#3=NUMERIC_PARAMETER('OD1 DIAMETER',MM,28,#5);

#4=NUMERIC_PARAMETER('OD1 LENGTH',MM,3,#6);

#5=TOLERANCE_RANGE(0.01,0.01,3);

#6=TOLERANCE_RANGE(0.01,0.01,3);

#7=REVOLVED_ROUND('REVRND1',#8,#9,#10,#11);

#8=ORIENTATION((0,0,3),(1,0,0,0,1,0,0,0,1));

#9=DIRECTION(-0.9563047,0,-0.2923717);

#10=NUMERIC_PARAMETER('REVRND1 REVOLVED FEATURE RADIUS',MM,13.31615,#14);

#11=PARTIAL_CIRCULAR_PROFILE($,#12,#13);

#12=NUMERIC_PARAMETER('REVRND1 SWEEP ANGLE',DEGREES,34,#15);

#13=NUMERIC_PARAMETER('REVRND1 SWEEP RADIUS',MM,4,#16);

```
#14=TOLERANCE_RANGE(0.01,0.01,3);

#15=TOLERANCE_RANGE(0.01,0.01,3);

#16=TOLERANCE_RANGE(0.01,0.01,3);

#17=OUTER_DIAMETER('OD2',#18,#19,#20,$);

#18=ORIENTATION((0,0,5.236772),(1,0,0,0,1,0,0,0,1));

#19=NUMERIC_PARAMETER('OD2 DIAMETER',MM,26.6,#21);

#20=NUMERIC_PARAMETER('OD2 LENGTH',MM,22,#22);

#21=TOLERANCE_RANGE(0.01,0.01,3);

#22=TOLERANCE_RANGE(0.01,0.01,3);

#23=OUTER_DIAMETER('OD3',#24,#25,#26,$);

#24=ORIENTATION((0,0,27.23677),(1,0,0,0,1,0,0,0,1));

#25=NUMERIC_PARAMETER('OD3 DIAMETER',MM,21,#27);

#26=NUMERIC_PARAMETER('OD3 LENGTH',MM,30,#28);

#27=TOLERANCE_RANGE(0.01,0.01,3);

#28=TOLERANCE_RANGE(0.01,0.01,3);

#29=OUTER_DIAMETER('OD4',#30,#31,#32,$);

#30=ORIENTATION((0,0,57.23677),(1,0,0,0,1,0,0,0,1));

#31=NUMERIC_PARAMETER('OD4 DIAMETER',MM,19,#33);

#32=NUMERIC_PARAMETER('OD4 LENGTH',MM,2.1,#34);

#33=TOLERANCE_RANGE(0.01,0.01,3);

#34=TOLERANCE_RANGE(0.01,0.01,3);

#35=REVOLVED_ROUND('CAP1',#36,#37,#38,#39);

#36=ORIENTATION((0,0,59.33677),(1,0,0,0,1,0,0,0,1));

#37=DIRECTION(-0.7071068,0,-0.7071068);

#38=NUMERIC_PARAMETER('CAP1 REVOLVED FEATURE
RADIUS',MM,6.083982E-08,#42);

#39=PARTIAL_CIRCULAR_PROFILE($,#40,#41);
```

#40=NUMERIC_PARAMETER('CAP1 SWEEP ANGLE',DEGREES,90,#43);

#41=NUMERIC_PARAMETER('CAP1 SWEEP RADIUS',MM,5,#44);

#42=TOLERANCE_RANGE(0.01,0.01,3);

#43=TOLERANCE_RANGE(0.01,0.01,3);

#44=TOLERANCE_RANGE(0.01,0.01,3);

#45=GROOVE('SQRUGRV1',#46,#47,#48,#49);

#46=ORIENTATION((10,0,11.2),(1,0,0,0,1,0,0,0,1));

#47=DIRECTION(-1,0,0);

#48=NUMERIC_PARAMETER('SQRUGRV1 GROOVE RADIUS',MM,10,#50);

#49=SQUARE_U_PROFILE($,#51,#52,#53,#54,#55);

#50=TOLERANCE_RANGE(0.01,0.01,3);

#51=NUMERIC_PARAMETER('SQRUGRV1 WIDTH',MM,12,#56);

#52=NUMERIC_PARAMETER('SQRUGRV1 FIRST RADIUS',MM,20,#57);

#53=NUMERIC_PARAMETER('SQRUGRV1 FIRST
ANGLE',DEGREES,14.013,#58);

#54=NUMERIC_PARAMETER('SQRUGRV1 SECOND RADIUS',MM,3.2,#59);

#55=NUMERIC_PARAMETER('SQRUGRV1 SECOND ANGLE',DEGREES,90,#60);

#56=TOLERANCE_RANGE(0.01,0.01,3);

#57=TOLERANCE_RANGE(0.01,0.01,3);

#58=TOLERANCE_RANGE(0.01,0.01,3);

#59=TOLERANCE_RANGE(0.01,0.01,3);

#60=TOLERANCE_RANGE(0.01,0.01,3);

#61=GROOVE('SQRUGRV2',#62,#63,#64,#65);

#62=ORIENTATION((5,0,45.7),(1,0,0,0,1,0,0,0,1));

#63=DIRECTION(-1,0,0);

#64=NUMERIC_PARAMETER('SQRUGRV2 GROOVE RADIUS',MM,5,#66);

#65=SQUARE_U_PROFILE($,#67,#68,#69,#70,#71);

#66=TOLERANCE_RANGE(0.01,0.01,3);

#67=NUMERIC_PARAMETER('SQRUGRV2 WIDTH',MM,11,#72);

#68=NUMERIC_PARAMETER('SQRUGRV2 FIRST RADIUS',MM,5,#73);

#69=NUMERIC_PARAMETER('SQRUGRV2 FIRST ANGLE',DEGREES,36.867,#74);

#70=NUMERIC_PARAMETER('SQRUGRV2 SECOND RADIUS',MM,3,#75);

#71=NUMERIC_PARAMETER('SQRUGRV2 SECOND ANGLE',DEGREES,38.15,#76);

#72=TOLERANCE_RANGE(0.01,0.01,3);

#73=TOLERANCE_RANGE(0.01,0.01,3);

#74=TOLERANCE_RANGE(0.01,0.01,3);

#75=TOLERANCE_RANGE(0.01,0.01,3);

#76=TOLERANCE_RANGE(0.01,0.01,3);

#77=CONSTANT_RADIUS_FILLET('FILLET1','OD2','OD3',#78,$,$);

#78=NUMERIC_PARAMETER('FILLET1 RADIUS',MM,2,#79);

#79=TOLERANCE_RANGE(0.01,0.01,3);

#80=CHAMFER('CHAMFER1',#81,#82);

#81=FIRST_OFFSET('OD4',#83);

#82=SECOND_CHAMFER_OFFSET('EDGE2',#84);

#83=NUMERIC_PARAMETER('CHAMFER1 FIRST_OFFSET',MM,1,#85);

#84=CHAMFER_ANGLE(#86);

#85=TOLERANCE_RANGE(0.01,0.01,3);

#86=NUMERIC_PARAMETER('CHAMFER1 ANGLE',DEGREES,45,#87);

#87=TOLERANCE_RANGE(0.01,0.01,3);

ENDSEC;

END-ISO-10303-21;

5.2.3 CNC Simulator Output

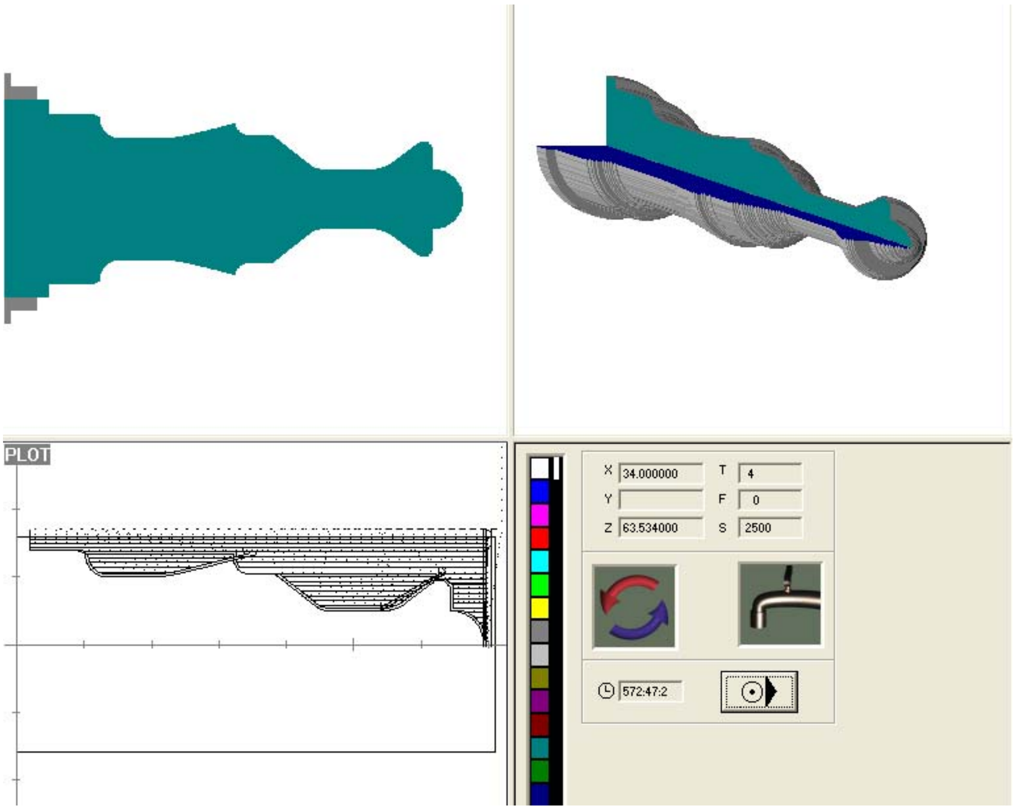

Figure 5.8. CNC Simulator Output for the Second Sample

5.2.4 Tools Used for Machining

CNC Turret

Tool	Tool Name	Tool Type	Kr °	Epsilon °	Feed (mm/rev)	Depth of Cut (mm)	Speed (rev/min)
T1	External 2	RHS Ext. - Roughing	93	55	0,1	0,7	1500
T2	External 1	RHS Ext. - Finishing	93	55	0,05	0,4	2500
T3	LHS External 2	LHS Ext. - Roughing	93	55	0,5	1	2500
T4	LHS External1	LHS Ext. - Finishing	93	55	0,05	0,5	2500
T5	not set	-	-	-	-	-	-
T6	not set	-	-	-	-	-	-
T7	not set	-	-	-	-	-	-
T8	not set	-	-	-	-	-	-
T9	not set	-	-	-	-	-	-
T10	not set	-	-	-	-	-	-

Figure 5.9. Tool List for the Second Sample

5.2.5 Produced Part

Figure 5.10. Photo of the Second Sample after production

5.3 Third Sample

5.3.1 Product Geometry

A complicated part composed of external features is selected for demonstration as seen in Figure 5.11 and dimensions are given in Figure 5.12. In this part, more complicated external turning including RHS and LHS directions, chuck side chamfer and multiple types of grooves (sqru, vee, rndu) handling capabilities of the machining module is presented. During the product design in feature modeler the following features are used:

- Straight outer diameter (D=60mm, L=30mm)
- Straight outer diameter (D=84mm, L=60mm)
- Straight outer diameter (D=80mm, L=90mm)
- Straight outer diameter (D=60mm, L=20mm)
- Revolved round (R_p=30mm, α=48°, r=40mm)
- Round Hole with conical bottom (D=10mm, L=30mm, α=118°)

- Rounded U groove (W=10mm, R_p=20mm, z_pos=18mm)
- Vee groove (R_p=18mm, r=10mm, α(tilt_angle)=38.667°, β(profile_angle)=110.383°, z_pos=50mm)
- Square U groove (W=50mm, R_p=15mm, α_1=32°, α_2=90°, r_1=80mm, r_2=10mm, z_pos=115mm)
- Fillet between OD3 and OD4 (r=10mm)
- Chamfer on edge1 of OD1 (D_1=5mm, α=45°)

