A FEATURE BASED DESIGN SOFTWARE FOR PARTS TO BE MACHINED IN A FOUR-AXIS CNC LATHE

DEMİRALP HATİPOĞLU

SEPTEMBER 2005

A FEATURE BASED DESIGN SOFTWARE FOR PARTS TO BE MACHINED IN A FOUR-AXIS CNC LATHE

A THESIS SUBMITTED TO
THE GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
OF
MIDDLE EAST TECHNICAL UNIVERSITY

BY

DEMİRALP HATİPOĞLU

IN PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR
THE DEGREE OF MASTER OF SCIENCE
IN
MECHANICAL ENGINEERING

SEPTEMBER 2005

Approval of the Graduate School of Natural and Applied Sciences

Prof. Dr. Canan ÖZGEN
Director

I certify that this thesis satisfies all the requirements as a thesis for the degree of Master of Science.

Prof. Dr. Kemal İDER
Head of Department

This is to certify that we have read this thesis and that in our opinion it is fully adequate, in scope and quality, as a thesis for the degree of Master of Science.

Prof. Dr. M. A. Sahir ARIKAN
Supervisor

Examining Committee Members

Prof. Dr. Mustafa İlhan GÖKLER (METU, ME)

Prof. Dr. M. A. Sahir ARIKAN (METU, ME)

Prof. Dr. Metin AKKÖK (METU, ME)

Prof. Dr. Can ÇOĞUN (GAZİ UNV., ME)

Asst. Prof. Dr. İlhan KONUKSEVEN (METU, ME)

I hereby declare that all information in this document has been obtained and presented in accordance with academic rules and ethical conduct. I also declare that, as required by these rules and conduct, I have fully cited and referenced all material and results that are not original to this work.

Name, Last name : Demiralp HATİPOĞLU

Signature :

ABSTRACT

A FEATURE BASED DESIGN SOFTWARE FOR PARTS TO BE MACHINED IN A FOUR-AXIS CNC LATHE

Hatipoğlu, Demiralp
M.S., Department of Mechanical Engineering
Supervisor: Prof. Dr. M.A. Sahir Arıkan

September 2005, 141 pages

A previously developed feature library and algorithm are improved in order to develop a software capable of designing of parts to be manufactured using a four-axis CNC lathe. The developed program is composed of sections which are; part geometry design, process planning, cutting tool and machining parameter selection, part program generation and G-code simulation for verification.

The developed program is capable of designing parts which contain axisymetric features for turning and related operations, and non-axisymetric features for milling on facial, lateral and cylindrical surfaces. Implemented design procedure defines a blank material shape that can be circular or polygonal cross-sectioned bar or a pre-manufactured part.

A detailed database is created for proper cutting tool selection and machining data determination. User can either define or let the program to automatically select the cutting parameters like cutting speed, feed rate and depth of cut for each cutting

tool with respect to the workpiece material being machined. After completing design and process planning procedures; information for generation of the CNC program becomes readily available for editing or transferring to the CNC machine tool. User can also simulate the NC program to verify the tool path.

User friendly interface, which runs on Autodesk's INVENTOR software to visualize the design process, allows design and modifications to be done very easily moreover previously designed parts can be redesigned to produce new modified parts.

Keywords: Computer Aided Design, Computer Aided Manufacturing, Feature Based Design, Process Planning, Tool Path and G-Code Generation.

ÖZ

DÖRT EKSENLİ CNC TORNA TEZGAHINDA İŞLENEBİLECEK PARÇALAR İÇİN UNSUR TABANLI TASARIM PROGRAMI

Hatipoğlu, Demiralp
Yüksek Lisans, Makina Mühendisliği Bölümü
Tez Yöneticisi: Prof. Dr. M.A. Sahir Arıkan

Eylül 2005, 141 sayfa

Önceden oluşturulmuş olan unsur kütüphanesi ve algoritma geliştirilip, parça tasarımı ve dört eksenli CNC torna tezgahında imalat yapabilme özelliği taşıyan bir bilgisayar programı oluşturulmuştur. Oluşturulan program çeşitli bölümlerden meydana gelmiş olup, bunlar sırası ile; parça tasarımı, işlem planlaması, kesici takım ve işleme parametreleri seçimi, parça programı oluşturma ve simulasyondur.

Geliştirilmiş olan program; tornalama ve benzeri işlemler gibi simetrik olan parçaların yanısıra; alın, yüzey ve silindirik frezeleme gibi simetrik olmayan operasyonları içeren kompleks parçaların tasarımınıda yapabilecek niteliktedir. Geliştirilen dizayn prosedürünce tanımlanan ham parça geometrisi; dairesel, prizmatik kesitli veya ön operasyondan geçmiş olabilir.

Uygun kesici takım seçimi ve kesme parametrelerinin belirlenmesi amacı ile detaylı bir veritabanı oluşturulmuş olup, kullanıcı ister kendisi ister program tarafından

otomatik olarak kesme hızı, oranı ve derinliği gibi parametreleri kesici takım ve ham parça malzemesi yardımı ile kolayca belirleyebilir. Tasarım ve işlem planlaması sonucu imalat için gerekli parça programı oluşturulmuş olup, üzerinde değişiklik yapmak veya tezgaha aktarmak mümkündür. Bunun yanısıra kullanıcı simulasyonla takım yollarının doğruluğunu görebilir.

Arka planında dizayn edilen parçayı görüntülemek amacı ile Autodesk firmasınca üretilmiş olan INVENTOR dizayn paketini çalıştıran program; kullanıcı dostu arayüzü ile kolay bir şekilde parça oluşturabilir veya parça üzerinde değişiklik yapabilir. Ayrıca önceden dizayn edilen parçalar yeni tasarımlar için tekrar kullanılabilir.

Anahtar Kelimeler: Bilgisayar Destekli Tasarım, Bilgisayar Destekli Üretim, Unsur Tabanlı Tasarım, İşlem Planlaması, Kesici Takım Yolu ve G-Kodu Oluşturulması.

ACKNOWLEDGEMENTS

I would like to express my deepest gratitude to my supervisor Prof. Dr. M.A. Sahir ARIKAN for his guidance, advice, criticism, encouragements and insight throughout the research.

I would like to express my thanks to Mr. Tolga Z. SANDER for his suggestions, comments and help in understanding his thesis.

Many thanks are due to personnel of BAŞKENT university, Prof. Dr. İmdat KARA and Prof. Dr. Can COĞUN for their help and encouragement.

I also would like to thanks Inst. Dr. Yusuf USTA, Mr. Halit ŞAHİN and METU-BILTIR CAD/CAM center for their valuable information about cutting tools and manufacturers.

Finally, I am grateful to my family and my dear Banu for their support and help throughout my study.

TABLE OF CONTENTS

APPENDICES

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

In today's developing industry; designing a product are rarely integrated in one person due to the separation of labor. Therefore manufacturing products requires specialists from several domains which are experts in design and manufacturing area. In order to make some tasks easier in the production process, a large amount of different software tools has come into usage with the advance of computers. It is very important that the right kind of information taken from these computer tools realize products that comply with customer demands and can also be manufactured at reasonable cost.

1.1 Concurrent Engineering

Concurrent engineering is a product development system which tries to achieve:

- Shorter lead times.
- Higher quality.
- Lower cost.
- Consideration of the total life cycle of the product.

It is easy to achieve the goals stated above by a team approach combining with multi-disciplinary task force. The success of concurrent engineering depends on co-ordination; cooperation and communication within and between the teams and also consists of design engineers, manufacturing engineers together with marketing specialists. Figure 1.1 indicates the main objectives of concurrent engineering.

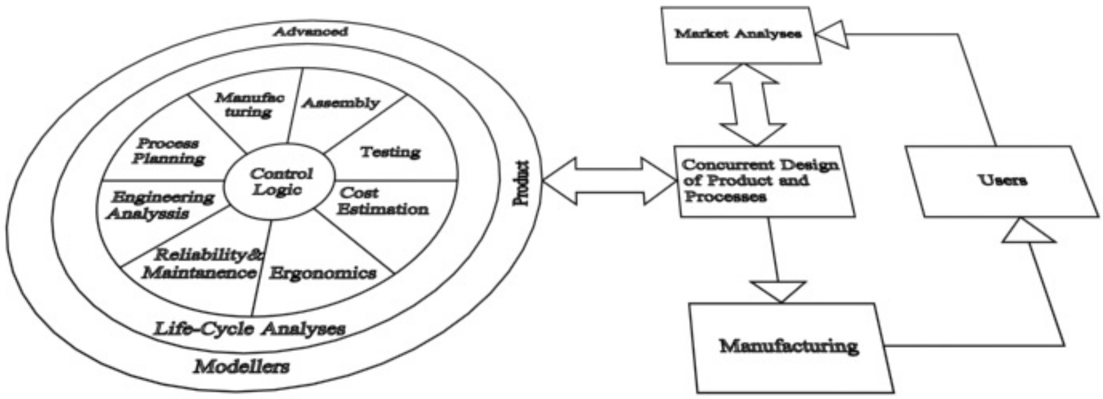


Figure 1.1: The concurrent engineering wheel.

Concurrent engineering has some aspects that are often different from the traditional way of engineering, sometimes referred to as "Over the Fence Engineering" or "Over the Wall Engineering" [21]. This kind of design is based on several steps. First the customer expectations are determined and design criteria are set. After the design is finished the detailed documentations and drawings are sent to the manufacturing department. The feasibility of production is examined by production engineers and additional investment to be able to put the part into production is considered if necessary. Cost analysis is performed after checking and modifying for the conflicting parts of the product design and manufacturing plan. Finally, the product is presented to the customer and modifications in the design are done according to the feedback received [23].

Design for manufacturing (DFM) is a method, which is usually used to implement some of the ideas that are parts of concurrent engineering. It is the practice of designing products with manufacturing in mind, so the product can be designed, assembled and tested in the least time with minimum development cost and have the desired levels of quality and reliability to satisfy customers in order to compete well in the marketplace.

The use of computer aided design packages allows geometric designs to be completed in much shorter time with better design quality since the designer is able to visualize the part in 3D before manufacturing. A lot of research has been done to integrate design and manufacturing by considering restrictions during design [11]. Feature based design approach is used to assure manufacturability of the computer aided designed parts by preparing the graphical feature libraries which are formed by using the capabilities of the available machine tools. The construction of the features based on machine capabilities and created by using boolean operations such as union and/or subtraction [7].

The main aim is to design parts that can pass from the stages of process planning and NC programming in order to be manufactured by using CNC machines. Programming CNC machines manually can easily be applied for simple parts but when the parts require complex operations, manual programming of these machines becomes inefficient, potentially inaccurate and time consuming.

Special programs or softwares have been developed to generate the digital numerical data and to prepare NC programs by using suitable post-processors for different machines. In the production process; these programs defines the workpiece by using primitive geometries which are mathematically defined and called as features. By selecting the proportional tools and manufacturing process plan; the tool paths and NC program are generated. In addition to using such programs, manufacturing knowledge is needed to specify parameters such as feed rate, depth of cut and cutting speed.

The developed program in this thesis is based on the primitive feature library and algorithm that Mr. Tolga Z. Sander was developed for his masters thesis [23]. The feature library is improved to work on a feature based CAD package and to produce most of the parts that can be machined by using a four-axis CNC turning center. The library contains features for internal and external turning and features for milling operations on planar, cylindrical and facial surfaces of the workpiece. Knowing that a four-axis CNC lathe exists in the Mechanical Engineering department of the Middle East Technical University, features formed by using two or three-axis lathes can also be manufactured by switching the non-machinable features off.