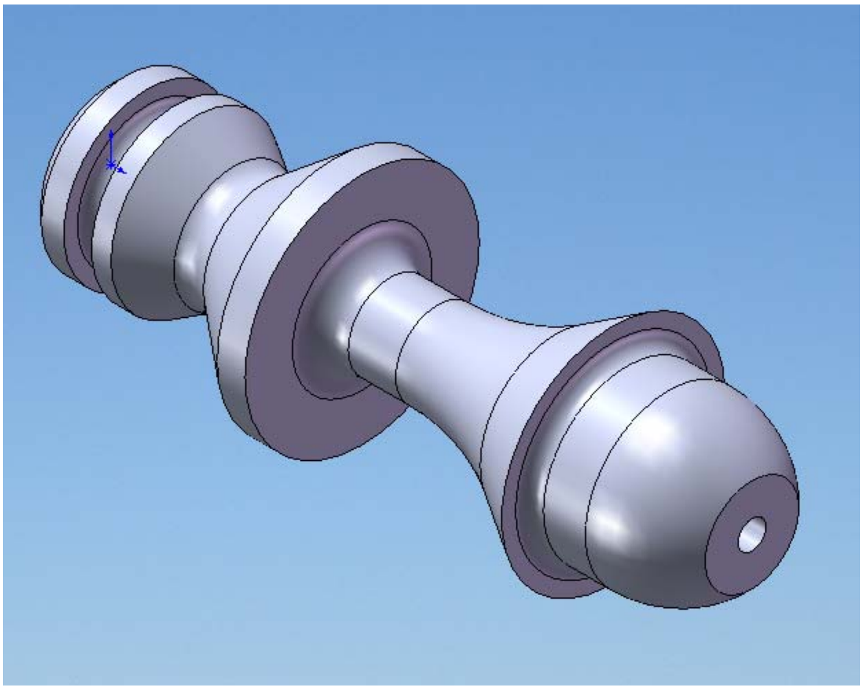

Figure 5.11. Geometry of the Third Sample

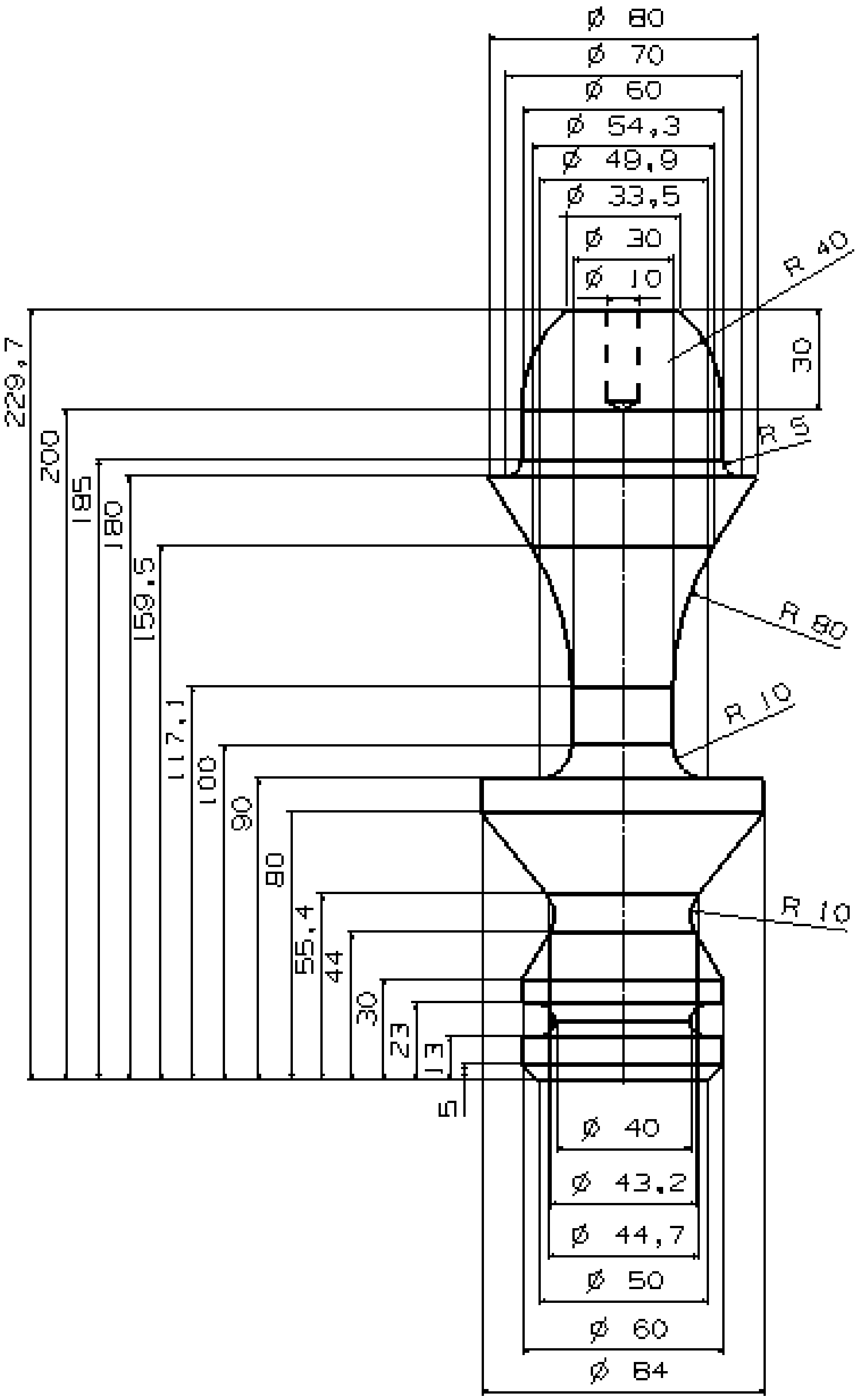

Figure 5.12. Dimensions of the Third Sample

5.3.2 STEP Part 21 Physical File Output

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ISO-10303-21;

HEADER;

FILE_DESCRIPTION(('ISO-10303-224,FEATURE BASED ROTATIONAL PART
CREATED BY',

'TurnDESIGN','COPYRIGHT','KADiR AKKUŞ'),'1');

FILE_NAME(('thesis_sample_3.part21','2011-6-3T01:53:47',

('KADİR AKKUŞ','METU-ME','ANKARA TURKİYE'),('METU-ME','ANKARA
TURKİYE'),

'TurnDESIGN VERSION1','TurnDESIGN','APPROVED BY S.ENGİN KILIÇ');

FILE_SCHEMA(('ROTATIONAL PART', 'PART FOR 2-AXIS TURNING'));

ENDSEC;

DATA;

#1=OUTER_DIAMETER('OD1',#2,#3,#4,$);

#2=ORIENTATION((0,0,0),(1,0,0,0,1,0,0,0,1));

#3=NUMERIC_PARAMETER('OD1 DIAMETER',MM,60,#5);

#4=NUMERIC_PARAMETER('OD1 LENGTH',MM,30,#6);

#5=TOLERANCE_RANGE(0.01,0.01,3);

#6=TOLERANCE_RANGE(0.01,0.01,3);

#7=OUTER_DIAMETER('OD2',#8,#9,#10,$);

#8=ORIENTATION((0,0,30),(1,0,0,0,1,0,0,0,1));

#9=NUMERIC_PARAMETER('OD2 DIAMETER',MM,84,#11);

#10=NUMERIC_PARAMETER('OD2 LENGTH',MM,60,#12);

#11=TOLERANCE_RANGE(0.01,0.01,3);

#12=TOLERANCE_RANGE(0.01,0.01,3);

#13=OUTER_DIAMETER('OD3',#14,#15,#16,$);

#14=ORIENTATION((0,0,90),(1,0,0,0,1,0,0,0,1));
```

```
#15=NUMERIC_PARAMETER('OD3 DIAMETER',MM,80,#17);

#16=NUMERIC_PARAMETER('OD3 LENGTH',MM,90,#18);

#17=TOLERANCE_RANGE(0.01,0.01,3);

#18=TOLERANCE_RANGE(0.01,0.01,3);

#19=OUTER_DIAMETER('OD4',#20,#21,#22,$);

#20=ORIENTATION((0,0,180),(1,0,0,0,1,0,0,0,1));

#21=NUMERIC_PARAMETER('OD4 DIAMETER',MM,60,#23);

#22=NUMERIC_PARAMETER('OD4 LENGTH',MM,20,#24);

#23=TOLERANCE_RANGE(0.01,0.01,3);

#24=TOLERANCE_RANGE(0.01,0.01,3);

#25=REVOLVED_ROUND('REVRND1',#26,#27,#28,#29);

#26=ORIENTATION((0,0,200),(1,0,0,0,1,0,0,0,1));

#27=DIRECTION(-0.9135454,0,-0.4067366);

#28=NUMERIC_PARAMETER('REVRND1 REVOLVED FEATURE
RADIUS',MM,16.76522,#32);

#29=PARTIAL_CIRCULAR_PROFILE($,#30,#31);

#30=NUMERIC_PARAMETER('REVRND1 SWEEP ANGLE',DEGREES,48,#33);

#31=NUMERIC_PARAMETER('REVRND1 SWEEP RADIUS',MM,40,#34);

#32=TOLERANCE_RANGE(0.01,0.01,3);

#33=TOLERANCE_RANGE(0.01,0.01,3);

#34=TOLERANCE_RANGE(0.01,0.01,3);

#35=ROUND_HOLE('RNDHOL1',#36,$,#37,#38,$,#39);

#36=ORIENTATION((0,0,229.7258),(1,0,0,0,1,0,0,0,1));

#37=LINEAR_PATH(#40,#41);

#38=NUMERIC_PARAMETER('RNDHOL1 DIAMETER',MM,10,#42);

#39=CONICAL_HOLE_BOTTOM(#44);

#40=NUMERIC_PARAMETER('RNDHOL1 DISTANCE',MM,30,#43);
```

```
#41=DIRECTION(0,0,-1);

#42=TOLERANCE_RANGE(0.01,0.01,3);

#43=TOLERANCE_RANGE(0.01,0.01,3);

#44=NUMERIC_PARAMETER('RNDHOL1 CONICAL BOTTOM TIP
ANGLE',MM,118,#45);

#45=TOLERANCE_RANGE(0.01,0.01,3);

#46=NUMERIC_PARAMETER('RNDHOL1 CONICAL BOTTOM TIP
RADIUS',MM,1,#47);

#47=TOLERANCE_RANGE(0.01,0.01,3);

#48=CHAMFER('CHAMFER1',#49,#50);

#49=FIRST_OFFSET('OD1',#51);

#50=SECOND_CHAMFER_OFFSET('EDGE1',#52);

#51=NUMERIC_PARAMETER('CHAMFER1 FIRST_OFFSET',MM,5,#53);

#52=CHAMFER_ANGLE(#54);

#53=TOLERANCE_RANGE(0.01,0.01,3);

#54=NUMERIC_PARAMETER('CHAMFER1 ANGLE',DEGREES,45,#55);

#55=TOLERANCE_RANGE(0.01,0.01,3);

#56=GROOVE('RNDUGRV1',#57,#58,#59,#60);

#57=ORIENTATION((20,0,18),(1,0,0,0,1,0,0,0,1));

#58=DIRECTION(-1,0,0);

#59=NUMERIC_PARAMETER('RNDUGRV1 GROOVE RADIUS',MM,20,#61);

#60=ROUNDED_U_PROFILE($,#62);

#61=TOLERANCE_RANGE(0.01,0.01,3);

#62=NUMERIC_PARAMETER('RNDUGRV1 WIDTH',MM,10,#63);

#63=TOLERANCE_RANGE(0.01,0.01,3);

#64=GROOVE('VEEGRV1',#65,#66,#67,#68);

#65=ORIENTATION((18,0,50),(1,0,0,0,1,0,0,0,1));
```

```
#66=DIRECTION(-1,0,0);

#67=NUMERIC_PARAMETER('VEEGRV1 GROOVE RADIUS',MM,18,#69);

#68=VEE_PROFILE($,#70,#71,#72,$,$);

#69=TOLERANCE_RANGE(0.01,0.01,3);

#70=NUMERIC_PARAMETER('VEEGRV1 PROFILE RADIUS',MM,10,#73);

#71=NUMERIC_PARAMETER('VEEGRV1 PROFILE
ANGLE',DEGREES,110.383,#74);

#72=NUMERIC_PARAMETER('VEEGRV1 TILT ANGLE',DEGREES,38.667,#75);

#73=TOLERANCE_RANGE(0.01,0.01,3);

#74=TOLERANCE_RANGE(0.01,0.01,3);

#75=TOLERANCE_RANGE(0.01,0.01,3);

#76=GROOVE('SQRUGRV1',#77,#78,#79,#80);

#77=ORIENTATION((15,0,115),(1,0,0,0,1,0,0,0,1));

#78=DIRECTION(-1,0,0);

#79=NUMERIC_PARAMETER('SQRUGRV1 GROOVE RADIUS',MM,15,#81);

#80=SQUARE_U_PROFILE($,#82,#83,#84,#85,#86);

#81=TOLERANCE_RANGE(0.01,0.01,3);

#82=NUMERIC_PARAMETER('SQRUGRV1 WIDTH',MM,50,#87);

#83=NUMERIC_PARAMETER('SQRUGRV1 FIRST RADIUS',MM,80,#88);

#84=NUMERIC_PARAMETER('SQRUGRV1 FIRST ANGLE',DEGREES,32,#89);

#85=NUMERIC_PARAMETER('SQRUGRV1 SECOND RADIUS',MM,10,#90);

#86=NUMERIC_PARAMETER('SQRUGRV1 SECOND ANGLE',DEGREES,90,#91);

#87=TOLERANCE_RANGE(0.01,0.01,3);

#88=TOLERANCE_RANGE(0.01,0.01,3);

#89=TOLERANCE_RANGE(0.01,0.01,3);

#90=TOLERANCE_RANGE(0.01,0.01,3);

#91=TOLERANCE_RANGE(0.01,0.01,3);
```

```
#92=CONSTANT_RADIUS_FILLET('FILLET1','OD3','OD4',#93,$,$);

#93=NUMERIC_PARAMETER('FILLET1 RADIUS',MM,5,#94);

#94=TOLERANCE_RANGE(0.01,0.01,3);

ENDSEC;

END-ISO-10303-21;
```

5.3.3 CNC Simulator Output

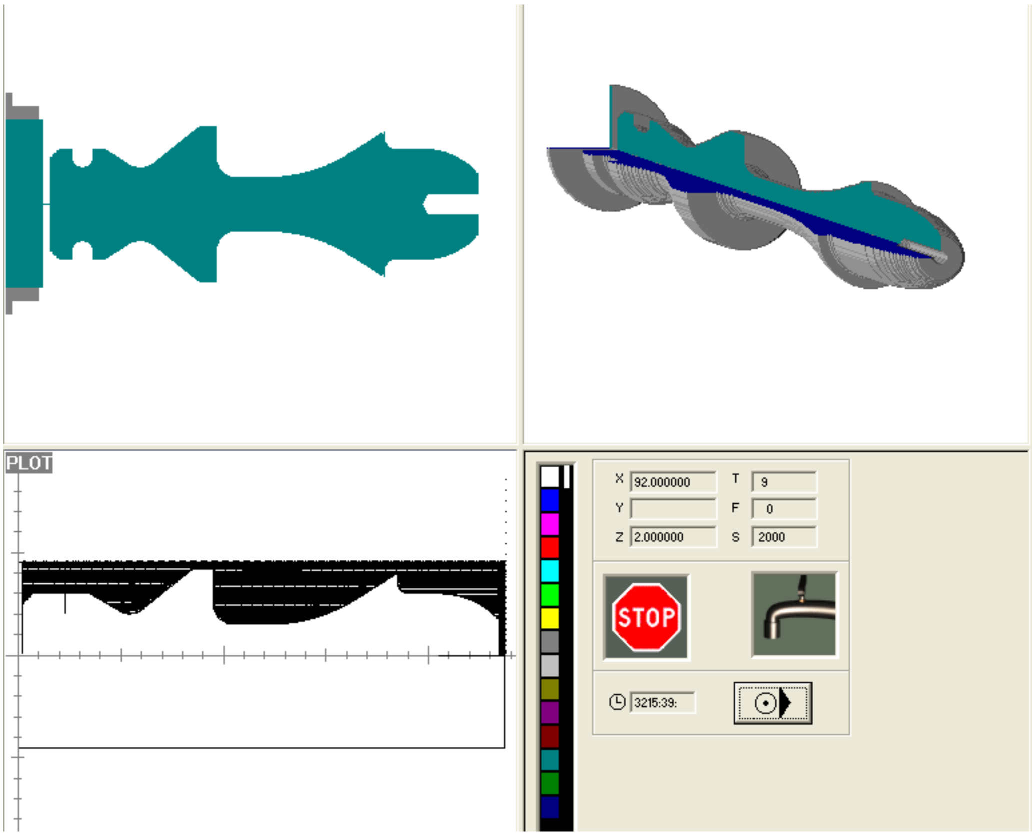

Figure 5.13. CNC Simulator Output for the Third Sample

5.3.4 Tools Used for Machining

CNC Turret

Tool	Tool Name	Tool Type	Kr	Epsilon	Feed (mm/rev)	Depth of Cut (mm)	Speed (rev/min)
T1	External 2	RHS Ext. - Roughing	93	55	0,1	0,7	1500
T2	External 1	RHS Ext. - Finishing	93	55	0,05	0,4	2500
T3	LHS External 2	LHS Ext. - Roughing	93	55	0,5	1	2500
T4	LHS External1	LHS Ext. - Finishing	93	55	0,05	0,5	2500
T5	Center Drill 3mm	Center Drill. - Roughing	3	5	0,2	0,8	3000
T6	Drill 10mm	Drill. - Roughing	118	1,4	1,5	5	2500
T7	Groove	Groove Ext. - Medium	40	3	0,05	1	2000
T8	Special Round	Special Tool (Round 'U')	0	0	0,05	1	2500
T9	not set	-	-	-	-	-	-
T10	not set	-	-	-	-	-	-

Figure 5.14. Tool List for the Third Sample

NOTE: For grooving tools "Kr" and "Epsilon" represent grooving width and grooving depth respectively.

This sample is not produced since the dimensions of the part are larger than the 2-axis CNC Lathe of the machine shop of Mechanical Engineering Department of METU.

CHAPTER 6

CONCLUSIONS AND RECOMMENDATIONS

6.1 Conclusion

This thesis work and the developed system is motivated by the industrial requirements of e-manufacturing paradigm requiring coherent data flow in the CAD-CAM-CNC chain, introduction of STEP based CNC controllers. This thesis work and developed system aimed to fill a gap in this integration, by enabling generation of a machining file for NC-code based CNC lathes directly using a STEP AP224 based product data.

The conclusions of the work can be discussed as:

- In this work, STEP standard is investigated with its documentation structure, data representation schemas, application protocols and implementation techniques. STEP AP224 is selected as the basis for the feature based modeler.
- A STEP AP224 based feature library specialized for rotational part design is built in an object oriented manner. This feature library not only enables geometrical description of a rotational part but also provides opportunity to built product data with high level information such as tolerance data.
- A feature modeler and an embedded preprocessor responsible for generating the neutral STEP file are developed using the STEP AP224 based feature library. The feature modeler, with the visual aid of interlinked SolidWorks application, is used to design a rotational part feature by feature and represent the resulting product data as a STEP

Part 21 physical file, which is the data element provided with high level information in the CAD-CAM-CNC chain.

- An NC-code generating system is developed especially for external turning, grooving and drilling operations of 2-axis turning, employing the STEP Part 21 physical file as the product data and processing this data using the machine tool and cutting tool information from the embedded machine tool and cutting tool libraries, built within this study.
- Usage of a well organized feature library based on STEP AP224 and AP238 is very efficient in both design stage and NC-code generation.

6.2 Recommendations for Future Work

Recommended feature work will be listed separately for different features of the developed system.

<u>Feature Library</u>

- Inner grooves and facial grooves may be added.
- Inner and outer threads may be added.
- Knurling may be added.
- Geometric tolerances may be added.

<u>Feature Modeler</u>

- A standalone geometry display environment may be developed and SolidWorks dependency of the system may be abolished.

<u>Preprocessor</u>

- Preprocessing STEP Part 28 or XML formatted neutral file may be enabled.

NC-code Generator

- Advanced computer engineering methods, such as neural networks and fuzzy logic may be employed to enhance the module and increase automation.
- CNC simulator module may be integrated.
- Canned cycle codes and NC-code standards of different CNC manufacturers may be added as an option to increase versatility.

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APPENDIX A

SAMPLE CODES

A.1. Linking to SolidWorks Application

The rotational part design module is visualized using SolidWorks 2007, for increasing the interactivity of the user interface and aiding product data creation with visual representation of product model. The link between Visual Basic (VB) and SolidWorks (SW) is created by the following codes embedded into VB.