Most of the modification done on the design and visualization by using Autodesk's INVENTOR software rather than AutoCAD. The main reason to use a feature based CAD program is to develop a design procedure that uses features within the CAD program to create some complex geometries that can not be constructed properly using the AutoCAD software. Modifications on part program generation contains algorithms for each process involving turning and milling related operations. In addition to that, process planning and the cutting tool selection modules are improved to use the exact tools and proper machining parameters stated in the catalogs of cutting tool manufacturers. Simulation of the tool paths are also created to enable the visualization of machining process and approve the reliability of the generated NC code by tool path verification.

A literature survey about the subject with previous studies are given in Chapter 2. The detailed explanation of the developed program is given in Chapter 3 and Chapter 4 contains case studies to visualize the work flow of entire program. Finally, conclusion and recommendations for future work are presented in Chapter 5 following with an Appendix containing user manual, program structure and basics of NC programming.

CHAPTER 2

LITERATURE SURVEY

2.1 CNC Machines and Four-Axis Technology

Computer numerically controlled machines has undergone a great deal of change as a result of rapid increase in computer development. Development of new and improved materials helps production of advanced tool geometries which can perform cutting operations at speeds and feeds that were formerly impossible to attain.

Today's manufacturing industries rise their production lines with the support of modern mill-turn centers. Machining complex shaped workpieces which requires both turning and milling operations is the primary advantage of these machines. In addition to their manufacturing flexibility and multi machining capability, modern mill-turn centers provides fast indexing with improved four-axis spindle and NC turret technology. Higher metal removal rates, increased machine utilization and fast setup timing are the main advantages of these machines over conventional two-axis lathes.

The developed software is especially programmed to machine workpiece by using four-axis CNC lathes which are also called as turning cells or turning centers. The machine axes are; X, Y, Z and C. Z-axis is along the spindle rotation and increases positively away from the spindle. X-axis is the radial direction of the workpiece and increases positively away from the origin to the workpiece. Y-axis is calculated with the right handed rectangular coordinate system formed by X and Z-axis. Finally C-axis is the rotation axis of the spindle which is counterclockwise about Z-axis. (Figure 2.1)

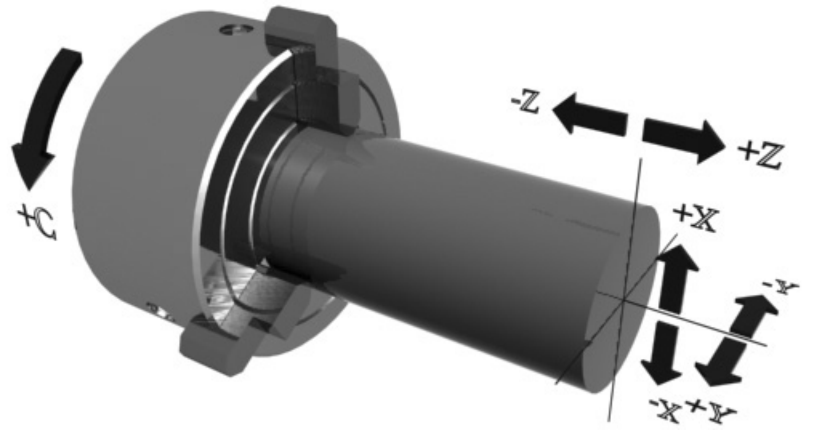


Figure 2.1: Axes of a turning center.

As in other conventional CNC lathes, X and Z-axis controls the motion of the turret head along radial and logitudinal directions. On the other hand Y-axis control is used for the movement of turret head along transverse direction in order to machine off-center holes, cavities and pockets on faces of workpiece.

Generally CNC lathes uses a reference point which is the intersection of the X and Z-axis. Therefore the value of X coordinate is zero along spindle centerline and the value of Z coordinate is zero along the facial plane of tailstock side of workpiece.

A numerical control (NC) program is a list of instructions that completely describes, in sequence, every operation to be carried out by a machine. When a program runs, each instruction is interpreted by a machine controller, which causes an action such as starting or stopping of a spindle or coolant, changing of spindle speed or rotation or moving the turret head a specified direction, distance or speed.

The form that program instructions can take and how programs are stored and loaded into the machine depends on the individual machine control system. However program instructions must be in a form that the machine controller can understand. Part programs are prepared as a list of these instructions and generally transferred to machine tool computer via RS-232 serial link.

2.2 Previous Studies

Chow J. G. and Sakal L.R. developed a CAD/CAM software to enhance design productivity and to assist the NC code generation facets of the machining process. Using computer-aided process planing (CAPP) techniques entire automated process planning is integrated in CAD/CAM applications and the workout outlines the design and development of an intelligent CAPP system by integrating two commercial CAD and CAM software packages which are AutoCAD and Mastercam [10].

Su C.J., Sun T.L., Wu C.N. and Mayer R.J. developed an efficient form-feature-based modelling system, and addressed the important issue of utilizing feature information for manufacturing by presenting an Euler operator-based approach for efficient and effective form-feature encoding with manipulation in a feature-based design environment. By developing feature interference resolving and machinability testing techniques and integrating with an efficient feature-based design system, this research enables the future development of an integrated feature-based design and manufacturing system [27].

Stainslaw Z. developed a specialized built in software for three axis lathes enabling integration of CAD/CAM systems same as in CNC Multi axis milling. For accurate part programming on three axis lathes a new interpolation is applied both, using both spiral segments of archimedes and circular segments rather than linear segments as applied in CNC milling. The software also includes modified post processors for three axis lathes [26].

Zezhong C. C., Zuomin D. and Geoffrey W. V. developed a new tool path generation method to automatically subdivide a complex sculptured surface into a number of easy-to-machine surface patches; identify the favorable machining setup and orientation and generate effective three-axis CNC tool paths. The developed work contributes to automated multiple-axis CNC tool path generation for sculptured part machining and forms a foundation for further research [29].

Elkeran A. and El-baz M. A. developed an approach for the generation of NC machining instruction for complicated surfaces directly from a solid modelling. The work includes extracting data for the machined part using STL format, and constructs the topological surface for the various cutting planes. The developed work also generates the tool path required for machining the part for both rough and fine operation [12].

2.2.1 Previous Studies in M.E.T.U.

Saygın C. developed an interpreted rule based process planning system to determine the sequence of machining operations for rotational components. The system inference engine is designed by using Prolog programming language and obtained the main part geometry from AutoCAD software to generate a process plan considering rough and finish cuts including the sequence of machining operations for external, internal and superimposed features as well as corresponding cutting tools and machining data [24].

İlik Y. developed a desktop CAD/CAM package for rotational parts in order to generate the NC codes necessary to machine the part from blank material to final shape. A script file is generated by the developed program to handle the design operations using AutoCAD software and with the help of the embedded post processor the generated NC codes can be converted to any NC code that can be used in any CNC machine [14].

Özturk F. developed a machinability and tool database for a previously developed desktop CAD/CAM package including automatic tool selection capability for any rotational workpiece by considering the tool collusion possibility. By analyzing the machining profile against tool collision, the system selects and recommends cutting tools that are suitable to machine the defined rotational part geometry [19].

Boz O. developed and implemented a CAD/CAM program for milling machines to provide users the opportunity to interact with the CAD system in a design by features environment for prismatic parts. After designing process by entering the manufacturing parameters and selecting the necessary cutting tools; the program can capable of producing the NC codes automatically which are necessary to machine the part from blank material to the final shape [8].

Sander T. Z. developed a feature library, an algorithm and an interactive computer program for design of parts to be manufactured using four-axis CNC lathe with milling capabilities in axial and radial directions. AutoCAD software is used for design process. The developed software in this thesis is based on the works that Sander was developed for his theses [23].

Susuzlu T. developed a verification module for a previously developed CAD/CAM package for rotational parts. The developed module utilizes AutoCAD program to perform NC verification and also capable of recognizing any standard tool, determining collisions, computing material removal rate and understanding tool nose radius compensation codes. Since the designed part is also modelled after the cutting tool simulation, machined part is compared with the designed part to identify the excess material and gouges [28].

Salihler S. developed a simulation and verification software for parts to be machined in turning centers. By entering the NC code, workpiece geometry and selecting proper tool and tool holders with the previously calculated cutting parameters; all of the machine motions with possible errors can be seen during simulation [22].

2.3 CAM Packages Used for Today's Industry

2.3.1 Unigraphics NX

NX is the preferred NC machining solution for thousands of companies worldwide. Tool path and machine simulation and verification in NX help manufacturing engineers quickly improve NC program quality and machine efficiency.

NX serves virtually any machine tool programming needs, providing comprehensive tools for everyday machining activities such as two and three-axis milling, turning, wire EDM, and drilling. In addition NX has a leading set of capabilities in more advanced areas from high speed machining, four and five-axis complex milling to programming of the latest combination mill-turn machining centers.

Turning in NX can use either 2D profiles or full solid models. It contains routines for roughing, multiple pass finishing, grooving, thread cutting and centerline drilling. Users control parameters such as feed rate, spindle speed and part clearance. NX is very flexible and allows programming in XY or ZX environments for horizontal, vertical, or inverted vertical orientations. Moreover NX has a reputation for driving the latest generation turning centers with an advantage of these completing jobs with fewer setups and considerable time and cost savings.

NX delivers comprehensive programming capabilities for milling operations. The fixed-axis milling module offers complete and comprehensive functions to produce three-axis motion tool paths. Optimization techniques in operations such as planar milling help reduce the time to cut parts with a large number of pockets.

For fixed axis and variable axis conditions; the NX surface contouring modules provides complex surface and contour milling together with a number of cut methods and patterns. Moreover variable-axis milling provides multiple drive methods with a wide range of tool axis control options, such as normal and relative to part or drive surfaces. More information can be achieved by visiting the web site stated in [6].

2.3.2 EdgeCAM

EdgeCAM is a simple to use, native windows based CAM system specifically designed for the busy production machining environment. EgdeCAM generates machining strategies and NC codes for solid models. Capable of importing various part files including INVENTOR, PRO/ENGINEER, SOLID WORKS, SOLID EDGE and CATIA; tool paths generated in EgdeCAM remain associative to the originating CAD model throughout the manufacturing process.

In addition to these features; EdgeCAM develops machining strategies that optimize tool paths, eliminate unnecessary air cutting, maximize tool life, reduce programming time and increase overall productivity.

EdgeCAM enables producing NC codes for a wide range of machine tools, including two-axis lathes, multi-turret configurations, sub-spindle turning centers and mill-turn machines. EdgeCAM produces advanced rough and finish turning cycles, together with the support for facing, boring, drilling and threading in either canned cycle or longhand format. Moreover the complete range of milling and hole machining cycles can be implemented on mill-turn machines. More information can be achieved by visiting the web site stated in [2].

2.3.3 Pro/Engineer Complete Machining Module

Pro/E Complete Machining offers a full solution for creation of all types of programs for CNC machines, from the simple to the very complex including multi-axis machining by using the entire capabilities of NC programming and tool libraries. 2 1/2- to 5-axis milling, multi-axis turning together with the mill/turn and 4-axis wire EDM capabilities improves product quality and manufacturing consistency by generating tool path directly on 3D designs. As part of an integral CAD/CAM solution, no data translation is required and the necessary time to machine the product into market is reduced with associative tool path updates in design changes. [5].

2.3.4 Mastercam X

Mastercam X delivers fully associative geometry and tool paths enabling modification of the geometry, tooling information or machining parameters and immediately get an accurate, updated tool path.

Some of the advanced features are; intelligent roughing of inside and outside diameters, including roughing to a boundary for casting operations. Finish contouring or profiling and setting a constant surface speed feed rate for the desired finish quality. Other benefits are; boring, drilling and point-to-point machining. Grooving at any angle with multiple depth of cuts and complete threading with multiple starts including calculation of the diameter by using the thread tables.