```
'Public variables for SolidWorks Application
Public swApp As SldWorks.SldWorks
Public swModel As SldWorks.ModelDoc2
Public swConfig As SldWorks.Configuration
Public swAppObj As Object
Public SelMgr As Object
Public Part As Object
Public boolstatus As Boolean
Public longstatus As Long, longwarnings As Long
Public Feature As Object
' Connecting to Solidworks from Visual Basic
  On Error Resume Next
    Set swApp = GetObject(, "SldWorks.Application")
    If Err Then
```

```
        Err.Clear
        Set swApp = CreateObject("SldWorks.Application")
        If Err Then
            MsgBox Err.Description
            Exit Sub
        End If
    End If
  Document1.Visible = True
  Set Part = swApp.ActivateDoc2("Part1", False, longstatus)
  Part.ShowNamedView2 "*Isometric", 7
```

A.2. Machining Features Class

Here is presented a sample part from the feature library of the rotational part design module. Rotational machining features extracted from AP224 are created as separate classes and declared as 'public' variables, to be accessed globally in the Visual Basic application.

```
Public OD(1 To 100000) As New outer_diameter
Public ODSHLD(1 To 100000) As New outer_diameter_to_shoulder
Public REVRND(1 To 100000) As New revolved_feature
Public REVFLT(1 To 100000) As New revolved_feature
Public SQRUGRV(1 To 100000) As New groove
Public RNDUGRV(1 To 100000) As New groove
Public CIRCGRV(1 To 100000) As New groove
Public VEEGRV(1 To 100000) As New groove
Public RNDHOL(1 To 100000) As New round_hole
Public CSKHOL(1 To 100000) As New csk_hole
Public CBORHOL(1 To 100000) As New cbor_hole
Public ROUND(1 To 100000) As New edge_round
```

```
Public FILLET(1 To 100000) As New fillet_tran

Public CHAMFER(1 To 100000) As New chamfer_tran
```

A.3. Creation of Part 21 File Segment for Revolved Flat Feature

```
' ****** create STEP part21 file **************

If count_feature = 1 Then Open file For Output As #1

If count_feature > 1 Then Open file For Append As #

file_string = "#" & count_line & "=REVOLVED_FLAT('" &
REVFLT(count_feature).revolved_flat.feature_ID & "',#" & (count_line + 1) &
",#" & (count_line + 2) & ",#" & (count_line + 3) & ",#" & (count_line + 4) &
");"

Print #1, file_string

file_string = "#" & (count_line + 1) & "=ORIENTATION((" &
Replace(REVFLT(count_feature).revolved_flat.placement.location.x, ",", ".") &
"," & Replace(REVFLT(count_feature).revolved_flat.placement.location.y, ",",
".") & "," & Replace(REVFLT(count_feature).revolved_flat.placement.location.z,
",", ".") & "),("

file_string = file_string &
REVFLT(count_feature).revolved_flat.placement.direction.xi & "," &
REVFLT(count_feature).revolved_flat.placement.direction.xj & "," &
REVFLT(count_feature).revolved_flat.placement.direction.xk & ","

file_string = file_string &
REVFLT(count_feature).revolved_flat.placement.direction.yi & "," &
REVFLT(count_feature).revolved_flat.placement.direction.yj & "," &
REVFLT(count_feature).revolved_flat.placement.direction.yk & ","

file_string = file_string &
REVFLT(count_feature).revolved_flat.placement.direction.zi & "," &
REVFLT(count_feature).revolved_flat.placement.direction.zj & "," &
REVFLT(count_feature).revolved_flat.placement.direction.zk & "));"

Print #1, file_string

file_string = "#" & (count_line + 2) & "=DIRECTION(" &
Replace(REVFLT(count_feature).material_side.x, ",", ".") & "," &
Replace(REVFLT(count_feature).material_side.y, ",", ".") & "," &
Replace(REVFLT(count_feature).material_side.z, ",", ".") & ");"
```

```
Print #1, file_string

file_string = "#" & (count_line + 3) & "=NUMERIC_PARAMETER('" & REVFLT(count_feature).radius_at_placement.parameter_name & "'," & REVFLT(count_feature).radius_at_placement.parameter_unit & ","

file_string = file_string & Replace(REVFLT(count_feature).radius_at_placement.parameter_value, ",", ".") & ",#" & (count_line + 6) & ");"

Print #1, file_string

file_string = "#" & (count_line + 4) & "=LINEAR_PROFILE($" & ",#" & (count_line + 5) & ");"

Print #1, file_string

file_string = "#" & (count_line + 5) & "=NUMERIC_PARAMETER('" & REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.parameter_name & "'," & REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.parameter_unit & ","

file_string = file_string & Replace(REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.parameter_value, ",", ".") & ",#" & (count_line + 7) & ");"

Print #1, file_string

If REVFLT(count_feature).radius.implicit_tolerance.tolerance_select = tolerance_range_type Then file_bool_word = "TOLERANCE_RANGE": tol1 = REVFLT(count_feature).radius.implicit_tolerance.tolerance_range.upper_range: tol2 = REVFLT(count_feature).radius.implicit_tolerance.tolerance_range.lower_range: tol3 = REVFLT(count_feature).radius.implicit_tolerance.tolerance_range.significant_digits:

If REVFLT(count_feature).radius.implicit_tolerance.tolerance_select = plus_minus_value_type Then file_bool_word = "PLUS_MINUS_VALUE": tol1 = REVFLT(count_feature).radius.implicit_tolerance.plus_minus_value.upper_limit : tol2 = REVFLT(count_feature).radius.implicit_tolerance.plus_minus_value.lower_limit : tol3 =
```

REVFLT(count_feature).radius.implicit_tolerance.plus_minus_value.significant_digits:

If REVFLT(count_feature).radius.implicit_tolerance.tolerance_select = limits_and_fits_type Then

file_bool_word = "LIMITS_AND_FITS": tol1 = REVFLT(count_feature).radius.implicit_tolerance.limits_and_fits.deviation: tol2 = REVFLT(count_feature).radius.implicit_tolerance.limits_and_fits.grade

If REVFLT(count_feature).radius.implicit_tolerance.limits_and_fits.fit_type = SHAFT Then tol3 = ".SHAFT."

If REVFLT(count_feature).radius.implicit_tolerance.limits_and_fits.fit_type = Hole Then tol3 = ".HOLE."

End If

tol1 = Replace(tol1, ",", "."): tol2 = Replace(tol2, ",", "."): tol3 = Replace(tol3, ",", ".")

file_string = "#" & (count_line + 6) & "=" & file_bool_word & "(" & tol1 & "," & tol2 & "," & tol3 & ");"

Print #1, file_string

If REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.tolerance_select = tolerance_range_type Then file_bool_word = "TOLERANCE_RANGE": tol1 = REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.tolerance_range.upper_range: tol2 = REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.tolerance_range.lower_range: tol3 = REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.tolerance_range.significant_digits:

If REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.tolerance_select = plus_minus_value_type Then file_bool_word = "PLUS_MINUS_VALUE": tol1 = REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.plus_minus_value.upper_limit: tol2 = REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.plus_minus_value.lower_limit: tol3 =

```
REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.plus_minus_value.significant_digits:

If REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.tolerance_select = limits_and_fits_type Then

file_bool_word = "LIMITS_AND_FITS": tol1 = REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.limits_and_fits.deviation: tol2 = REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.limits_and_fits.grade

If REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.limits_and_fits.fit_type = SHAFT Then tol3 = ".SHAFT."

If REVFLT(count_feature).revolved_flat.flat_edge_shape.profile_length.implicit_tolerance.limits_and_fits.fit_type = Hole Then tol3 = ".HOLE."

End If

tol1 = Replace(tol1, ",", "."): tol2 = Replace(tol2, ",", "."): tol3 = Replace(tol3, ",", ".")

file_string = "#" & (count_line + 7) & "=" & file_bool_word & "(" & tol1 & "," & tol2 & "," & tol3 & ");"

Print #1, file_string

count_line = count_line + 8

Close #1
```

A.4. Creation of Model in SolidWorks for Revolved Flat Feature

```
' ****** create feature using SolidWorks

p1(0) = p1(0) / measure.convertion_factor

p1(1) = p1(1) / measure.convertion_factor

p1(2) = p1(2) / measure.convertion_factor

p2(0) = p2(0) / measure.convertion_factor

p2(1) = p2(1) / measure.convertion_factor
```

```
p2(2) = p2(2) / measure.convertion_factor

p3(0) = p3(0) / measure.convertion_factor

p3(1) = p3(1) / measure.convertion_factor

p3(2) = p3(2) / measure.convertion_factor

p4(0) = p4(0) / measure.convertion_factor

p4(1) = p4(1) / measure.convertion_factor

p4(2) = p4(2) / measure.convertion_factor


count_sketch(count_feature) = last_sketch_no

count_rev(count_feature) = last_rev_no

REVFLT(count_feature).ID_sketch = "3DSketch" &
count_sketch(count_feature)

Form_Revolved_Flat.Visible = False

On Error Resume Next

Part.Insert3DSketch

Part.SketchManager.CreateSketchPlane 0, 0, 0

boolstatus = Part.Extension.SelectByID2("Plane1", "SKETCHSURFACES", 0, 0,
0, False, 0, Nothing, 0)

Part.ActivateSelectedFeature

Part.CreateLine2 p1(0), p1(1), p1(2), p2(0), p2(1), p2(2)

Part.CreateLine2 p2(0), p2(1), p2(2), p3(0), p3(1), p3(2)

Part.CreateLine2 p3(0), p3(1), p3(2), p4(0), p4(1), p4(2)

Part.CreateLine2 p4(0), p4(1), p4(2), p1(0), p1(1), p1(2)

Part.ClearSelection2 True

Part.SketchManager.InsertSketch True

boolstatus = Part.Extension.SelectByID2(REVFLT(count_feature).ID_sketch,
"SKETCH", 0, 0, 0, False, 0, Nothing, 0)

boolstatus = Part.Extension.SelectByID2("Plane1@" &
REVFLT(count_feature).ID_sketch, "EXTSKETCHSURFACES", 0, 0, 0, False, 0,
Nothing, 0)
```

```
boolstatus = Part.Extension.SelectByID2("Line4@" & REVFLT(count_feature).ID_sketch, "EXTSKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

Part.ClearSelection2 True

boolstatus = Part.Extension.SelectByID2("Plane1@" & REVFLT(count_feature).ID_sketch, "EXTSKETCHSURFACES", 0, 0, 0, False, 0, Nothing, 0)

boolstatus = Part.Extension.SelectByID2("Line4@" & REVFLT(count_feature).ID_sketch, "EXTSKETCHSEGMENT", 0, 0, 0, True, 4, Nothing, 0)

boolstatus = Part.Extension.SelectByID2(REVFLT(count_feature).ID_sketch, "SKETCH", 0, 0, 0, True, 0, Nothing, 0)

Part.FeatureManager.FeatureRevolve 6.28318530718, False, 0, 0, 0, 1, 1, 1

Part.SelectionManager.EnableContourSelection = 0
```

A.5. Cutting Tool Change

```
Private Function tool_change_NC(ByVal row_cnt As Integer, ByVal tl_no As Integer, ByVal finish_cut_control As Boolean) As Integer

    turn.tool_change = False

    If turn.prv_tl_no <> tl_no Then

            turn.tool_change = True

            Print #1, "N" & row_cnt & " G00 X" & Replace(str(turn.start_x), ",", ".") & " Z" & Replace(str(calc_z_pos(turn.start_z)), ",", "."): row_cnt = row_cnt + 5

        If for_sim_flag = True Then

            Print #1, "N" & row_cnt & " T" & tool_file_no(tl_no) & " M06": row_cnt = row_cnt + 5