On the other hand fully operational C-axis milling can be applied with the addition of milling package. The milling feature enables face and cross contour and drill including the ability to sort points in a clockwise or counterclockwise sweep direction and machining C-axis contours with three-dimensional or flat geometry. More information can be achieved by visiting the web site stated in [4].

2.3.5 CATIA NC-Manufacturing

CATIA NC Manufacturing module offers basic NC capabilities such as tool path verification, material removal simulation, remaining material analysis, tool path edition and creation of shop floor documentation. Furthermore, different manufacturing data such as APT files, CL files and ISO code can be imported and reviewed.

In lathe machining mode, user can easily define two-axis turning and drilling operations using quick tool path definition algorithms, for both horizontal and vertical machines. In addition to that tools can easily be created and stored in tool catalogs. Entire manufacturing process is covered from tool path definition and computation, tool path verification including material removal simulation to NC code and shop floor documentation generation.

The advanced machining mode covers machining of complex 3D parts for aerospace, hydraulics, and turbo machinery and offers machining operations covering milling from 2.5-axis mode up to 5-axis mode as well as axial machining. In addition to the complete set of machining operations; multi-axis flank contouring, 5-Axis Helix Machining for Turbo-machinery parts, and advanced multi-axis processes involving multi-pocket strategies dedicated to Structural Part Machining. More information about prismatic, surface and pocket machining together with manufacturing verification properties can be achieved by visiting the web site stated in [1].

CHAPTER 3

THE DEVELOPED PROGRAM

3.1 Overview of the Developed Software

The developed software called "InventorNC" is composed of designing, process planning and code generation modules. Created especially for four-axis CNC turning centers by using the source codes of a previous study [23]. Parts which can be machined by using these machines are quite different than the ones that can be machined by conventional lathes or milling machines. Four-axis turning centers can machine facial, lateral and circumferential surfaces of a workpiece with one set-up and clamping which contains both turning and milling operations.

The developed program was built on Autodesk‘s INVENTOR software for visualization of the designed part in three dimension. In modelling axisymetric parts; two-dimensional views can be sufficient for the definition of part profile. The part geometry is created by extruding the two dimensional profile by rotating about the central axis which also called as revolve operation in most computer aided design programs. On the other hand modelling unaxisymetric parts requires three dimensional viewing so as to machine by using a milling machine or milling features of a four-axis turning center.

The previously developed modelling method uses feature based design approach to guarantee the manufacturability of designs on four-axis turning centers. The library was constructed after a survey on available feature based databases like the one given by Peklenik and Sekolonik [20] and coding systems which have been mainly developed for group technology applications like Opitz and KK-3 [9].

The reconstructed graphical feature library comprises the features related to machining operations. These are basically features for constructing inner and outer blank geometry definition, features for constructing inner and outer part geometry definition for turning operations and features for constructing part geometry definition for milling operations to remove material from facial, lateral and cylindrical surfaces of the workpiece. To determine the machinability of the part for the selected CNC machine tool is very important. The redeveloped feature library contains manufacturable features that can be machined by using four-axis turning centers therefore the parts which can not be manufactured on these machines can also not be designed by using the developed software.

3.2 Blank Material Geometry

The geometry of part which is going to be machined by using design specifications is called blank material. It can be a raw stock bar, a cast material, a forged material or a pre-machined product. Blank material geometry must be defined previously in the program to generate the necessary NC codes. This can be done using blank definition features. The available features for defining the outer and inner profile of the blank part are : Cylinder, Taper, Prism, Concave and Convex.(Figure 3.1)

User can easily create blank material geometry by selecting and placing the required features starting from the spindle side of the lathe to the tailstock side. Therefore every new feature is placed to the right of the previous feature and Boolean addition is performed with previously placed features automatically by the program to form the blank outer profile geometry.(Figure 3.2) On the other side blank inner profile geometry must be defined after a previously defined outer profile geometry. Same features in Figure 3.1 can be used to define the inner profile geometry. However, features are placed to the left of the previous feature starting from the tailstock side and Boolean subtraction is performed from the outer profile geometry.(Figure 3.3)

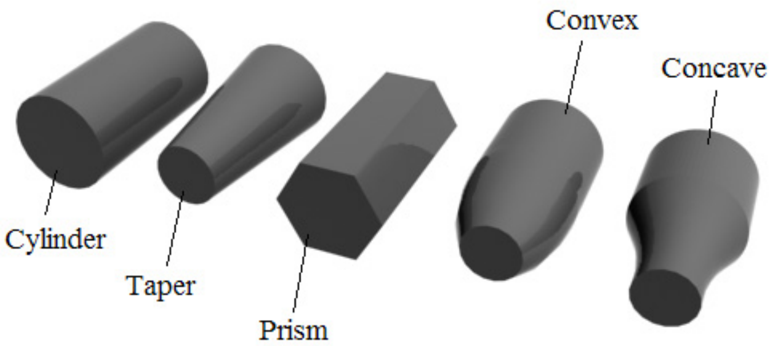


Figure 3.1: Features used to define blank geometry.

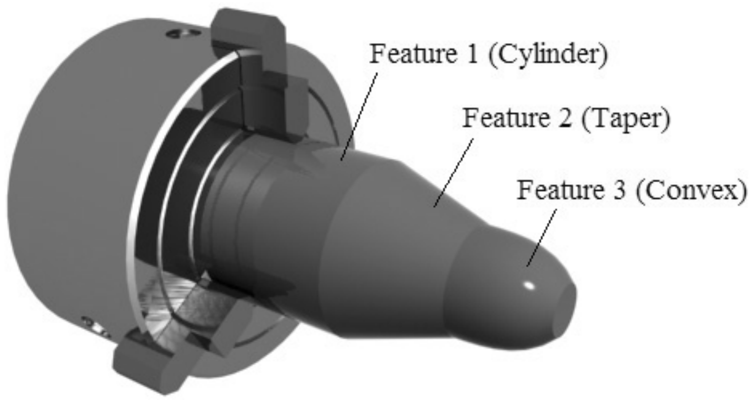


Figure 3.2: Blank outer profile creation.

The program automatically places new features to the right of the last added feature for outer profile geometry definition and to the left of the last subtracted feature for the inner profile geometry definition. Therefore, adding a new feature does not need an information about the placement except other parameters.

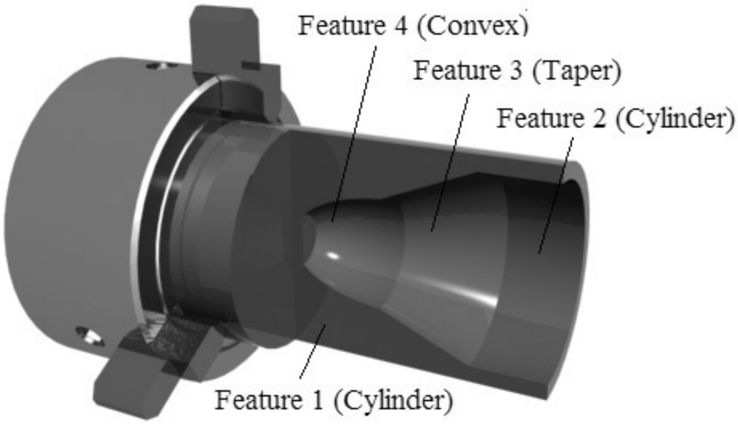


Figure 3.3: Blank inner profile creation.

3.2.1 Cylinder Feature

Length : Length of cylinder

Diameter : Diameter of cylinder

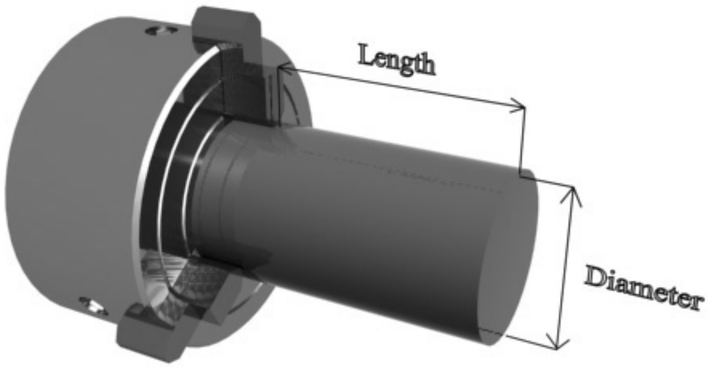


Figure 3.4: Cylinder feature parameters.

Cylinder feature is especially used for defining straight surfaces for inner and outer profile geometry.(Figure 3.4) Holes can easily be created by using cylinder feature in inner profile geometry definition. But a hole feature must be used for advanced operations like counterboring, countersinking or threading. In addition to that; diameter of simultaneous cylinder features must decrease from left to right for outer profile and right to left for inner profile geometry definition.(Figure 3.5) Because large diameter differences create grooves which can not be manufactured by using a cylinder feature instead a groove feature must be used.(Figure 3.6)

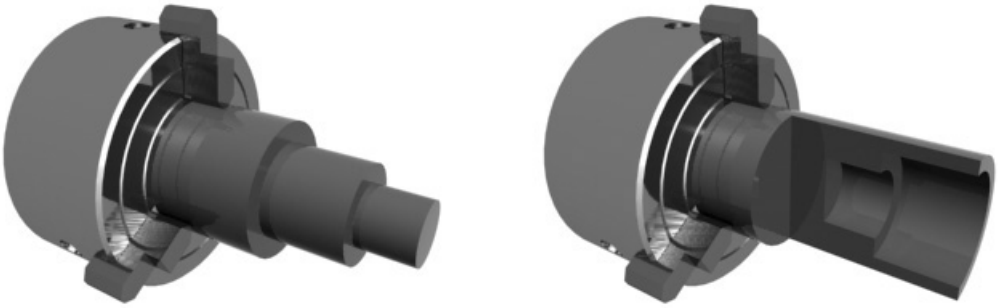


Figure 3.5: Simultaneous cylinder definitions for inner and outer profile.

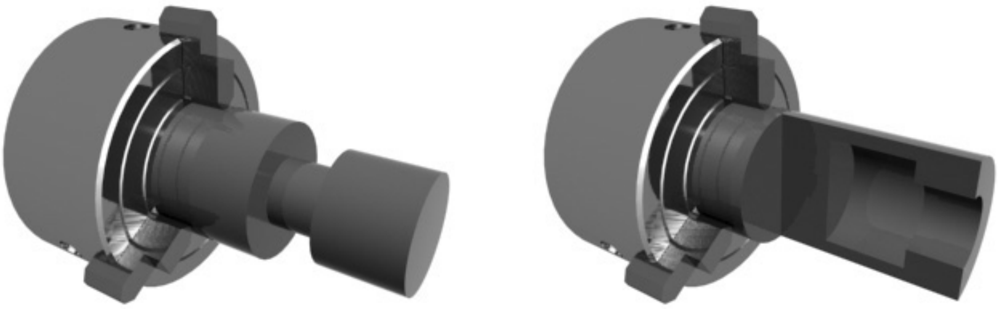


Figure 3.6: Wrong cylinder definitions creating undefined grooves.

3.2.2 Taper Feature

Length : Length of taper

Diameter1 : Spindle side diameter of taper

Diameter2 : Tailstock side diameter of taper

Angle : Taper Angle

Taper feature is used for inclined and declined surfaces. The main parameters are given in Figure 3.7. In order to create a taper feature; user must define a length value following by two diameters or one diameter and a taper angle. Diameter 1 is bigger than diameter 2 for negative values of taper angle and diameter 1 is smaller than diameter 2 for the positive values of taper angle.

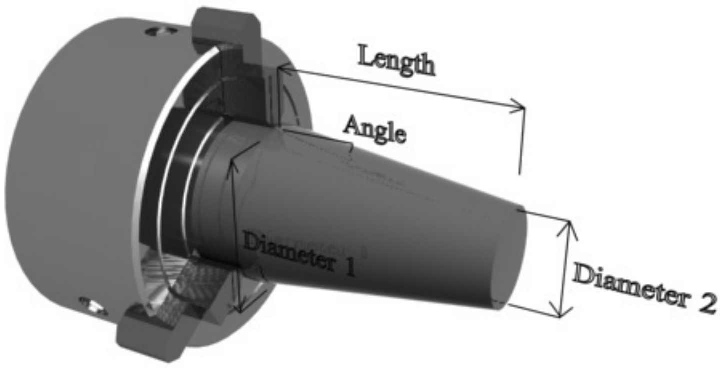


Figure 3.7: Taper feature parameters.

3.2.3 Convex/Concave Features

Length : Length of convex/concave feature

Diameter1 : Spindle side diameter of convex/concave

Diameter2 : Tailstock side diameter of convex/concave

Radius : Radius of curvature

Cx : Distance between center of curvature to Z-axis in X direction.

Cz : Distance between center of curvature to spindle side of feature in Z direction

Convex feature is used for creating rounded surfaces composed of convex planes whereas concave feature is used for creating rounded surfaces composed of concave planes. (Figure 3.8) Either R, or Cx and Cz parameters are required. Cx and Cz values can be positive or negative to obtain the required radius value.

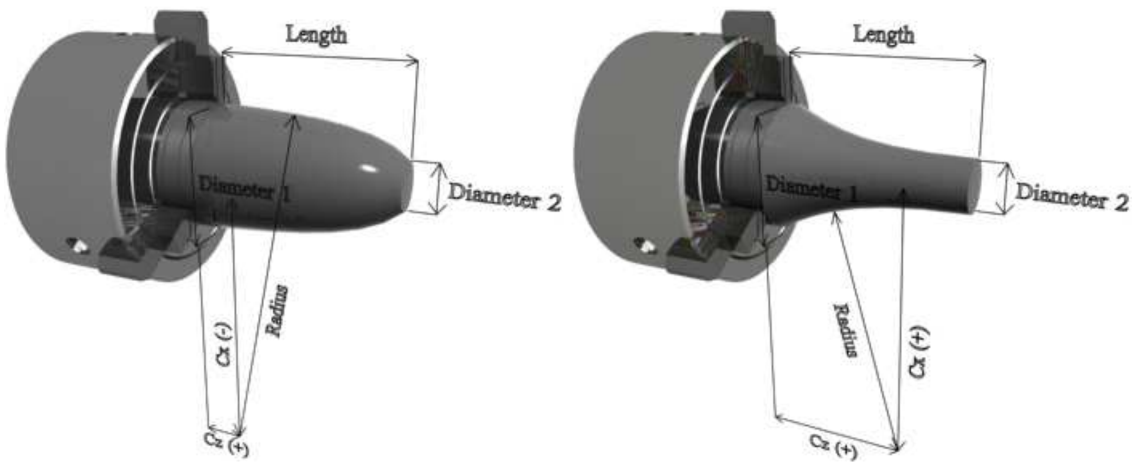


Figure 3.8: Convex and concave feature parameters.

3.2.4 Prism Feature

Length : Length of prism

Diameter : Diameter of circumscribing or inscribing circle

Angle : Rotation angle of prism about Z axis

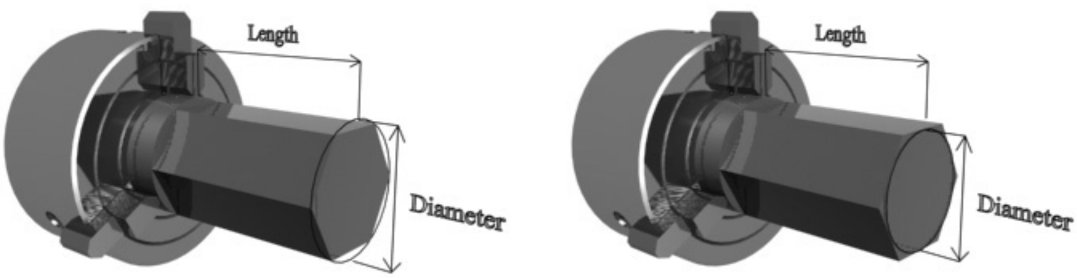


Figure 3.9: Inscribed and circumscribed prism feature parameters.

Prism feature has a polygonal profile which can be inscribed in or circumscribed about a circle and have N number of sides to extrude about Z axis . It must be machined by using the milling capabilities of turning center. Prism feature also produces step features combined each other by a revolution axis. Therefore using prism feature instead of step in some circumstances can be more practical for users.

3.3 Part Material Geometry (Turning)

The main part geometry is designed after blank geometry definition and produced by using axisymetric features which will be machined by turning capabilities of the machine. Defining the outer part profile geometry is very similar to outer blank profile geometry. The features are created starting from the spindle side to the tailstock side. The same primitives are also used to form the internal cavity of the part. As for the blank definition, features are placed starting from the tailstock side and subtracted from the part shape to form internal cavity.

The features used for part geometry definition are cylinder, taper, prism, convex and concave which are same as in blank geometry definition. Parameters required for these features are also same as in outer and inner blank profile geometry definition. Therefore in this section; groove and thread features which are only used for creating part geometry definition will be introduced. (Figure 3.10)

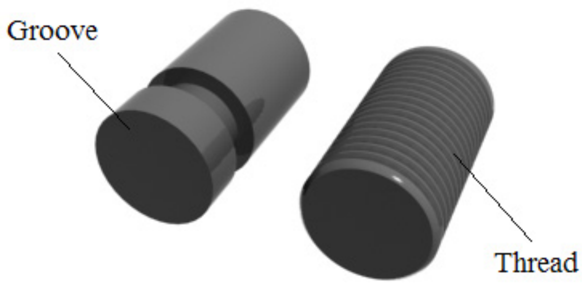


Figure 3.10: Features used to define part turning geometry.

3.3.1 Thread Feature

Standard : Standard of the thread

Type : Type of thread(Major Diameter,Pitch)

Angle : Taper angle of thread

Left/Right handed : Selection right/left hand of thread.

Thread feature is used for threading outer and inner circumferential surfaces of the part. The main parameters are given in Figure 3.11. The developed program comprise three standards which are Iso Metric, Fine Iso Metric and Conical Iso Metric. The necessary parameters for creating these standardized threads are taken from a machine elements handbook [25].

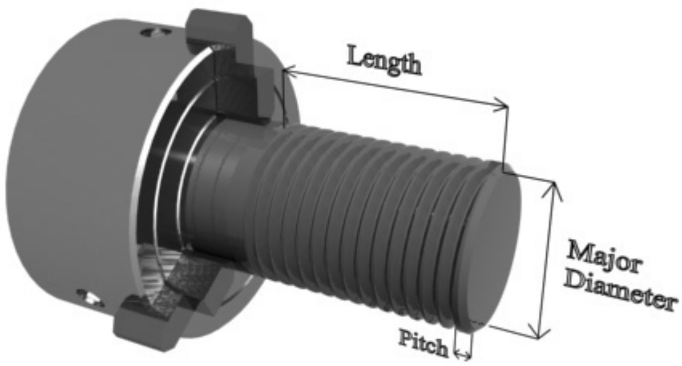


Figure 3.11: Thread feature parameters.

3.3.2 Groove Feature

Length : Distance from the spindle side of the part to the left edge of the groove

Width : Groove width

Depth : Groove depth

Diameter : Diameter of part where the groove feature is placed

Radius : Radius of the rounded groove

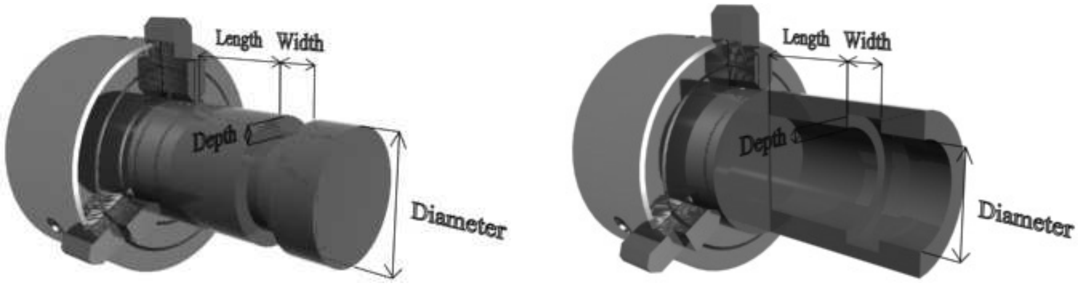


Figure 3.12: Groove feature parameters.

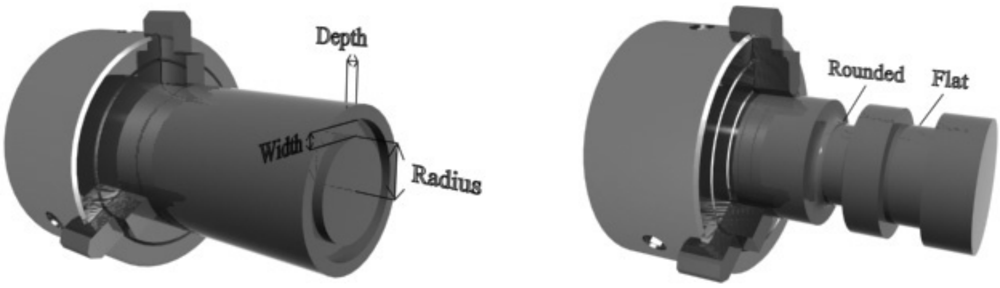


Figure 3.13: Facial groove feature and groove types.

Groove feature is superimposed on outer and inner circumferential or facial surfaces of the part. Produced with a round or flat bottom,(Figure 3.13b) side groove is available both for inner and outer profile geometries (Figure 3.12) but face groove is available only for outer profile geometry. (Figure 3.13a)

3.4 Chamfer/Round Settings

It can be possible to chamfer or round(fillet) the edges of the features used for inner and outer blank and part shape definition. The required inputs for chamfering are; direction which can be outwards or inwards followed by two lengths or one length and an angle.(Figure 3.14) For creating a round on an edge of a feature; direction and a radius value is required.(Figure 3.15)

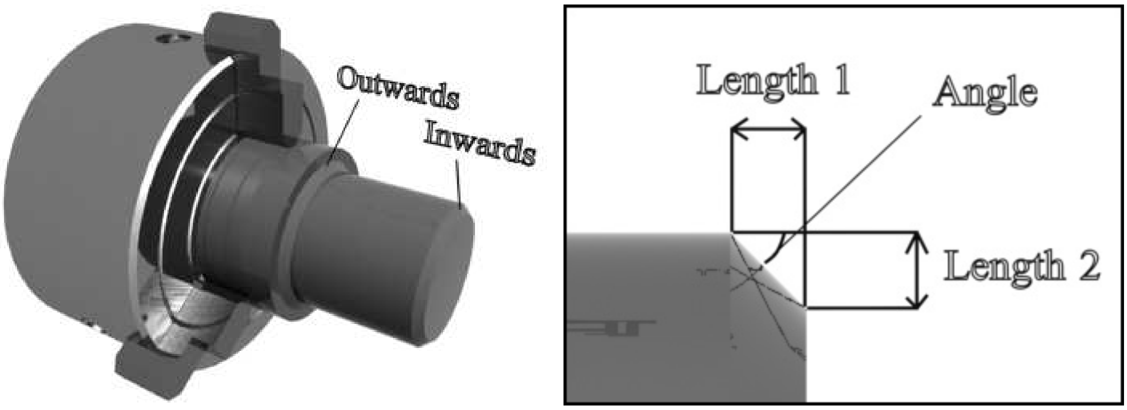


Figure 3.14: Chamfer feature parameters.