        ElseIf for_sim_flag = False Then

            Print #1, "N" & row_cnt & " G91 G28 X 0 Z 0": row_cnt = row_cnt + 5

            Print #1, "N" & row_cnt & " G90": row_cnt = row_cnt + 5

            Print #1, "N" & row_cnt & " T" & tl_no & " M06": row_cnt = row_cnt + 5
```

```
        End If
        If tool_rad_comp = True And finish_cut_control = True Then
            If Mid(tl_tool_type(tl_no), 1, 3) = "RHS" Then Print #1, "N" & row_cnt
& " G42": row_cnt = row_cnt + 5
            If Mid(tl_tool_type(tl_no), 1, 3) = "LHS" Then Print #1, "N" & row_cnt
& " G41": row_cnt = row_cnt + 5
        ElseIf tool_rad_comp = False Or finish_cut_control = False Then
            Print #1, "N" & row_cnt & " G40": row_cnt = row_cnt + 5
        End If
        If const_surf_spd = True Then Print #1, "N" & row_cnt & " G96 S" &
spindle_speed(tl_no): row_cnt = row_cnt + 5
        If const_surf_spd = False Then Print #1, "N" & row_cnt & " G97 S" &
spindle_speed(tl_no): row_cnt = row_cnt + 5
    End If
            If coolant_on_state = True And coolant_on = False Then
            Print #1, "N" & row_cnt & " M09": row_cnt = row_cnt + 5
            coolant_on_state = False
        ElseIf coolant_on_state = False And coolant_on = True Then
            Print #1, "N" & row_cnt & " M08": row_cnt = row_cnt + 5
            coolant_on_state = True
        End If
        tool_change_NC = row_cnt


End Function
```

APPENDIX B

NC-CODE OUTPUTS FOR SAMPLE PARTS

B.1. First Sample Output

```
;!!!This code is generated by 'TurnDESIGN' for 'thesis_sample_1.part21'
N5 G291
N10 G54
N15 G21
N20 G99
N25 G91 G28 X 0 Z 0
N30 G90
N35 T1 M06
N40 G40
N45 G97 S1500
N50 M03
N55 G00 X 34 Z 72.467
N60 G01 X-1 Z 72.467 F .1
N65 G00 X 34 Z 72.467
N70 G00 X 34 Z 71.934
N75 G01 X-1 Z 71.934 F .1
N80 G00 X 34 Z 71.934
N85 G00 X 34 Z 71.401
N90 G01 X-1 Z 71.401 F .1
N95 G00 X 34 Z 71.401
N100 G91 G28 X 0 Z 0
N105 G90
N110 T2 M06
N115 G42
N120 G97 S2500
N125 G00 X 34 Z 71
N130 G01 X-1 Z 71 F .05
N135 G91 G28 X 0 Z 0
N140 G90
N145 T1 M06
N150 G40
N155 G97 S1500
N160 G00 X 30.9 Z 71.5
N165 G01 X 30.9 Z 2 F .1
N170 G01 X 32.5 Z 2
N175 G00 X 32.5 Z 71.5
N180 G00 X 29.8 Z 71.5
N185 G01 X 29.8 Z 2 F .1
N190 G01 X 31.4 Z 2
N195 G00 X 31.4 Z 71.5
N200 G00 X 28.4 Z 71.5
```

N205 G01 X 28.4 Z 41.4

N210 G01 X 30.3 Z 41.4

N215 G00 X 30.3 Z 71.5

N220 G00 X 27 Z 71.5

N225 G01 X 27 Z 41.4

N230 G01 X 28.9 Z 41.4

N235 G00 X 28.9 Z 71.5

N240 G00 X 26.8 Z 71.5

N245 G01 X 26.8 Z 41.4

N250 G01 X 28.7 Z 41.4

N255 G00 X 28.7 Z 71.5

N260 G00 X 25.4 Z 71.5

N265 G01 X 25.4 Z 41.408

N270 G02 X 26 Z 41.4 R 5.6

N275 G00 X 26 Z 71.5

N280 G00 X 24 Z 71.5

N285 G01 X 24 Z 41.49

N290 G02 X 25.4 Z 41.408 R 5.6

N295 G00 X 25.4 Z 71.5

N300 G00 X 22.6 Z 71.5

N305 G01 X 22.6 Z 41.664

N310 G02 X 24 Z 41.49 R 5.6

N315 G00 X 24 Z 71.5

N320 G00 X 21.2 Z 71.5

N325 G01 X 21.2 Z 41.94

N330 G02 X 22.6 Z 41.664 R 5.6

N335 G00 X 22.6 Z 71.5

N340 G00 X 19.8 Z 71.5

N345 G01 X 19.8 Z 42.336

N350 G02 X 21.2 Z 41.94 R 5.6

N355 G00 X 21.2 Z 71.5

N360 G00 X 18.4 Z 71.5

N365 G01 X 18.4 Z 42.887

N370 G02 X 19.8 Z 42.336 R 5.6

N375 G00 X 19.8 Z 71.5

N380 G00 X 17 Z 71.5

N385 G01 X 17 Z 43.667

N390 G02 X 18.4 Z 42.887 R 5.6

N395 G00 X 18.4 Z 71.5

N400 G00 X 15.6 Z 71.5

N405 G01 X 15.6 Z 44.922

N410 G02 X 17 Z 43.667 R 5.6

N415 G00 X 17 Z 71.5

N420 G00 X 13.39 Z 71.5

N425 G01 X 13.39 Z 54.565

N430 G01 X 15.29 Z 49.815

N435 G00 X 15.29 Z 71.5

N440 G00 X 11.99 Z 71.5

N445 G01 X 11.99 Z 58.065

N450 G01 X 13.89 Z 53.315

N455 G00 X 13.89 Z 71.5

N460 G00 X 10.59 Z 71.5

N465 G01 X 10.59 Z 61.565

N470 G01 X 12.49 Z 56.815

N475 G00 X 12.49 Z 71.5

N480 G00 X 34 Z 74

N480 G91 G28 X 0 Z 0

N485 G90

N490 T2 M06

N495 G42

N500 G97 S2500

N505 G00 X 29 Z 71.5

```
N510 G01 X 10 Z 71 F .05
N515 G01 X 10 Z 61
N520 G01 X 14 Z 51
N525 G01 X 14 Z 47
N530 G02 X 26 Z 41 R 6
N535 G01 X 29 Z 41
N540 G01 X 29 Z 2
N545 G01 X 29 Z 2
N550 G01 X 34 Z 2
N555 G00 X 34 Z 2
N560 G91 G28 X 0 Z 0
N565 G90
N570 T1 M06
N575 G40
N580 G97 S1500
N585 G00 X 34 Z 74
N590 G00 X 34 Z 2
N595 G00 X 29.4 Z 37.046
N600 G01 X 28 Z 35.834
N605 G01 X 28 Z 10.4
N610 G01 X 29.4 Z 10.4
N615 G00 X 28.2 Z 36.007
N620 G01 X 26.6 Z 34.622
N625 G01 X 26.6 Z 10.4
N630 G01 X 28.2 Z 10.4
N635 G00 X 26.8 Z 34.795
N640 G01 X 25.2 Z 33.409
N645 G01 X 25.2 Z 10.4
N650 G01 X 26.8 Z 10.4
N655 G00 X 25.4 Z 33.582
N660 G01 X 23.8 Z 32.197
N665 G01 X 23.8 Z 10.4
N670 G01 X 25.4 Z 10.4
N675 G00 X 24 Z 32.37
N680 G01 X 22.4 Z 30.984
N685 G01 X 22.4 Z 10.4
N690 G01 X 24 Z 10.4
N695 G00 X 22.6 Z 31.158
N700 G01 X 21 Z 29.772
N705 G01 X 21 Z 10.4
N710 G01 X 22.6 Z 10.4
N715 G00 X 21.2 Z 29.945
N720 G01 X 20.8 Z 29.599
N725 G01 X 20.8 Z 10.4
N730 G01 X 21.2 Z 10.4
N735 G00 X 34 Z 10.4
N740 G91 G28 X 0 Z 0
N745 G90
N750 T3 M06
N755 G40
N760 G97 S2500
N765 G00 X 34 Z 74
N770 G00 X 34 Z 29.599
N775 G00 X 29.4 Z 37.346
N780 G00 X 28 Z 35.634
N785 G01 X 28 Z 37.1
N790 G01 X 29.4 Z 37.1
N795 G00 X 28 Z 36.134
N800 G00 X 26.6 Z 34.422
N805 G01 X 26.6 Z 37.1
N810 G01 X 28 Z 37.1
N815 G00 X 26.6 Z 34.922
```

```
N820 G00 X 25.2 Z 33.209
N825 G01 X 25.2 Z 37.1
N830 G01 X 26.6 Z 37.1
N835 G00 X 25.2 Z 33.709
N840 G00 X 23.8 Z 31.997
N845 G01 X 23.8 Z 37.1
N850 G01 X 25.2 Z 37.1
N855 G00 X 23.8 Z 32.497
N860 G00 X 22.4 Z 30.784
N865 G01 X 22.4 Z 37.1
N870 G01 X 23.8 Z 37.1
N875 G00 X 22.4 Z 31.284
N880 G00 X 21 Z 29.572
N885 G01 X 21 Z 37.1
N890 G01 X 22.4 Z 37.1
N895 G00 X 21 Z 30.072
N900 G00 X 20.8 Z 29.399
N905 G01 X 20.8 Z 37.1
N910 G01 X 21 Z 37.1
N915 G00 X 34 Z 29.399
N920 G91 G28 X 0 Z 0
N925 G90
N930 T2 M06
N935 G42
N940 G97 S2500
N945 G00 X 34 Z 74
N950 G00 X 34 Z 37.1
N955 G00 X 29.9 Z 37.5
N960 G01 X 29 Z 37.5
N965 G01 X 20 Z 29.706
N970 G01 X 20 Z 10
N975 G01 X 29 Z 10
N980 G00 X 34 Z 10
N985 G91 G28 X 0 Z 0
N990 G90
N995 T4 M06
N1000 G41
N1005 G97 S2500
N1010 G00 X 34 Z 74
N1015 G00 X 34 Z 37.1
N1020 G00 X 20.9 Z 29.706
N1025 G01 X 20 Z 29.706
N1030 G01 X 20 Z 37.5
N1035 G01 X 29 Z 37.5
N1040 G00 X 34 Z 37.5
N1045 G91 G28 X 0 Z 0
N1050 G90
N1055 T5 M06
N1060 G40
N1065 G97 S3000
N1070 G00 X 0 Z 74
N1075 G01 X 0 Z 64.5
N1080 G00 X 0 Z 74
N1085 G91 G28 X 0 Z 0
N1090 G90
N1095 T6 M06
N1100 G40
N1105 G97 S3000
N1110 G00 X 0 Z 74
N1115 G01 X 0 Z 61
N1120 G00 X 0 Z 74
```

B.2. Second Sample Output

```
;!!!This code is generated by 'TurnDESIGN' for 'thesis_sample_2.part21'
N5 G291
N10 G54
N15 G21
N20 G99
N25 G91 G28 X 0 Z 0
N30 G90
N35 T1 M06
N40 G40
N45 G97 S1500
N50 M03
N55 G00 X 34 Z 70.368
N60 G01 X-1 Z 70.368 F .1
N65 G00 X 34 Z 70.368
N70 G00 X 34 Z 69.736
N75 G01 X-1 Z 69.736 F .1
N80 G00 X 34 Z 69.736
N85 G91 G28 X 0 Z 0
N90 G90
N95 T2 M06
N100 G42
N105 G97 S2500
N110 G00 X 34 Z 69.337
N115 G01 X-1 Z 69.337 F .05
N120 G91 G28 X 0 Z 0
N125 G90
N130 T1 M06
N135 G40
N140 G97 S1500
N145 G00 X 30.933 Z 69.837
N150 G01 X 30.933 Z 2 F .1
N155 G01 X 32.5 Z 2
N160 G00 X 32.5 Z 69.837
N165 G00 X 29.866 Z 69.837
N170 G01 X 29.866 Z 2 F .1
N175 G01 X 31.433 Z 2
N180 G00 X 31.433 Z 69.837
N185 G00 X 28.799 Z 69.837
N190 G01 X 28.799 Z 2 F .1
N195 G01 X 30.366 Z 2
N200 G00 X 30.366 Z 69.837
N205 G00 X 27.399 Z 69.837
N210 G01 X 27.399 Z 10.382
N215 G03 X 28 Z 8.4 R 4.4
N220 G00 X 28 Z 69.837
N225 G00 X 27.442 Z 69.837
N230 G01 X 27.442 Z 10.348
N235 G03 X 27.399 Z 10.382 R 4.4
N240 G00 X 27.399 Z 69.837
N245 G00 X 26.042 Z 69.837
N250 G01 X 26.042 Z 32.637
N255 G01 X 27.942 Z 32.637
N260 G00 X 27.942 Z 69.837
N265 G00 X 25.8 Z 69.837
N270 G01 X 25.8 Z 32.637
N275 G01 X 27.7 Z 32.637
N280 G00 X 27.7 Z 69.837
```

N285 G00 X 24.4 Z 69.837

N290 G01 X 24.4 Z 32.665

N295 G02 X 25 Z 32.637 R 1.6

N300 G00 X 25 Z 69.837

N305 G00 X 23 Z 69.837

N310 G01 X 23 Z 32.988

N315 G02 X 24.4 Z 32.665 R 1.6

N320 G00 X 24.4 Z 69.837

N325 G00 X 20.79 Z 69.837

N330 G01 X 20.79 Z 62.637

N335 G01 X 22.69 Z 62.637

N340 G00 X 22.69 Z 69.837

N345 G00 X 19.39 Z 69.837

N350 G01 X 19.39 Z 62.637

N355 G01 X 21.29 Z 62.637

N360 G00 X 21.29 Z 69.837

N365 G00 X 17.99 Z 69.837

N370 G01 X 17.99 Z 64.408

N375 G01 X 19.89 Z 63.458

N380 G00 X 19.89 Z 69.837

N385 G00 X 17.8 Z 69.837

N390 G01 X 17.8 Z 64.503

N395 G01 X 19.7 Z 63.553

N400 G00 X 19.7 Z 69.837

N405 G00 X 16.4 Z 69.837

N410 G01 X 16.4 Z 64.737

N415 G01 X 18.3 Z 64.737

N420 G00 X 18.3 Z 69.837

N425 G00 X 15 Z 69.837

N430 G01 X 15 Z 64.737

N435 G01 X 16.9 Z 64.737

N440 G00 X 16.9 Z 69.837

N445 G00 X 13.6 Z 69.837

N450 G01 X 13.6 Z 64.737

N455 G01 X 15.5 Z 64.737

N460 G00 X 15.5 Z 69.837

N465 G00 X 12.2 Z 69.837

N470 G01 X 12.2 Z 64.737

N475 G01 X 14.1 Z 64.737

N480 G00 X 14.1 Z 69.837

N485 G00 X 10.8 Z 69.837

N490 G01 X 10.8 Z 64.737

N495 G01 X 12.7 Z 64.737

N500 G00 X 12.7 Z 69.837

N505 G00 X 10.8 Z 69.837

N510 G01 X 10.8 Z 64.737

N515 G01 X 12.7 Z 64.737

N520 G00 X 12.7 Z 69.837

N525 G00 X 9.4 Z 69.837

N530 G01 X 9.4 Z 66.996

N535 G03 X 10 Z 64.737 R 5.4

N540 G00 X 10 Z 69.837

N545 G00 X 8 Z 69.837

N550 G01 X 8 Z 67.965

N555 G03 X 9.4 Z 66.996 R 5.4

N560 G00 X 9.4 Z 69.837

N565 G00 X 6.6 Z 69.837

N570 G01 X 6.6 Z 68.611

N575 G03 X 8 Z 67.965 R 5.4

N580 G00 X 8 Z 69.837

N585 G00 X 5.2 Z 69.837

N590 G01 X 5.2 Z 69.07

N595 G03 X 6.6 Z 68.611 R 5.4

N600 G00 X 6.6 Z 69.837

N605 G00 X 3.8 Z 69.837

N610 G01 X 3.8 Z 69.392

N615 G03 X 5.2 Z 69.07 R 5.4

N620 G00 X 5.2 Z 69.837

N625 G00 X 2.4 Z 69.837

N630 G01 X 2.4 Z 69.602

N635 G03 X 3.8 Z 69.392 R 5.4

N640 G00 X 3.8 Z 69.837

N645 G00 X 1 Z 69.837

N650 G01 X 1 Z 69.714

N655 G03 X 2.4 Z 69.602 R 5.4

N660 G00 X 2.4 Z 69.837

N665 G00 X .8 Z 69.837

N670 G01 X .8 Z 69.722

N675 G03 X 1 Z 69.714 R 5.4

N680 G00 X 1 Z 69.837

N685 G00 X 34 Z 72

N685 G91 G28 X 0 Z 0

N690 G90

N695 T2 M06

N700 G42

N705 G97 S2500

N710 G00 X 28 Z 69.837

N715 G01 X 0 Z 69.337 F .05

N720 G03 X 10 Z 64.337 R 5

N725 G01 X 17 Z 64.337

N730 G01 X 19 Z 63.337

N735 G01 X 19 Z 62.237

N740 G01 X 21 Z 62.237

N745 G01 X 21 Z 34.237

N750 G02 X 25 Z 32.237 R 2

N755 G01 X 26.6 Z 32.237

N760 G01 X 26.6 Z 10.237

N765 G01 X 26.632 Z 10.237

N770 G03 X 28 Z 8 R 4

N775 G01 X 28 Z 5

N780 G01 X 28 Z 2

N785 G01 X 34 Z 2

N790 G00 X 34 Z 5

N795 G91 G28 X 0 Z 0

N800 G90

N805 T1 M06

N810 G40

N815 G97 S1500

N820 G00 X 34 Z 72

N825 G00 X 34 Z 5

N830 G00 X 27 Z 34.572

N835 G01 X 25.6 Z 31.767

N840 G01 X 25.6 Z 10.629

N845 G02 X 27 Z 10.616 R 2.8

N850 G00 X 25.8 Z 32.168

N855 G01 X 24.2 Z 28.963

N860 G01 X 24.2 Z 10.825

N865 G02 X 25.8 Z 10.616 R 2.8

N870 G00 X 24.4 Z 29.363

N875 G01 X 22.8 Z 26.158

N880 G01 X 22.8 Z 11.255

N885 G02 X 24.4 Z 10.785 R 2.8

N890 G00 X 23 Z 26.558

N895 G01 X 21.4 Z 23.353