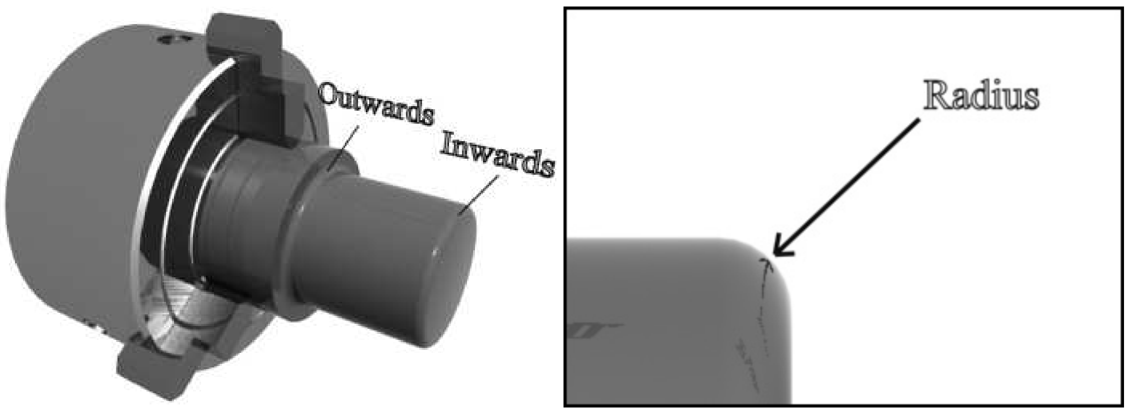


Figure 3.15: Round feature parameters.

3.5 Part Material Shape Definition (Milling)

Machining parts containing non-axisymetric features requires; milling machine or turning center which can be capable of milling on facial, lateral and circumferential surfaces. For this purpose the designer must use milling features after creating the part profile by using outer and inner turning features. Features for creating milling geometry are; hole, pocket, pocket-island, step and path.

Milling operation on a four-axis turning center consists of three main planes depending on the number of axes required to machine these features. (Figure 3.16) These are facial, lateral and circumference planes.

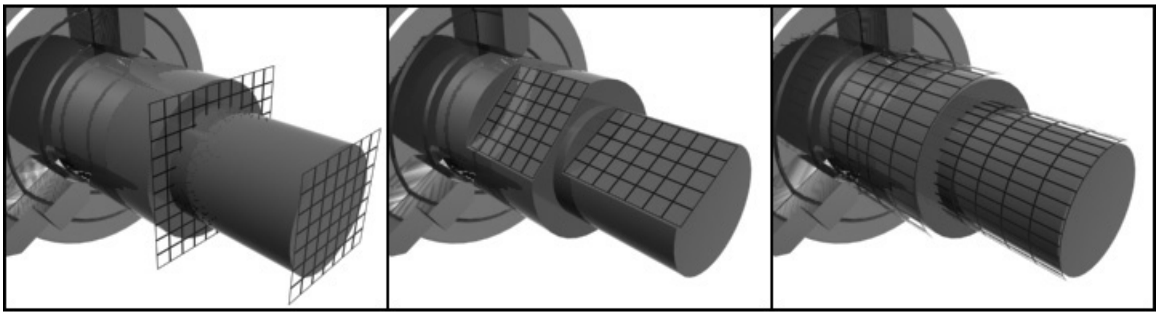


Figure 3.16: Visualization of facial, lateral and circumferential planes.

Facial plane features are planar features that are placed on faces parallel to the XY plane of the part. They require use of Z-axis rotating tool and tool movements along X,Y and C axis. Lateral features are also planar features which are placed on the side of the part and can be machined by using an X-axis rotating tool and requires X,Y,Z and C axes movements. Finally circumferential features are placed on the side of the part to remove material from the circular surface of the main geometry and can be machined by using X-axis rotating tool and X,Z and C axes movements. The detailed explanation and visualization of each feature are given in next figures starting from Figure 3.18 to Figure 3.30.

3.5.1 Hole Feature

Diameter : Diameter of hole

Depth : Depth of hole

Pattern : Single/Radial/Rectangular

Hole Style : Plain/Counterbore/Countersink

Thread Info : Major diameter and depth

Bottom Style : Flat/Angled

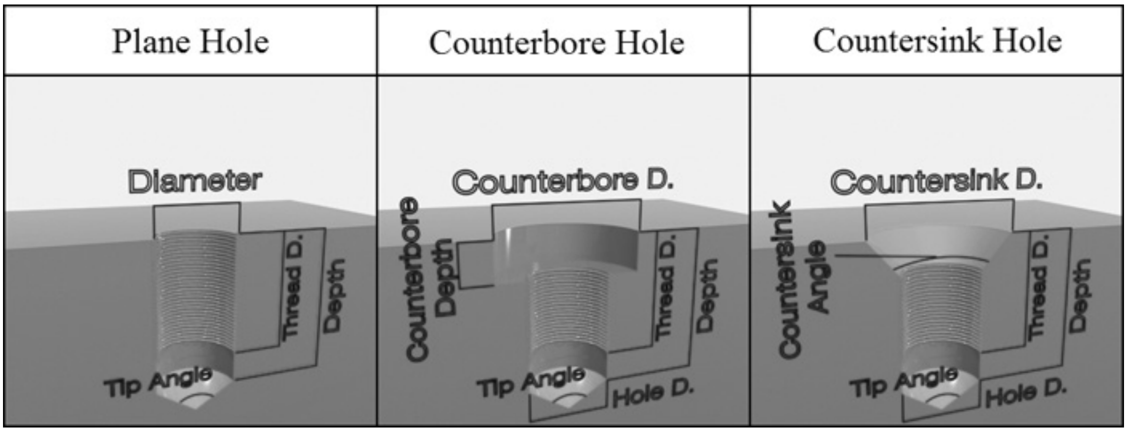


Figure 3.17: Hole feature types and parameters.

As it can easily be seen from Figure 3.17; A hole feature is defined as plain, counterbore or countersink while its bottom may be flat or angled produced by a drill tip. Threaded holes can also be defined by selecting the type of thread from the list of standards together with a given depth therefore the designer does not have to refer anywhere else to find out standard thread diameters and pitch values. A bore diameter and a depth value is needed to define a counterbore hole on the other side a depth value following by an angle must be entered to define a countersink hole.

Hole features can be defined as single, rectangular patterned or radial patterned, which are placed on the facial, lateral and circumferential planes. The necessary parameters for creating a single hole on facial plane are; distance between center coordinates of the hole to the origin of the XY plane(CenterX,CenterY) and to the spindle side of workpiece. Radially patterned holes are created by entering the distance between center of pitch to the origin of XY plane and to spindle side of workpiece following by a start angle, fill angle, pitch diameter and number of holes. (Figure 3.18)

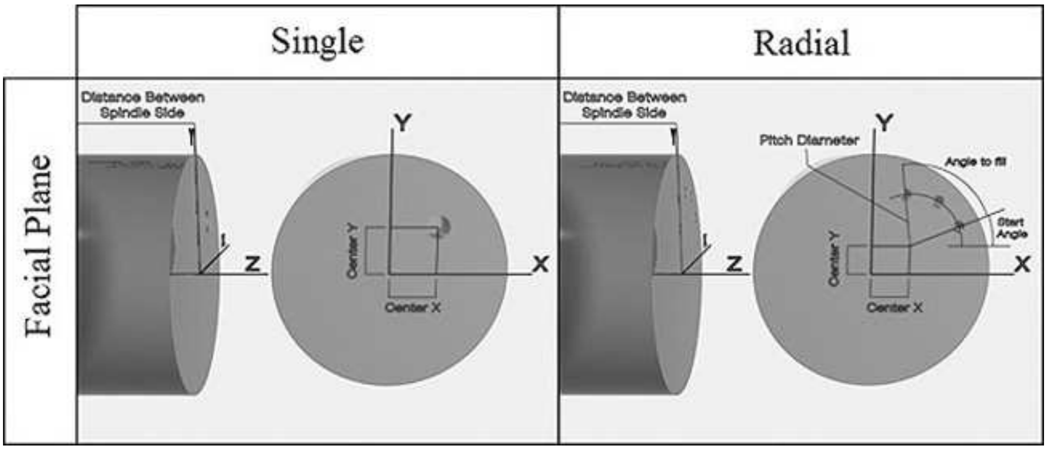


Figure 3.18: Single and radial patterned hole feature on facial plane.

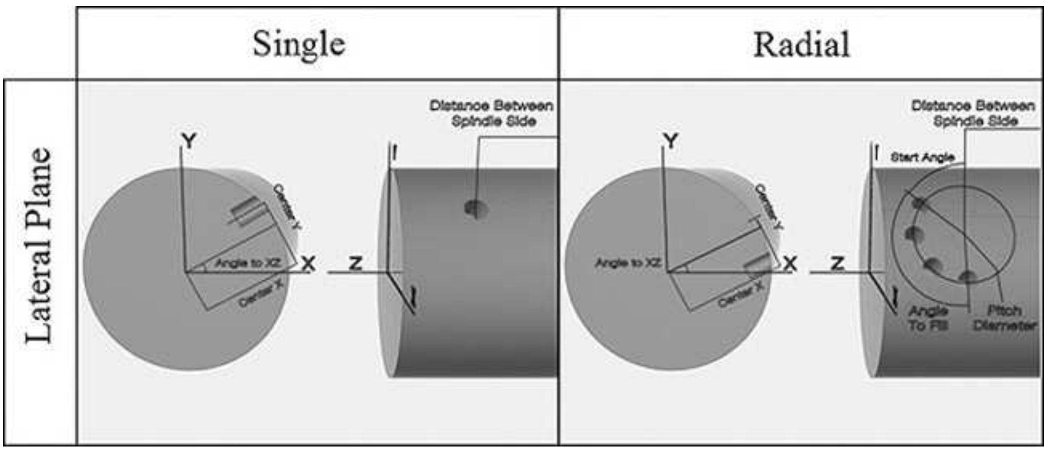


Figure 3.19: Single and radial patterned hole feature on lateral plane.

Holes created on lateral planes are created by using different parameters. These are the radial distance between the origin of XY plane to the lateral plane in which the hole center point is lying on (CenterX) and the projection of off-center distance on XY plane. (CenterY) The angle between centerX to XZ plane and the distance between Z coordinate of the hole center point to the spindle side of the workpiece. Radially patterned holes are created by using the same parameters except this time CenterX and CenterY refers to the center of the pitch where holes are radially arranged. (Figure 3.19)

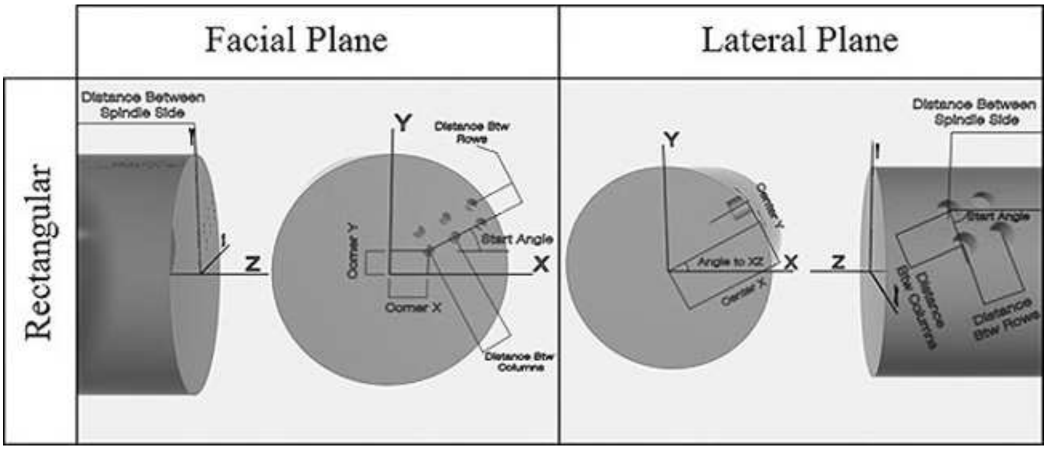


Figure 3.20: Rectangular patterned hole feature on facial and lateral plane.