```
N900 G01 X 21.4 Z 12.139
N905 G02 X 23 Z 11.175 R 2.8
N910 G00 X 21.6 Z 23.682
N915 G02 X 20.8 Z 19.742 R 19.6
N920 G01 X 20.8 Z 13.399
N925 G02 X 21.6 Z 11.958 R 2.8
N930 G00 X 34 Z 13.399
N935 G00 X 27.5 Z 35.423
N940 G01 X 26.6 Z 35.423
N945 G01 X 21.19 Z 24.585
N950 G02 X 20 Z 19.742 R 20
N955 G01 X 20 Z 13.4
N960 G02 X 26.4 Z 10.2 R 3.2
N965 G01 X 28 Z 10.2
N970 G00 X 34 Z 10.2
N975 G00 X 21.4 Z 63.081
N980 G01 X 20 Z 61.868
N985 G01 X 20 Z 39.482
N990 G01 X 21.4 Z 38.591
N995 G00 X 20.2 Z 62.041
N1000 G01 X 18.6 Z 60.656
N1005 G01 X 18.6 Z 40.373
N1010 G01 X 20.2 Z 39.355
N1015 G00 X 18.8 Z 60.829
N1020 G01 X 17.2 Z 59.443
N1025 G01 X 17.2 Z 41.265
N1030 G01 X 18.8 Z 40.246
N1035 G00 X 17.4 Z 59.616
N1040 G01 X 15.8 Z 58.231
N1045 G01 X 15.8 Z 42.156
N1050 G01 X 17.4 Z 41.137
N1055 G00 X 16 Z 58.404
N1060 G01 X 14.4 Z 57.018
N1065 G01 X 14.4 Z 43.047
N1070 G01 X 16 Z 42.028
N1075 G00 X 14.6 Z 57.192
N1080 G01 X 13 Z 55.806
N1085 G01 X 13 Z 43.938
N1090 G01 X 14.6 Z 42.92
N1095 G00 X 13.2 Z 55.979
N1100 G01 X 11.6 Z 54.594
N1105 G01 X 11.6 Z 44.829
N1110 G01 X 13.2 Z 43.811
N1115 G00 X 11.8 Z 54.767
N1120 G01 X 10.8 Z 53.901
N1125 G01 X 10.8 Z 46.237
N1130 G02 X 11.8 Z 44.704 R 2.6
N1135 G00 X 34 Z 46.237
N1140 G91 G28 X 0 Z 0
N1145 G90
N1150 T3 M06
N1155 G40
N1160 G97 S2500
N1165 G00 X 34 Z 72
N1170 G00 X 34 Z 53.901
N1175 G00 X 21.4 Z 63.381
N1180 G00 X 20 Z 61.668
N1185 G01 X 20 Z 62.2
N1190 G01 X 21.4 Z 63.134
N1195 G00 X 20 Z 62.168
N1200 G00 X 18.6 Z 60.456
N1205 G01 X 18.6 Z 61.267
```

```
N1210 G01 X 20 Z 62.2
N1215 G00 X 18.6 Z 60.956
N1220 G00 X 17.2 Z 59.243
N1225 G01 X 17.2 Z 60.334
N1230 G01 X 18.6 Z 61.267
N1235 G00 X 17.2 Z 59.743
N1240 G00 X 15.8 Z 58.031
N1245 G01 X 15.8 Z 59.4
N1250 G01 X 17.2 Z 60.334
N1255 G00 X 15.8 Z 58.531
N1260 G00 X 14.4 Z 56.818
N1265 G01 X 14.4 Z 58.467
N1270 G01 X 15.8 Z 59.4
N1275 G00 X 14.4 Z 57.318
N1280 G00 X 13 Z 55.606
N1285 G01 X 13 Z 57.533
N1290 G01 X 14.4 Z 58.467
N1295 G00 X 13 Z 56.106
N1300 G00 X 11.6 Z 54.394
N1305 G01 X 11.6 Z 56.41
N1310 G03 X 13 Z 57.518 R 4.6
N1315 G00 X 11.6 Z 54.894
N1320 G00 X 10.8 Z 53.701
N1325 G01 X 10.8 Z 54.533
N1330 G03 X 11.6 Z 56.41 R 4.6
N1335 G00 X 34 Z 53.701
N1340 G91 G28 X 0 Z 0
N1345 G90
N1350 T1 M06
N1355 G42
N1360 G97 S1500
N1365 G00 X 34 Z 72
N1370 G00 X 34 Z 54.533
N1375 G00 X 21.9 Z 63.534
N1380 G01 X 21 Z 63.534
N1385 G01 X 10 Z 54.008
N1390 G01 X 10 Z 46.237
N1395 G02 X 11.282 Z 44.384 R 3
N1400 G01 X 21 Z 38.198
N1405 G00 X 34 Z 38.198
N1410 G91 G28 X 0 Z 0
N1415 G90
N1420 T4 M06
N1425 G41
N1430 G97 S2500
N1435 G00 X 34 Z 72
N1440 G00 X 34 Z 54.533
N1445 G00 X 10.9 Z 54.008
N1450 G01 X 10 Z 54.008
N1455 G01 X 10 Z 54.533
N1460 G03 X 12 Z 57.533 R 5
N1465 G01 X 21 Z 63.534
N1470 G00 X 34 Z 63.534
```

B.3. Third Sample Output