Rectangular pattern are only allowed on facial and lateral planes. The necessary parameters are; distance between center of the principle hole to the origin of the XY plane and to the spindle side of the workpiece, start angle from the positive X axis, number of rows, number of columns, distance between rows and distance between columns. (Figure 3.20)

The necessary parameters for creating a single hole feature on circumference plane are; CenterX, angle between CenterX and XZ plane and the distance between Z coordinate of hole center point to the spindle side of the workpiece. Radially patterned holes requires same parameters combined with fill angle, number of holes, number of rows and distance between rows. (Figure 3.21)

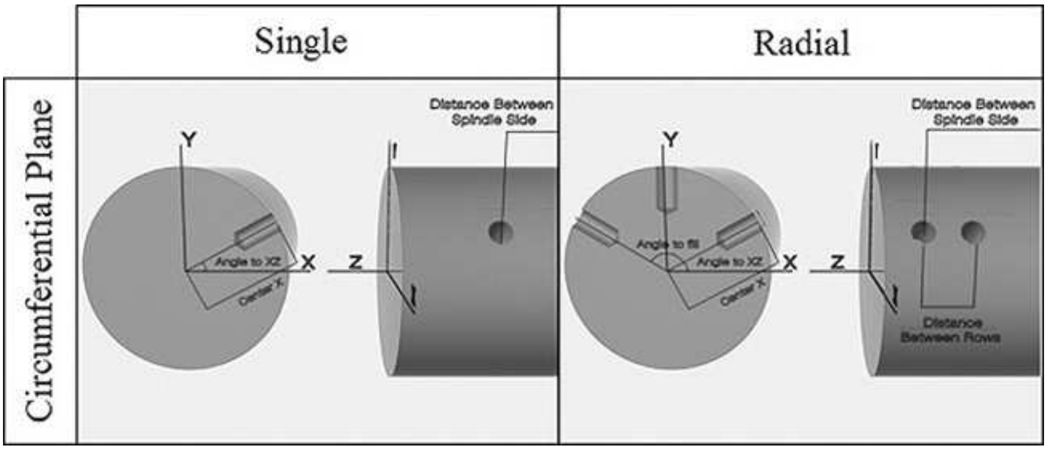


Figure 3.21: Single and radial patterned hole feature on circumferential plane.

3.5.2 Pocket Feature

Type : Rectangular/Circular

Depth : Depth of pocket

Width/Length : Width/Length of rectangular pocket

Radius : Corner radius of rectangular pocket

Diameter : Diameter of circular pocket

Pocket feature is used to mill out pocket formed shapes from the main geometry. Pocket features can be circular or rectangular and are placed on facial lateral and circumferential planes. The required parameters to create a pocket feature on facial plane are; distance between center coordinates of the pocket to the origin of XY plane (OffsetX,OffsetY) and to the spindle side of the workpiece. Diameter and depth values are necessary for creating circular pocket on the other side length, width, corner radius and depth values are necessary for creating rectangular pocket. (Figure 3.22)

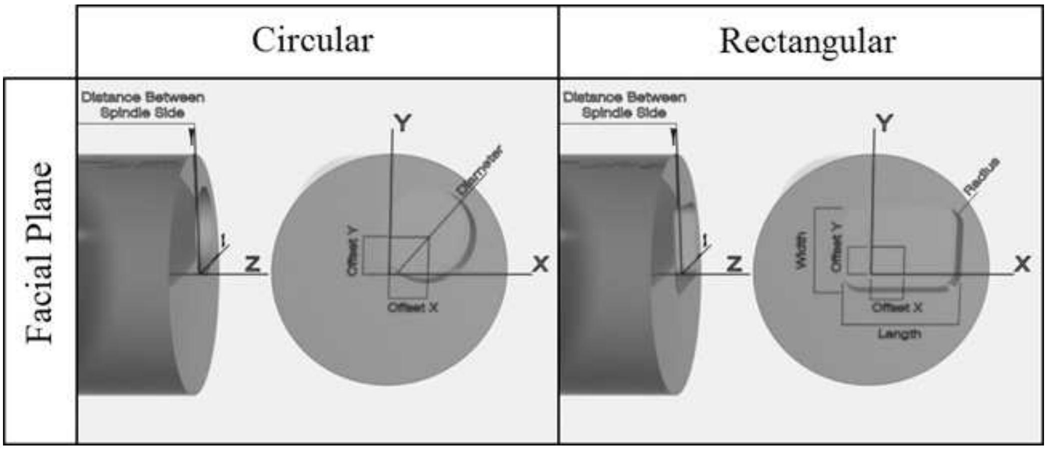


Figure 3.22: Circular and rectangular pocket feature on facial plane.

The necessary parameters for creating a pocket feature on lateral plane are; The radial distance between the origin of XY plane to the lateral plane in which the pocket is machined (Distance), the depth of pocket, the projection of off-center distance on XY plane (OffsetY), the angle to XZ plane and the distance between Z coordinate of the center of pocket to the spindle side of the workpiece. Other values are same as in facial plane. (Figure 3.23)

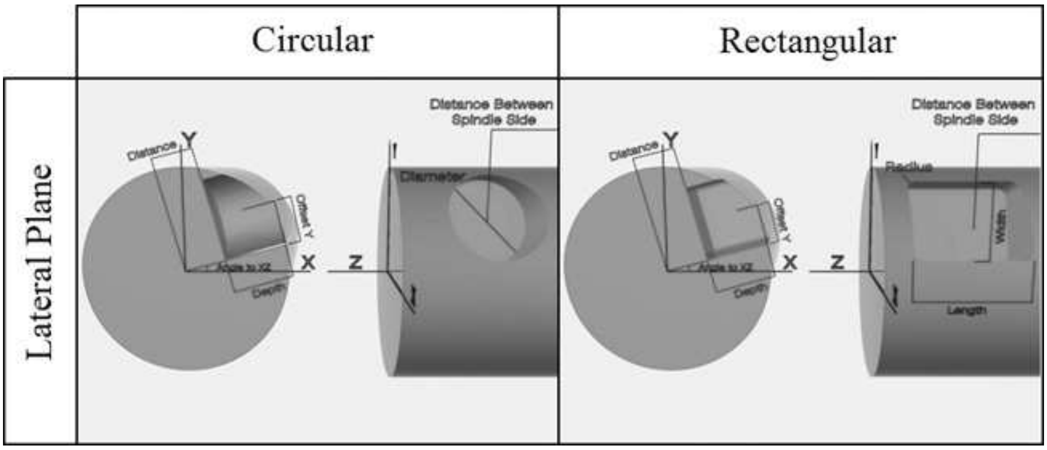


Figure 3.23: Circular and rectangular pocket feature on lateral plane.

Only rectangular pocket feature is allowed to define on circumferential plane. (Figure 3.24) The required parameters are same as in lateral plane except angle XZ is start angle of pocket and width value is replaced by included angle in order to define in polar coordinates.

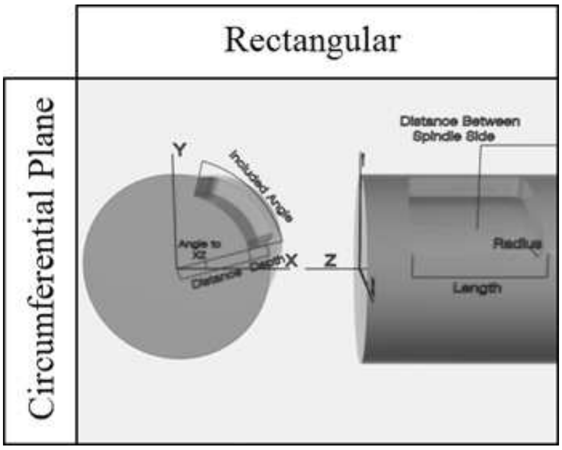


Figure 3.24: Rectangular pocket feature on circumferential plane.

3.5.3 Pocket-Island Feature

Type : Rectangular/Circular pocket-island

Depth : Depth of pocket-island

Length : Length of rectangular pocket-island

Width : Width of rectangular pocket-island

Radius : Corner radius of rectangular pocket-island

Diameter : Diameter of circular pocket island

Pocket-Island feature is very similar to pocket feature except its extrusions which is called islands. First the outer pocket geometry is defined similar to the pocket feature then island is created inside the pocket with given parameters. Most of the parameters are same except; width and offsetY values which are used for defining rectangular pocket-island on facial and lateral planes and are replaced with included angle and offset angle parameters to define on circumferential plane. (Figure 3.25) The profile definition of island is done similar to pocket feature since their profiles are grouped as circular and rectangular. Therefore parameters of island are defined with respect to the center of the owner pocket.

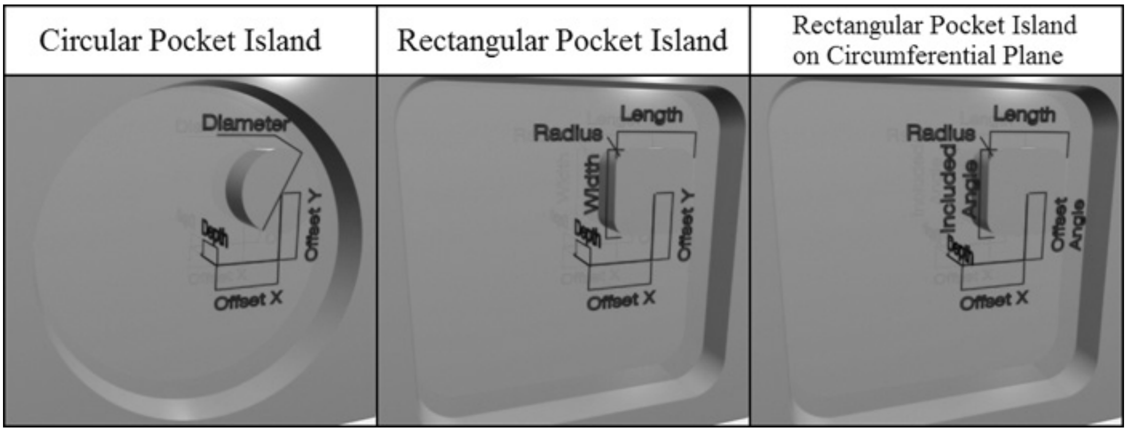


Figure 3.25: Circular and rectangular pocket-island feature on each milling plane.