```
;!!!This code is generated by 'TurnDESIGN' for 'TURNDesign.part21'
N5 G291
N10 G54
N15 G21
N20 G99
N25 G91 G28 X 0 Z 0
N30 G90
N35 T1 M06
N40 G40
N45 G97 S1500
N50 M03
N55 G00 X 92 Z 236.375
N60 G01 X-1 Z 236.375 F .1
N65 G00 X 92 Z 236.375
N70 G00 X 92 Z 235.75
N75 G01 X-1 Z 235.75 F .1
N80 G00 X 92 Z 235.75
N85 G00 X 92 Z 235.125
N90 G01 X-1 Z 235.125 F .1
N95 G00 X 92 Z 235.125
N100 G91 G28 X 0 Z 0
N105 G90
N110 T2 M06
N115 G42
N120 G97 S2500
N125 G00 X 92 Z 234.726
N130 G01 X-1 Z 234.726 F .05
N135 G91 G28 X 0 Z 0
N140 G90
N145 T1 M06
N150 G40
N155 G97 S1500
N160 G00 X 88.7 Z 235.226
N165 G01 X 88.7 Z 2 F .1
N170 G01 X 90.5 Z 2
N175 G00 X 90.5 Z 235.226
N180 G00 X 87.4 Z 235.226
N185 G01 X 87.4 Z 2 F .1
N190 G01 X 89.2 Z 2
N195 G00 X 89.2 Z 235.226
N200 G00 X 86.1 Z 235.226
N205 G01 X 86.1 Z 2 F .1
N210 G01 X 87.9 Z 2
N215 G00 X 87.9 Z 235.226
N220 G00 X 84.8 Z 235.226
N225 G01 X 84.8 Z 2 F .1
N230 G01 X 86.6 Z 2
N235 G00 X 86.6 Z 235.226
N240 G00 X 83.4 Z 235.226
N245 G01 X 83.4 Z 95.4
N250 G01 X 85.3 Z 95.4
N255 G00 X 85.3 Z 235.226
N260 G00 X 82 Z 235.226
N265 G01 X 82 Z 95.4
N270 G01 X 83.9 Z 95.4
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N3035 G01 X 30.8 Z 122.056
N3040 G03 X 31.4 Z 128.96 R 79.6
N3045 G00 X 92 Z 122.063
N3050 G91 G28 X 0 Z 0
N3055 G90
N3060 T3 M06
N3065 G40
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N3080 G00 X 92 Z 85
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N3090 G00 X 84.125 Z 83.8
N3095 G01 X 83.762 Z 84.348
N3100 G01 X 84.448 Z 84.777
N3105 G00 X 83.761 Z 84.134
N3110 G00 X 83.435 Z 83.192
N3115 G01 X 82.95 Z 83.841
N3120 G01 X 83.636 Z 84.269
N3125 G00 X 83.069 Z 83.526
N3130 G00 X 82.745 Z 82.584
N3135 G01 X 82.14 Z 83.333
N3140 G01 X 82.826 Z 83.762
N3145 G00 X 82.379 Z 82.918
N3150 G00 X 82.053 Z 81.976
N3155 G01 X 81.328 Z 82.826
N3160 G01 X 82.014 Z 83.255
N3165 G00 X 81.689 Z 82.31
N3170 G00 X 81.363 Z 81.368
N3175 G01 X 80.516 Z 82.319
N3180 G01 X 81.202 Z 82.748
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N3190 G00 X 80.673 Z 80.76
N3195 G01 X 79.704 Z 81.812
N3200 G01 X 80.39 Z 82.241
N3205 G00 X 80.307 Z 81.094
N3210 G00 X 79.981 Z 80.152
N3215 G01 X 78.892 Z 81.305
N3220 G01 X 79.578 Z 81.733
N3225 G00 X 79.617 Z 80.486
N3230 G00 X 79.291 Z 79.544
N3235 G01 X 78.082 Z 80.797
N3240 G01 X 78.768 Z 81.226
N3245 G00 X 78.927 Z 79.878
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N3255 G01 X 77.27 Z 80.29
N3260 G01 X 77.956 Z 80.719
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N3270 G00 X 77.911 Z 78.328
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N3290 G00 X 77.219 Z 77.72
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N3455 G01 X 69.154 Z 75.218
N3460 G01 X 69.84 Z 75.647
N3465 G00 X 71.329 Z 73.19
N3470 G00 X 71.005 Z 72.248
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N3485 G00 X 70.639 Z 72.582
N3490 G00 X 70.312 Z 71.64
N3495 G01 X 67.532 Z 74.203
N3500 G01 X 68.218 Z 74.632
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N3640 G01 X 62.536 Z 71.082
N3645 G00 X 65.115 Z 67.717
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N3655 G01 X 61.038 Z 70.146
N3660 G01 X 61.724 Z 70.574
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N4230 G01 X 33.53 Z 234.726 F .05
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N4240 G01 X 60 Z 190
N4245 G02 X 70 Z 185 R 5
N4250 G01 X 80 Z 185
N4255 G01 X 84 Z 95
N4260 G01 X 84 Z 85
N4265 G01 X 60 Z 35
N4270 G01 X 60 Z 10
N4275 G01 X 60 Z 2
N4280 G01 X 92 Z 2
N4285 G00 X 92 Z 10
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N4295 G01 X 80 Z 185
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N4305 G01 X 30 Z 105
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N4345 G00 X 92 Z 35
N4350 G91 G28 X 0 Z 0
N4355 G90
N4360 T4 M06
N4365 G41
N4370 G97 S2500
N4375 G00 X 92 Z 238
N4380 G00 X 92 Z 185
N4385 G00 X 30.9 Z 122.056
N4390 G01 X 30 Z 122.056
N4395 G03 X 54.32 Z 164.456 R 80
N4400 G01 X 80 Z 185
N4405 G00 X 92 Z 185
N4410 G91 G28 X 0 Z 0
N4415 G90
N4420 T4 M06
N4425 G41
N4430 G97 S2500
N4435 G00 X 92 Z 238
N4440 G00 X 92 Z 85
N4445 G00 X 44.054 Z 49.038
N4450 G01 X 43.154 Z 49.038
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N4465 G00 X 92 Z 85
N4470 G91 G28 X 0 Z 0
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N4490 G97 S2500
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N4505 G00 X 92 Z 23
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N4515 G91 G28 X 0 Z 0
N4520 G90
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N4550 G00 X 0 Z 238
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N4810 G00 X 61 Z 6
N4815 G00 X 61 Z 7
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N4825 G01 X 61.263 Z 6.5
N4830 G00 X 61 Z 6.5
N4835 G00 X 61 Z 7.5
N4840 G01 X 63.263 Z 7.5
N4845 G01 X 62.263 Z 7
N4850 G00 X 61 Z 7
N4855 G00 X 60 Z 7
N4860 G01 X 50 Z 2
N4865 G00 X 61 Z 2
N4870 M01
N4875 G00 X 92 Z 2
N4880 G01 X -2 Z 2
N4885 G00 X 92 Z 2
N4890 M30
```

APPENDIX C

USER GUIDE FOR THE DEVELOPED SOFTWARE

C.1. Running the Software

Running TurnDESIGN is the same as any Windows based '.exe' application. Just double click on the 'TurnDESIGN.exe' after copying all the disk content in any location in the computer. The only point critical for a safe run is that; the TurnDESIGN folder and its contents must be in the same location with the 'TurnDESIGN.exe' file.

C.2. General View

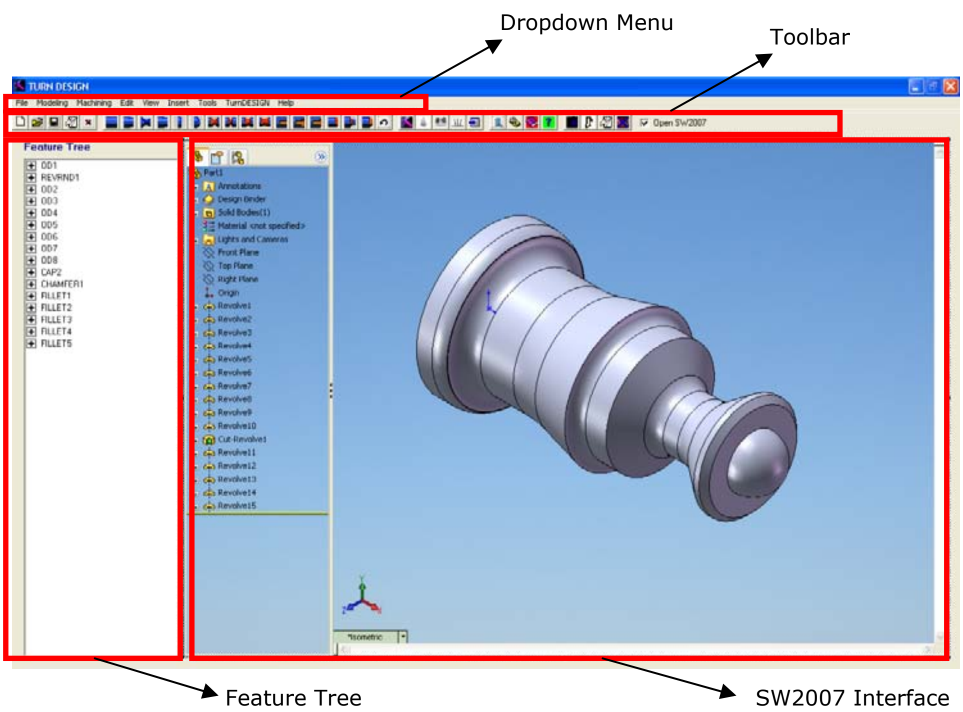


Figure C.1. General View for Modeling Module

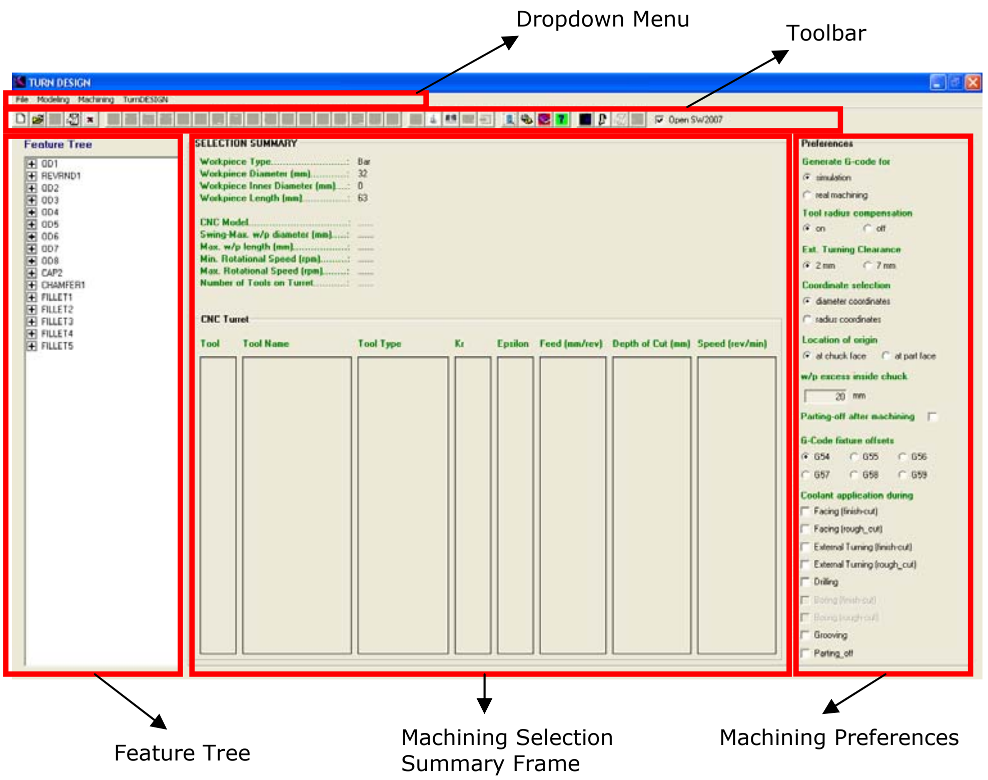


Figure C.2. General View for Machining Module

C.3. Menu Structure and Toolbar

The software is built with both "drop down" menus and a "toolbar". All menu commands also take place on the toolbar. Moreover, shortcut keys are assigned for some popular commands.

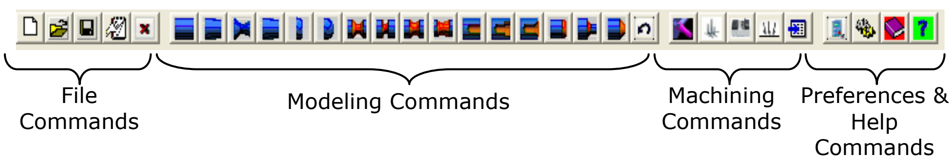


Figure C.3. Toolbar Structure

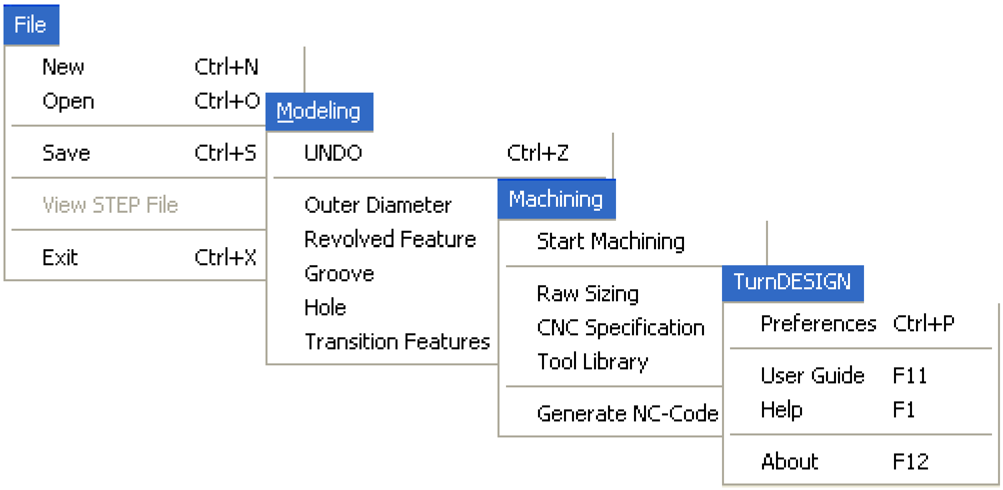


Figure C.4. Dropdown Menu Structure

CAUTION: With the SW2007 integrated to the software, additional dropdown menu commands specific for SW2007 also appears besides the "TurnDESIGN" specific menu commands. This enables extra options for the user, but the design edit using SW2007 commands will be discarded by "TurnDESIGN" during NC-code generation.

C.3.1. File Commands

 New File:

Opens a new design file and starts SW2007 automatically.

Menu Location: File → New

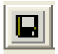
Shortcut Key: Ctrl+N

 Open File:

Opens a previously saved design file and starts SW2007 automatically.