3.5.4 Path Feature

Type : Cross section of path feature (Straight/Round/Dovetail/T-slot/Y-slot)

Depth : Depth of path feature

Width : Width of path feature

Tail Depth : Tail depth of path feature for T-slot and Y-slot

Neck Width : Neck width of path feature for T-slot and Y-slot

Angle : Angle of path feature for dovetail and Y-slot

Path feature is obtained by driving a mill cutter along a path. The cross section of path feature is defined as rectangular for an end mill cutter, T-shaped for a T-slot cutter, dovetail shaped for dovetail cutter or rounded for a ball end mill. (Figure 3.26) Path features are defined as linear or circular. Linear path feature created on facial plane is defined by entering X and Y coordinates of start and end points with respect to the origin of XY plane and Z coordinate to the spindle side of workpiece. However circular path feature is defined by entering the center coordinates of the circular path following with a start angle, end angle and the radius of the path. (Figure 3.27)

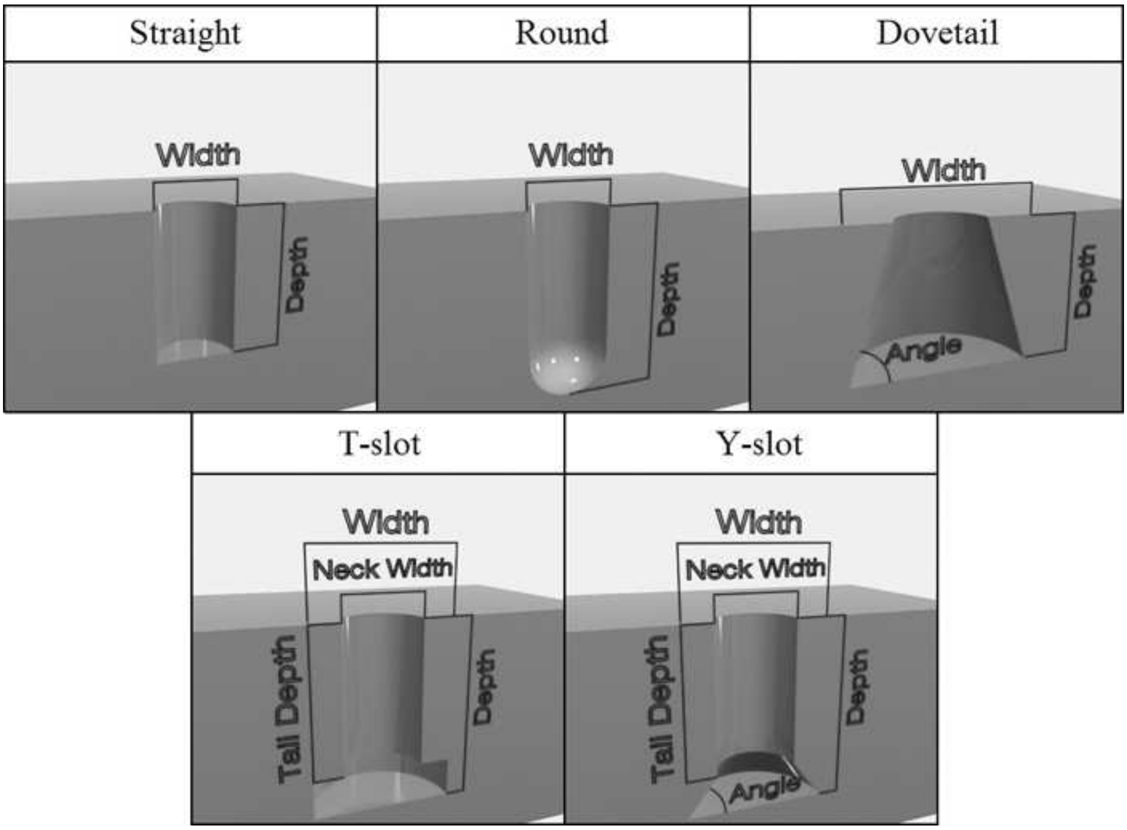


Figure 3.26: Visualization of path cross-sections.

Linear path feature created on lateral plane is defined by the radial distance between the origin of XY plane to the lateral plane where the path is created on (CenterX), the projection of off-center distance for start and end points on XY plane (StartY, EndY), the angle between centerX to XZ plane and the distance between Z coordinate of the start and end points to the spindle side of the workpiece whereas the parameters required for circular path is same except centerY.(Figure 3.28)

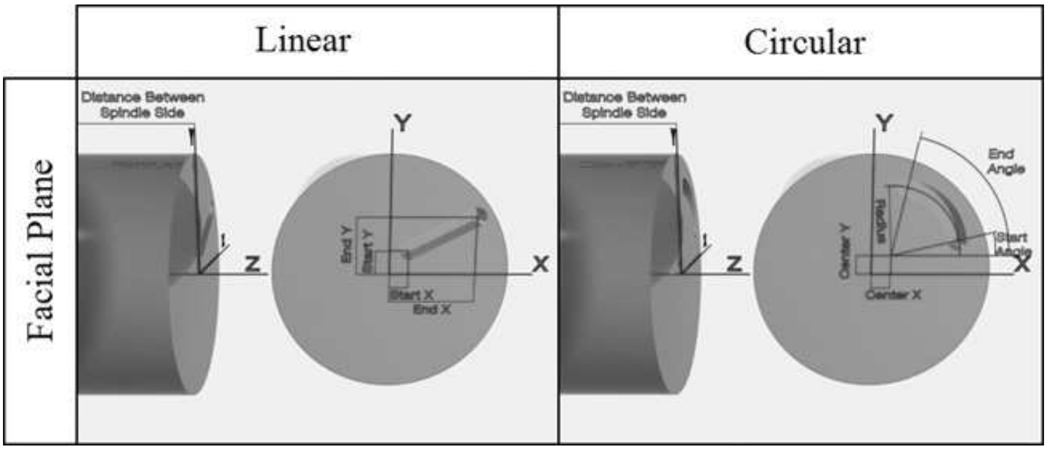


Figure 3.27: Linear and circular path on facial plane.

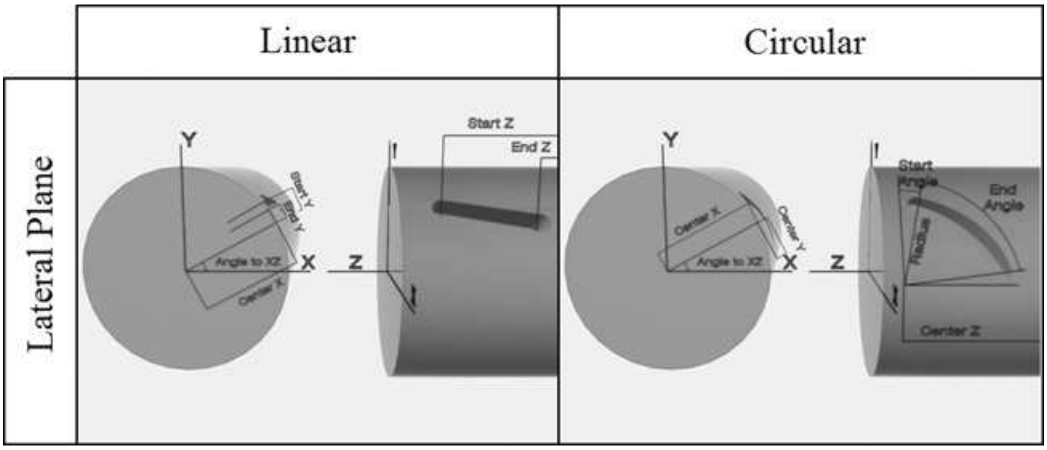


Figure 3.28: Linear and circular path on lateral plane.

Only linear path feature is allowed to define on circumferential plane. Therefore the necessary parameters are; CenterX, which is defined same as in lateral plane, start and end angles and the distance between Z coordinate of the start and end points of the path to the spindle side of the workpiece.(Figure 3.29)

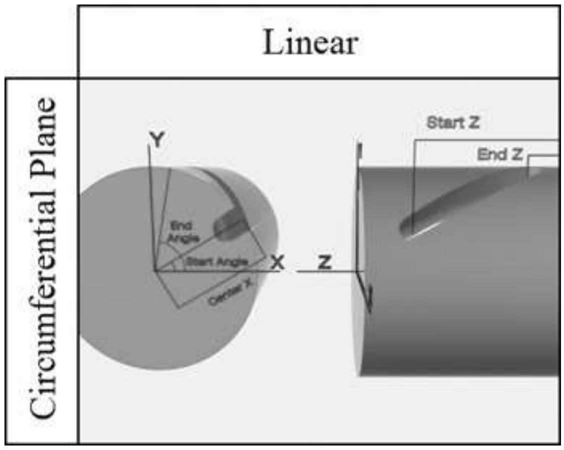


Figure 3.29: Linear path on circumferential plane.

3.5.5 Step Feature

Depth : Depth of step feature

Length : Length of step feature

Step feature is used to define side and face milled out sections of the part. Especially used for creating rectangular or prismatic parts by cutting the side surfaces of cylindrical raw parts using milling operations.

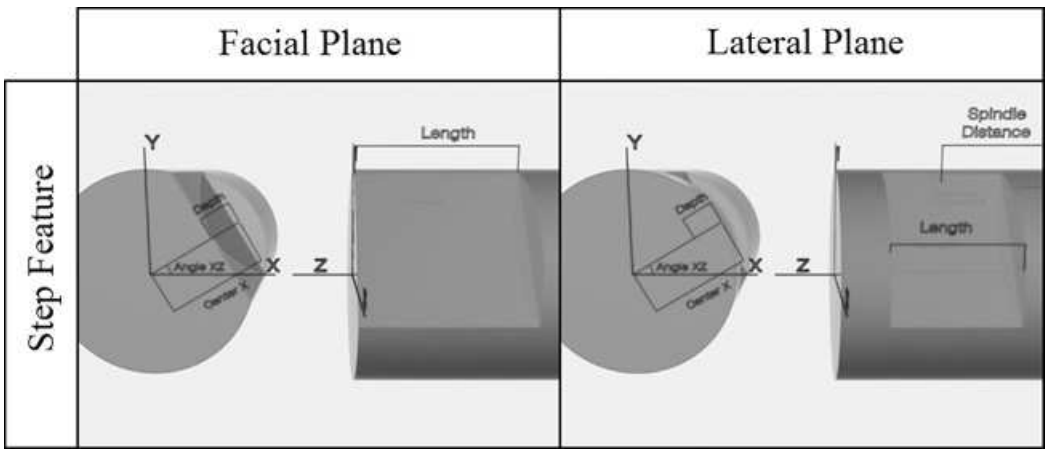


Figure 3.30: Visualization of facial and lateral step feature.

A radial distance defined by an angle which is calculated with respect to the XZ plane, depth and length of the step feature are the necessary parameters required to create on facial plane. In addition to these parameters the distance between the center of step which is on the middle of its length, to the spindle side of the workpiece are necessary to create a step feature on lateral plane.

3.6 Defining geometry in terms of CAD features

A feature based design software has simple and complex features. These features are used by designers to create parts that are going to manufacture and consists of some parameters which can be easily changed for necessary corrections or redesigns. The most common used features in CAD programs are extrude, revolve, chamfer and fillet following by loft, sweep, coil, hole, thread and so on. The machining features in previous section were created individually by using these which are most relevant to the operation to be manufactured.

3.6.1 Revolve Feature

Revolve features represents the nature of turning operation. A sketch with the 2D profile of the part, a revolution axis and an angle are needed for creating a revolve feature. (Figure 3.31) Cylinder, taper, concave, convex features are created by boolean addition whereas groove feature is created by boolean subtraction.