Menu Location: File → Open

Shortcut Key: Ctrl+O

 Save File:

Saves a design file in STEP part21 format.

Menu Location: File → Save

Shortcut Key: Ctrl+S

 View STEP File:

Displays the content of saved design file in a separate window.

Menu Location: File → View STEP File

Shortcut Key: Not Applicable

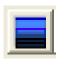
 Exit:

Closes the design file and exits the program.

Menu Location: File → Exit

Shortcut Key: Ctrl+X

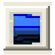
C.3.2. Modeling Commands

 Straight Outer Diameter:

Opens the "Straight Outer Diameter" creation dialog.

Menu Location: Modeling → Outer Diameter→ Straight Outer Diameter

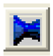
Shortcut Key: Not Applicable

 Tapered Outer Diameter:

Opens the "Tapered Outer Diameter" creation dialog.

Menu Location: Modeling → Outer Diameter→ Tapered Outer Diameter

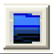
Shortcut Key: Not Applicable

 Outer Diameter to Shoulder:

Opens the "Outer Diameter to Shoulder" creation dialog.

Menu Location: Modeling → Outer Diameter → Outer Diameter to Shoulder

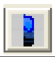
Shortcut Key: Not Applicable

 Revolved Flat:

Opens the "Revolved Flat" creation dialog. Revolved Flat is identical to Tapered Outer Diameter.

Menu Location: Modeling → Revolved Feature → Revolved Flat

Shortcut Key: Not Applicable

 Revolved Round:

Opens the "Revolved Round" creation dialog.

Menu Location: Modeling → Revolved Feature → Revolved Round

Shortcut Key: Not Applicable

 Spherical Cap:

Opens the “Spherical Cap” creation dialog.

Menu Location: Modeling → Revolved Feature → Spherical Cap

Shortcut Key: Not Applicable

 Square U Groove:

Opens the “Square U Groove” creation dialog.

Menu Location: Modeling → Groove → Square U Groove

Shortcut Key: Not Applicable

 Rounded U Groove:

Opens the “Rounded U Groove” creation dialog.

Menu Location: Modeling → Groove → Rounded U Groove

Shortcut Key: Not Applicable

 Partial Circular Groove:

Opens the “Partial Circular Groove” creation dialog.

Menu Location: Modeling → Groove → Partial Circular Groove

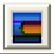
Shortcut Key: Not Applicable

 Vee Groove:

Opens the “Vee Groove” creation dialog.

Menu Location: Modeling → Groove → Vee Groove

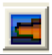
Shortcut Key: Not Applicable

 Round Hole:

Opens the "Round Hole" creation dialog.

Menu Location: Modeling → Hole → Round Hole

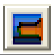
Shortcut Key: Not Applicable

 Counterbore Hole:

Opens the "Counterbore Hole" creation dialog.

Menu Location: Modeling → Hole → Counterbore Hole

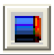
Shortcut Key: Not Applicable

 Countersunk Hole:

Opens the "Countersunk Hole" creation dialog.

Menu Location: Modeling → Hole → Countersunk Hole

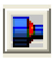
Shortcut Key: Not Applicable

 Round:

Opens the "Round & Fillet" creation dialog.

Menu Location: Modeling → Transition Feature → Round

Shortcut Key: Not Applicable

 Fillet:

Opens the "Round & Fillet" creation dialog.

Menu Location: Modeling → Transition Feature → Fillet

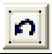
Shortcut Key: Not Applicable

 Chamfer:

Opens the “Chamfer” creation dialog.

Menu Location: Modeling → Transition Feature → Chamfer

Shortcut Key: Not Applicable

 UNDO:

Deletes the last design feature up to the first feature.

Menu Location: Modeling → UNDO

Shortcut Key: Ctrl+Z

C.3.3. Machining Commands

 Start Machining:

Closes SW2007 and starts machining module. As this command is clicked first calculations for the NC-code generation are held. So editing design is not possible anymore.

Menu Location: Modeling → Machining → Start Machining

Shortcut Key: Not Applicable

 Raw Sizing:

The raw size and shape is determined automatically as the machining starts and displayed on the “machining summary” frame. This command opens “Raw Sizing” dialog for manual input.

Menu Location: Modeling → Machining → Raw Sizing

Shortcut Key: Not Applicable

 CNC Specification:

Opens "CNC Specification" dialog for adding, editing and selecting CNC machine tool will be used for machining.

Menu Location: Modeling → Machining → CNC Specification

Shortcut Key: Not Applicable

 Tool Library:

Opens "Tool Library" dialog for adding, editing and selecting cutting tool set will be attached to the turret of selected CNC machine tool during machining.

Menu Location: Modeling → Machining → Tool Library

Shortcut Key: Not Applicable

 Generate NC-Code:

Generates NC-Code for the selected design, using the machining module settings and preferences.

Menu Location: Modeling → Machining → Generate NC-Code

Shortcut Key: Not Applicable

C.3.4. Preferences & Help Commands

 Preferences:

Opens "Preferences" dialog for preliminary settings of the program. It is not possible to change after modeling starts.

Menu Location: Modeling → TurnDESIGN → Preferences

Shortcut Key: Ctrl+P

 User Guide:

Opens HTML based "User Guide" dialog.

Menu Location: Modeling → TurnDESIGN → User Guide

Shortcut Key: F11

 Help:

Opens HTML based "User Guide" dialog.

Menu Location: Modeling → TurnDESIGN → Help

Shortcut Key: F1

C.4. Feature Creation Dialog

The elements of feature creation dialogs are almost same, thus only "Outer Diameter" dialog elements are discussed for representation.

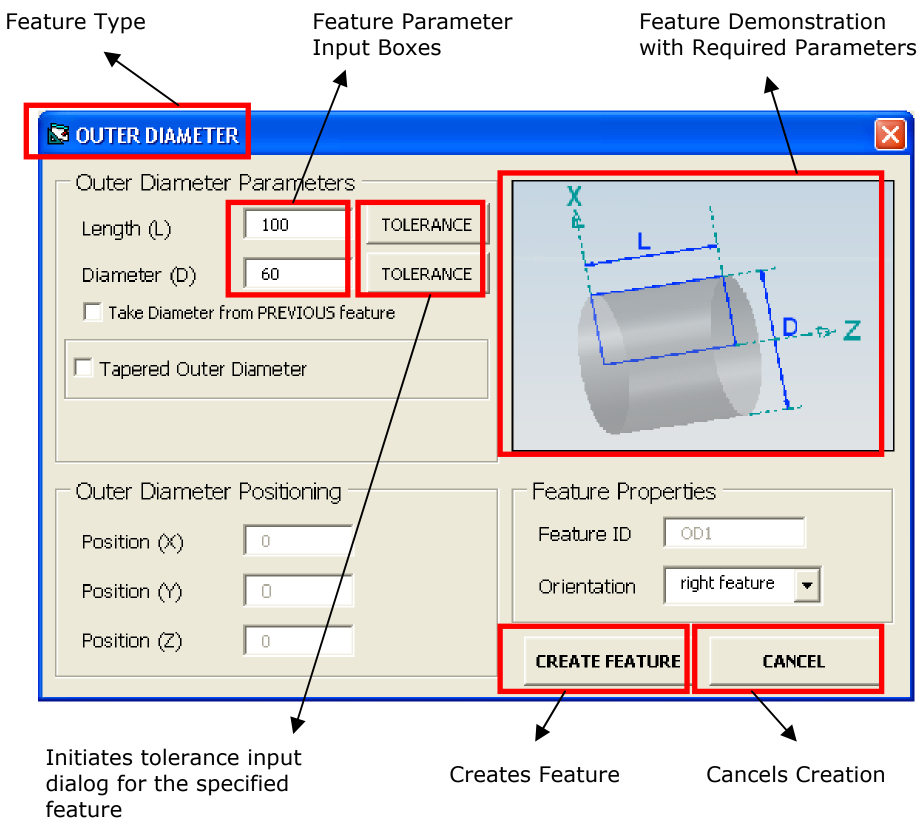


Figure C.5. Feature Creation Dialog

C.5. CNC Specifications Dialog

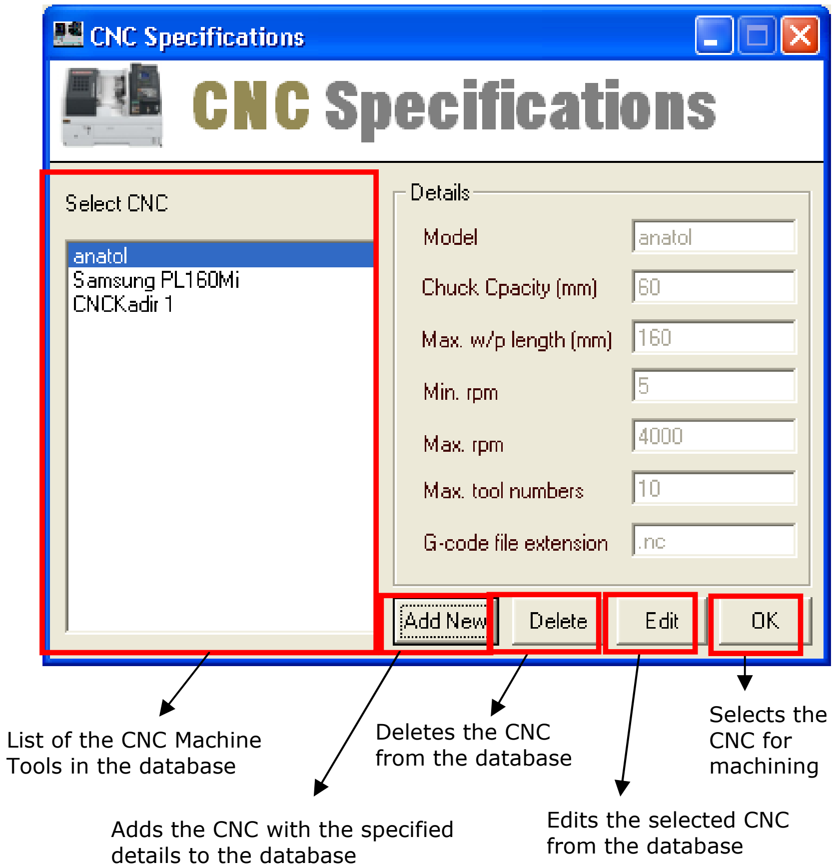


Figure C.6. CNC Specifications Dialog

C.6. Tool Library Dialog

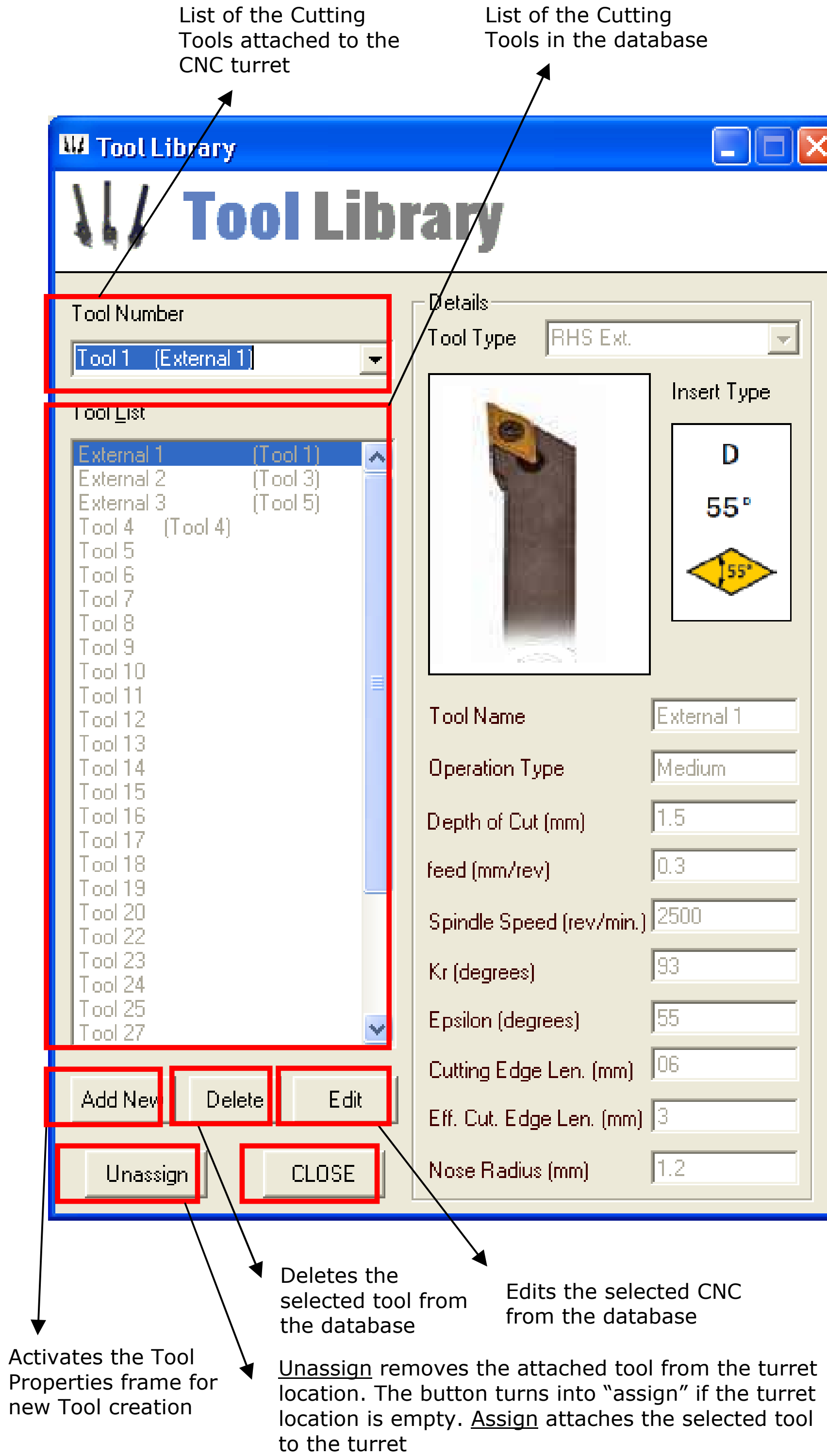
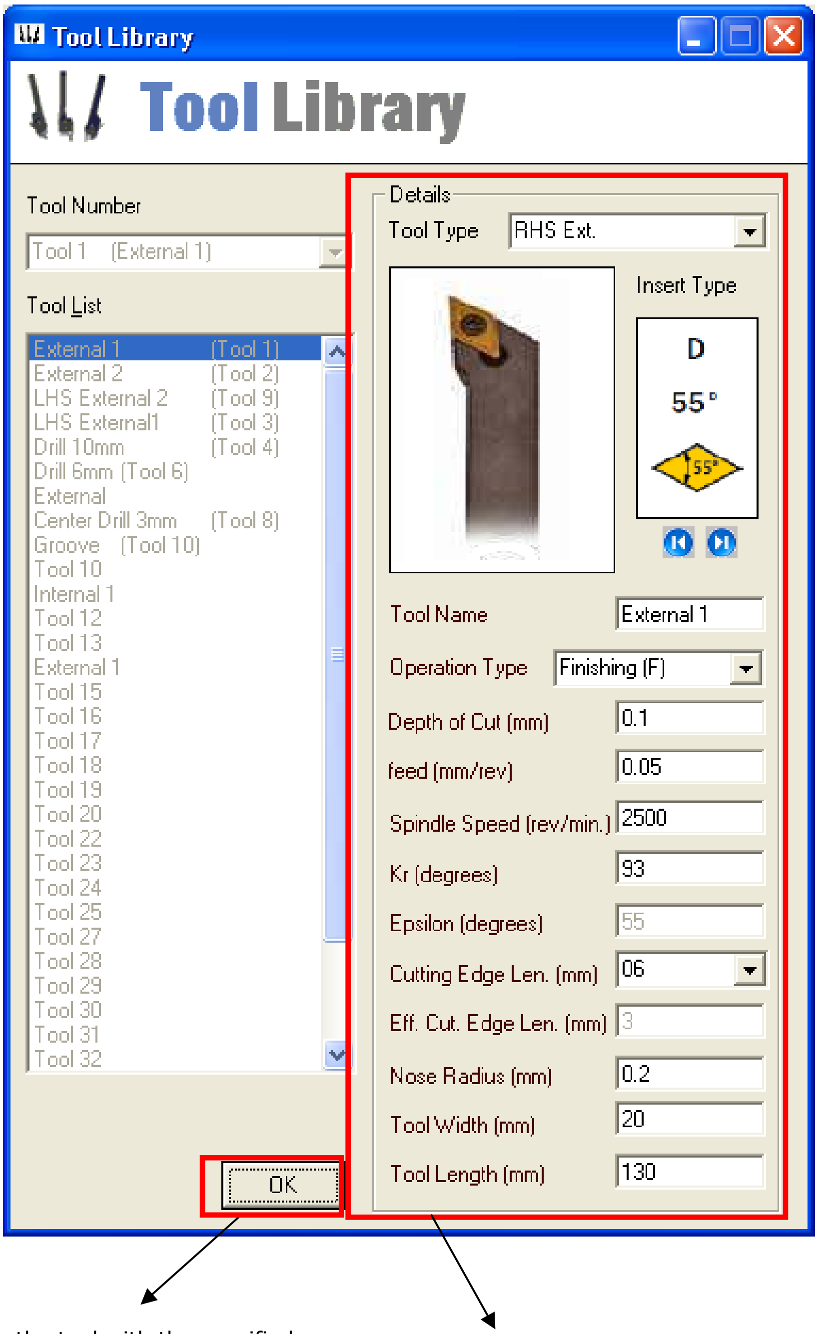


Figure C.7. Tool Library Dialog

The tool library dialog changes state by activating the inactive tool details frame when "add" or "edit" buttons is clicked.

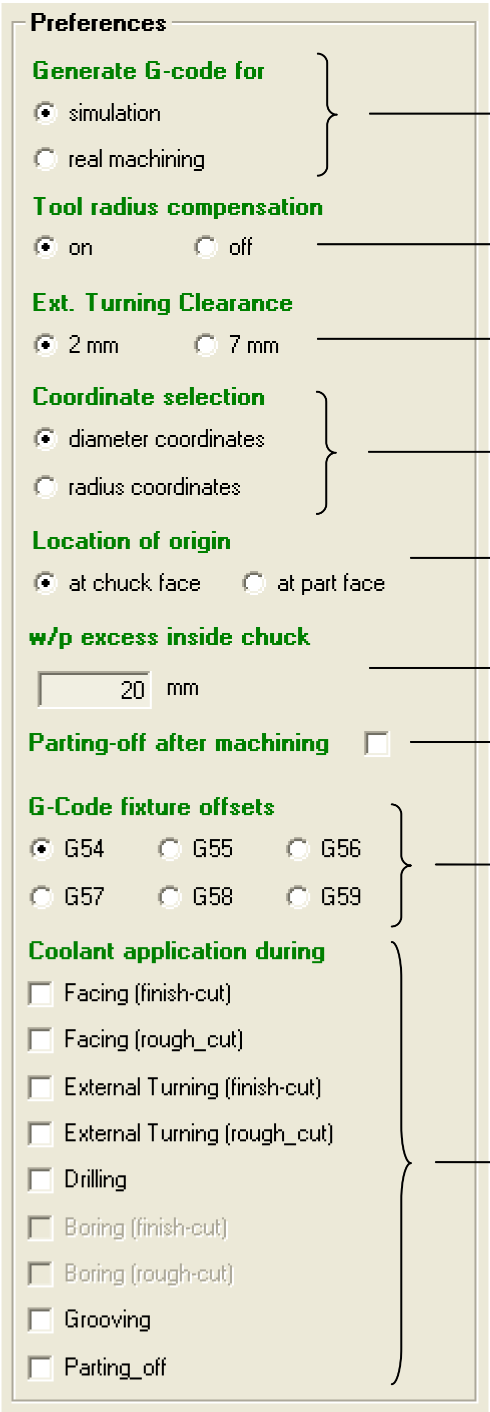


Adds the tool with the specified parameters.
Or Edits the tool according to the specified parameters.

The parameter selected frame for the tool will be added or edited

Figure C.8. Tool Library Dialog in Add or Edit Mode

C.7. Machining Preferences



Selection of NC-code generation purpose. This selection is important since crucialy "tool number assignments" inside the code changes.

Tool nose radius compensation is employed during finish cut stage if this option is on

Determines the min. Clearence value between the cutting tool "end relief angle" and "w/p" face inclination.

Selection of NC-code generation coordinate system in the "X" direction of a lathe

Selection of NC-code generation coordinate system in the "Z" direction of a lathe

Setting of the excess length that will be inside the chuck for w/p holding. It will be accounted for "real machining option"

Should be clicked to enable parting-off

Selection of CNC origin used during machining. Should be compatible with the CNC and machining prefereneces

Enables selection of cutting fluid usage according to the type of operation employed during machining

C.8. Error Messages and Error Handling

As stated in section 3.3.4, the error handling algorithms are used during software programming, against the possible user faults during data input stage. The user is warned with pop up messages, and informed about what to do. Since the software has interface with API, SW2007 and other software systems there are still unpredicted errors. In that case the user is advised to save the data, if possible, and restart the program by opening the saved data.

Here are the series of error messages are represented for familiarization.

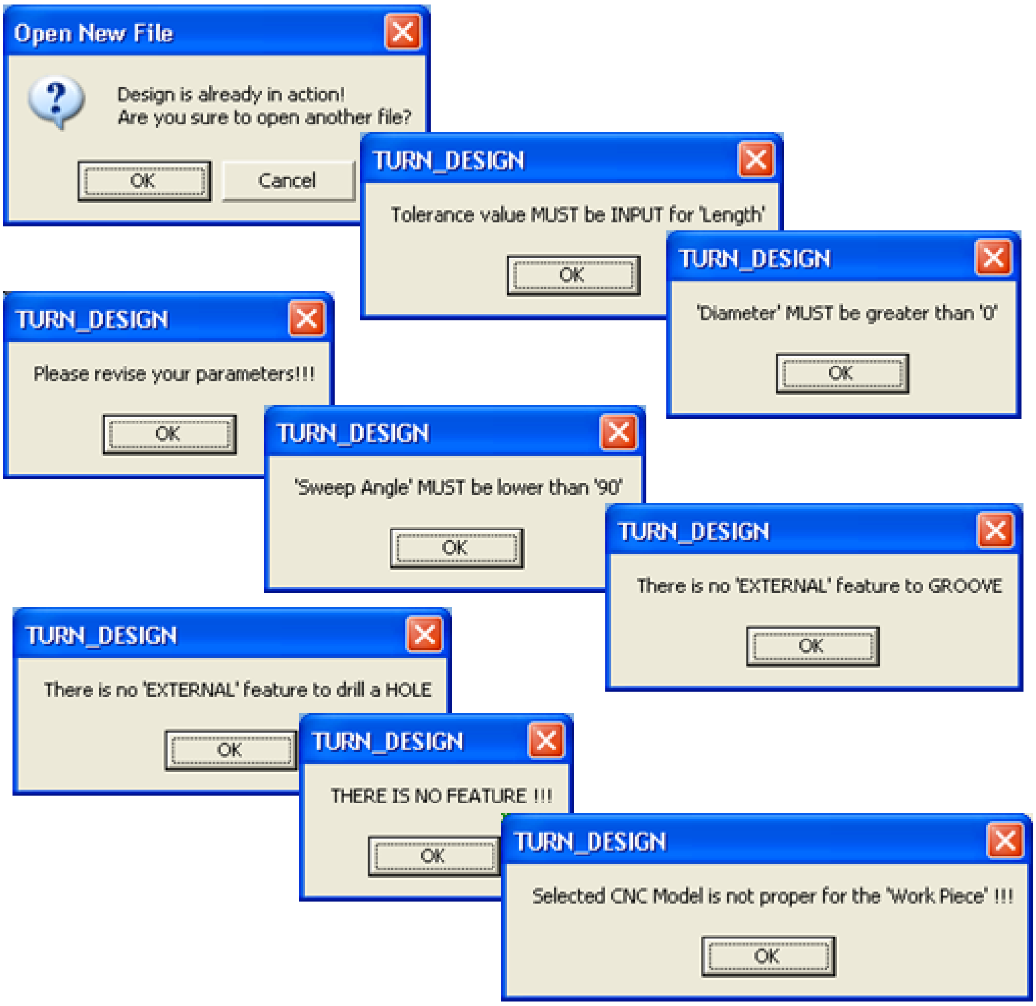


Figure C.9. Error Messages

APPENDIX D

FEATURE LIBRARY

Table D.1. Feature Library

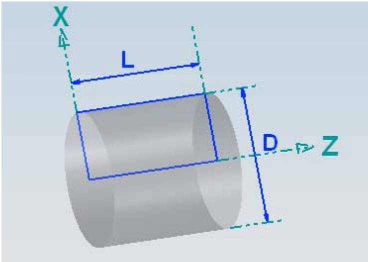
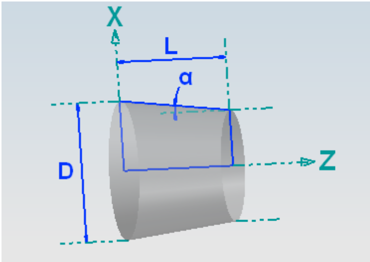
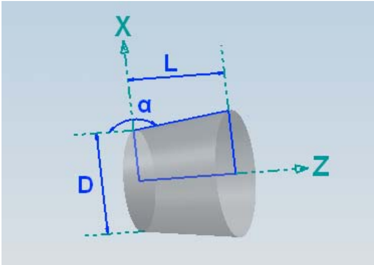
Feature	Parameters	Resulting Model
Outer Diameter (OD)	D = Diameter L = Feature Length	
Tapered Outer Diameter (ODT) 'Decreasing Diameter'	D = Diameter L = Feature Length ɑ = Taper Angle $*D_F$ = Final Diameter * Alternative for 'ɑ'	
Tapered Outer Diameter (ODT) 'Increasing Diameter'	D = Diameter L = Feature Length ɑ = Taper Angle $*D_F$ = Final Diameter * Alternative for 'ɑ'	

Table D.1. Feature Library (Continued)

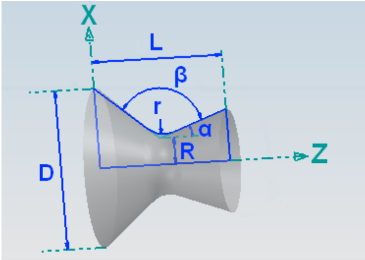
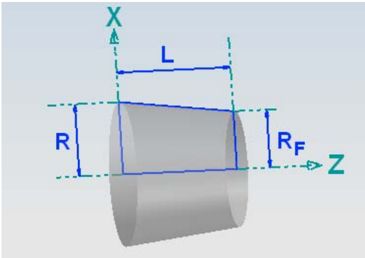
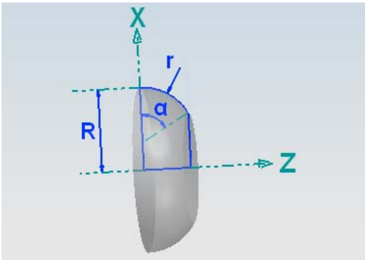
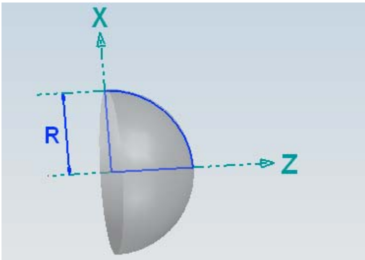
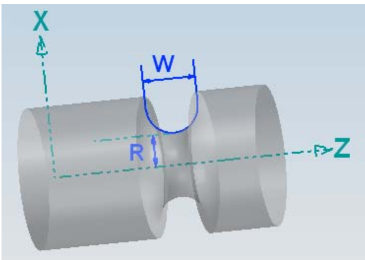
Feature	Parameters	Resulting Model
Outer Diameter To Shoulder (ODSHLD)	D = Diameter R = Radius at Placement L = Feature Length α = Tilt Angle β = Profile Angle r = Profile Radius *D_F = Final Diameter * Alternative for ‘L’	
Revolved Flat (REVFLT)	R = Radius at Placement L = Feature Length R_F = Final Diameter	
Revolved Round (REVRND)	R = Radius at Placement α = Sweep Angle r = Sweep Radius	
* Spherical Cap (CAP) * Feature generated using a special case for Revolved Round	R = Radius at Placement α = 90° r = R	
Rounded U Groove (RNDUGRV)	R = Radius at Placement W = Groove Width	

Table D.1. Feature Library (Continued)

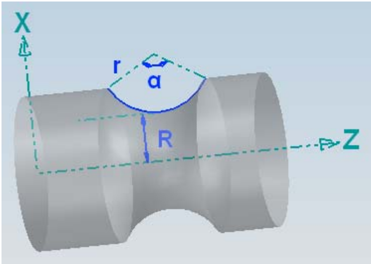
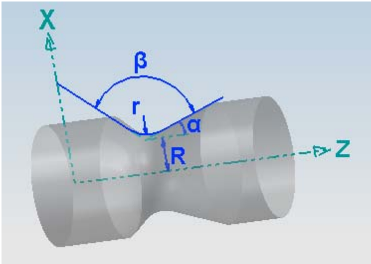
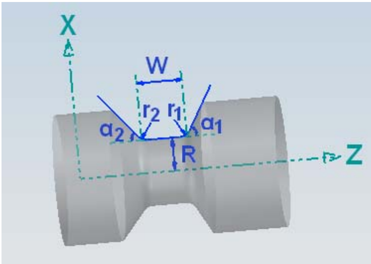
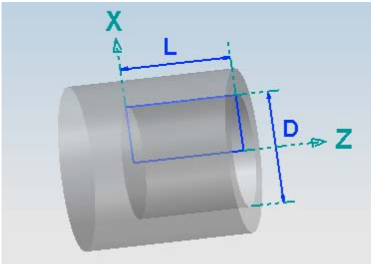
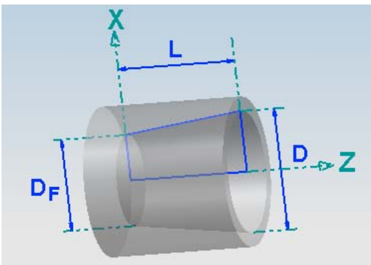
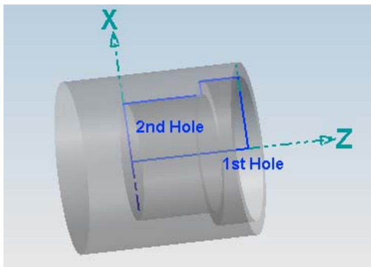
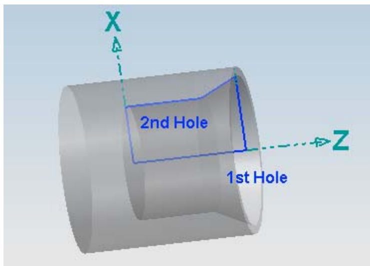
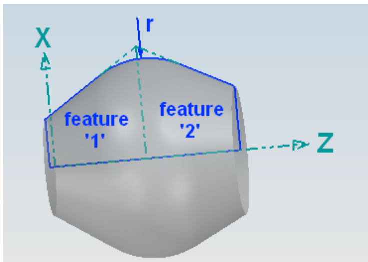
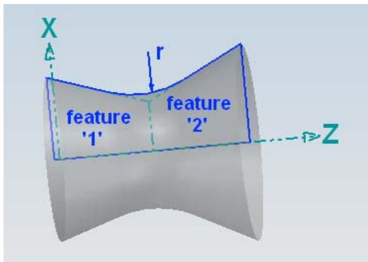
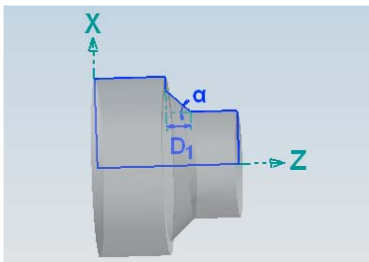
Feature	Parameters	Resulting Model
Partial Circular Groove (CIRCGRV)	R = Radius at Placement α = Sweep Angle r = Sweep Radius	
Vee Groove (VEEGRV)	R = Radius at Placement α = Tilt Angle β = Profile Angle r = Profile Radius	
Square U Groove (SQRUGRV)	R = Radius at Placement W = Groove Width $α_1$ = First Angle r_1 = First Radius $α_2$ = Second Angle r_2 = Second Radius	
Round Hole (RNDHOL)	D = Diameter L = Feature Length BC = Bottom Condition	
Tapered Round Hole (RNDHOL)	D = Diameter L = Feature Length α = Taper Angle BC = Bottom Condition *D_F = Final Diameter * Alternative for 'α'	

Table D.1. Feature Library (Continued)

Feature	**Parameters**	**Resulting Model**
Counterbore Hole (CBORHOL)	Compound feature: *1st Hole:* Tapered Round Hole with 'flat bottom' *2nd Hole:* Round Hole	
Countersunk Hole (CBORHOL)	Compound feature: *1st Hole:* Round Hole with 'flat bottom' *2nd Hole:* Round Hole	
Edge Round (ROUND)	1st Feature 2nd Feature r = Radius	
Fillet (FILLET)	1st Feature 2nd Feature r = Radius	
Chamfer (CHAMFER)	1st Feature 2nd Feature D_1 = First Offset α = Chamfer Angle * D_2 = Second Offset * Alternative for 'α'	

APPENDIX E

CUTTING TOOLS & INSERTS LIBRARY

E.1. Cutting Tools Library

Table E.1. Cutting Tools Library

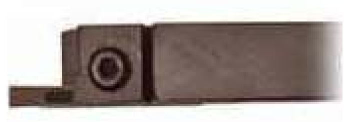
Cutting Tool	**Tool Definition**	**Related Parameters**	**Tool Figure**
RHS External Turning Tool	Used for external turning, profiling, face turning. Feed direction is in (-z)	L= Tool Length W= Tool Width κ_r= Side Cutting Edge Angle	
LHS External Turning Tool	Used for external turning, profiling, face turning. Feed direction is in (+z)	L= Tool Length W= Tool Width κ_r= Side Cutting Edge Angle	
Neutral External Turning Tool	Used for external turning, profiling, face turning. Feed direction is in both (-z) & (+z)	L= Tool Length W= Tool Width κ_r= Side Cutting Edge Angle	
Grooving	Used for grooving and parting-off	W_g=Grooving width W_d=Grooving epth	

Table E.1. Cutting Tools Library (Continued)

Cutting Tool	**Tool Definition**	**Related Parameters**	**Tool Figure**
Center Drill	Used to center a tool shank prior to drilling.		
Drilling Tool	Used for drilling and enlarging a hole. Feed direction is (-z)	L= Tool Length D= Tool Shank Diameter α= Tool point angle	
RHS Internal Turning Tool (RHS Boring Tool)	Used for internal turning, profiling, face turning. Feed direction is in (-z)	L= Tool Length W= Tool Width κ_r= Side Cutting Edge Angle	
LHS Internal Turning Tool (LHS Boring Tool)	Used for internal turning, profiling, face turning. Feed direction is in (+z)	L= Tool Length W= Tool Width κ_r= Side Cutting Edge Angle	

E.2. Inserts Library

Table E.2. Inserts Library

Insert Shape		Nose Angle	Eff. Cutting Length Coefficient	Insert Size	Cutting Depth (a_p),mm
Rhombic		80°	2/3	06	0 – 2.5
				09	0 – 3.5
				12	0 – 5
				16	1.5 – 7
				19	1.5 – 8
				25	2 – 9.5
Rhombic		55°	1/2	06	0 – 2.7
				11	0 – 3.8
				15	0 – 5
Square		90°	2/3	09	0 – 3.5
				12	0 – 5
				15	1.5 – 6.5
				19	1.5 – 8
				25	2.1 – [illegible]
				31	2.1 – 13
				38	2.1 - 16
Triangular		60°	1/2	05	0 – 1.4
				06	0 – 1.5
				09	0 – 1.8
				11	0 – 2.5
				16	0 - 4
				22	0 – 6
				27	1.5 – 7.9
				33	2 – 10
Rhombic		35°	1/4	11	0 – 2.5
				16	0 – 4
				22	0 – 5.1

Table E.2. Inserts Library (Continued)

Insert Shape		Nose Angle	Eff. Cutting Len. Coefficient	Insert Size	Cutting Depth (a_p),mm
Trigon		80°	1/4	04	0 – 2
				06	0 – 3.3
				08	0 – 4.1