3.6.2 Extrude Feature

Prism, step, pocket and pocket-island features are created by using extrude feature. The feature is defined by extruding a 2D sketch to a predefined height. A taper angle can be given optionally. Prism feature is created by boolean addition but milling features are created by boolean subtraction. (Figure 3.32)

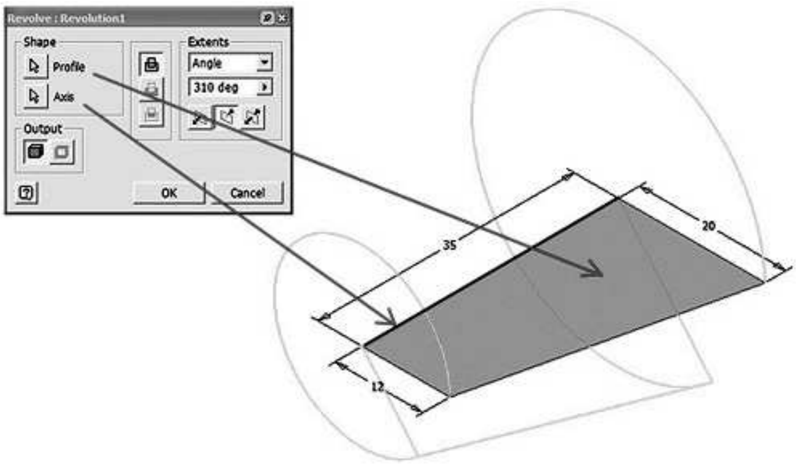


Figure 3.31: Revolve feature representation.

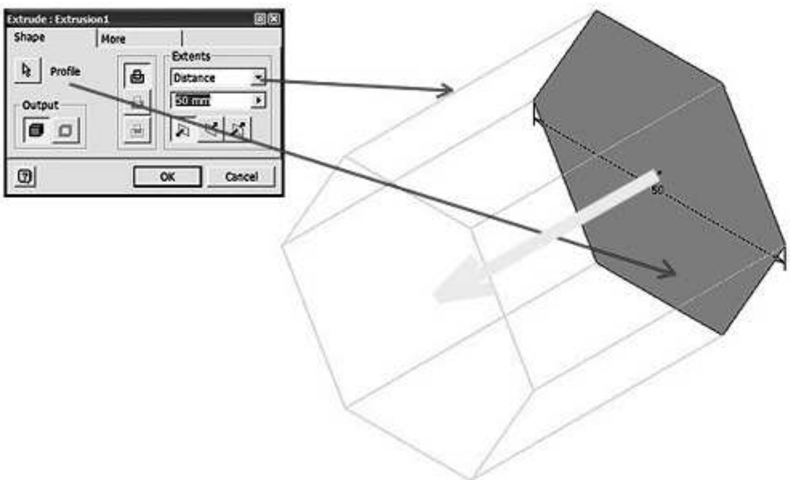


Figure 3.32: Extrude feature representation.

3.6.3 Chamfer and Fillet Feature

Chamfer and fillet features are used for chamfering and creating round edges in turning operation for blank and part geometry definition. A basic chamfer feature has three optional creating methods in CAD programs. Distance, Distance-Angle and Distance-Distance. The first method creates a chamfer of a distance given by the user and assumes angle of 45 degrees. Second method takes angle and distance values individually and finally last method creates a chamfer by using two different distances. (Figure 3.33) However a fillet feature requires only an edge and a radius value. (Figure 3.34)

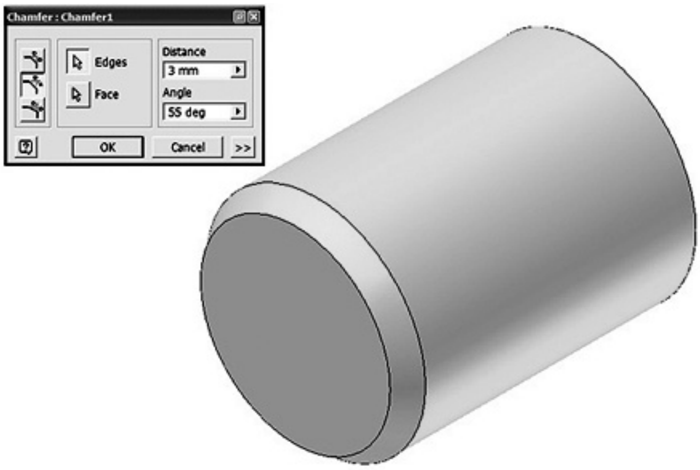


Figure 3.33: Chamfer feature representation.

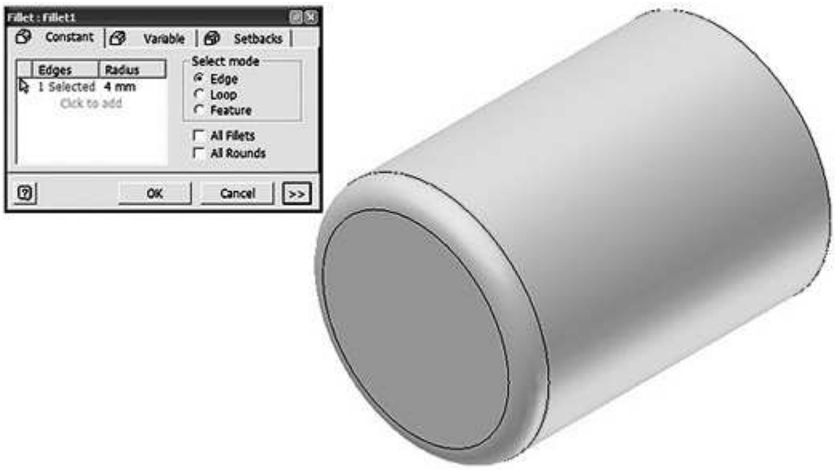


Figure 3.34: Fillet feature representation.

3.6.4 Coil Feature

Coil feature is used for creating inner and outer threads in turning of part material design. In order to create a full sized metric thread; a cylinder feature with a major diameter value of the thread is created. Then a 2D profile of the thread is sketched by using its major diameter and pitch value with respect to the given parameters in machining handbook [25]. By knowing the full size height of the thread; the helical path of the coil feature is created using the method of "Pitch and Height" and boolean subtracted from the cylinder feature. Coil features are also used for creating conical threads by subtracting the feature created with an optionally given taper angle from a previously created taper feature.(Figure 3.35)

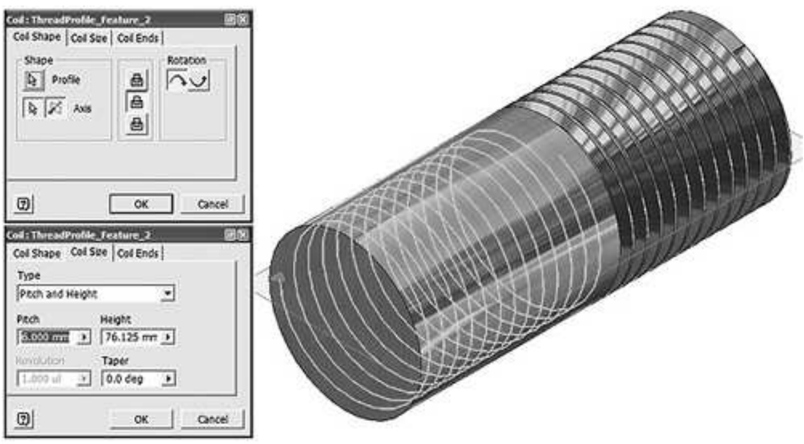


Figure 3.35: Coil feature representation for creating threads.

Path features on circumferential plane is also created by coil feature but this time 2D sketch must be the profile of the end mill cutter. It can be straight, round, T-slot, dovetail or Y-slot. A revolution value is calculated with the given parameters of path feature. Finally the method of "Revolution and Height" is used and path feature is created by boolean subtraction. (Figure 3.36)

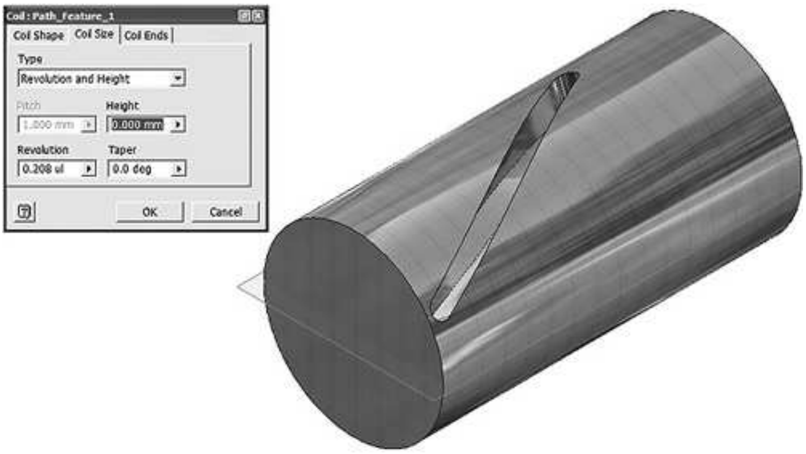


Figure 3.36: Coil feature representation for circumferential path.

3.6.5 Hole Feature

Hole feature is used especially for creating single, rectangular or radial patterned holes on facial, lateral and circumferential planes. (Figure 3.37) Radial patterned holes defined on circumferential plane is created by using circular pattern feature shown in Figure 3.38.

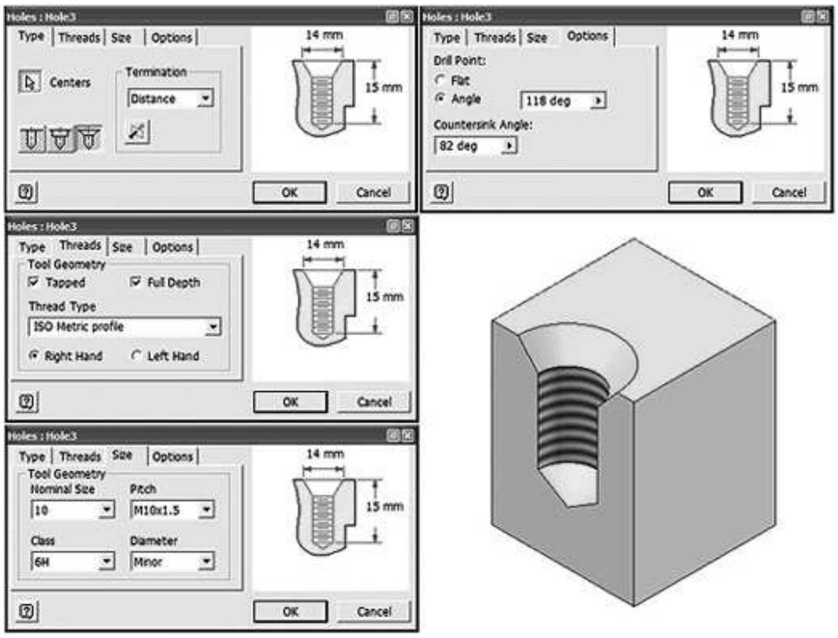


Figure 3.37: Hole feature representation.

Figure 3.38: Radial patterned hole on circumference plane.

3.6.6 Subsets of primitive library for 2 and 3-Axis machining

It is possible to design manufacturable parts for two-axis lathes and three-axis turning centers by limiting the use of features existing in the program because feature sets for two-axis and three-axis turning center machining are subsets of the primitive library that can be manufactured using four-axis turning center.

Turning features which represents turning operations can be manufactured by using either two(X,Z) or three(X,Z,C) axis lathes because X and Z axis are enough for machining these features. Milling features placed on facial and circumferential planes can manufactured by using three or four(X,Y,Z,C) axis CNC turning centers which are capable of C axis movement. In addition to that Y axis is required to manufacture prism and milling features created on lateral surface of the part therefore these features can only be machined by using four-axis CNC turning centers. Table 3.1 shows availability of features for two-axis, three-axis and four-axis machining from the developed feature library.

Table 3.1: Availability of features for 2-axis, 3-axis and 4-axis machining.

	2-Axis (X,Z)	**3-Axis (X,Z,C)**	**4-Axis (X,Y,Z,C)**
Turning Features			
Cylinder	√	√	√
Taper	√	√	√
Convex	√	√	√
Concave	√	√	√
Thread	√	√	√
Groove	√	√	√
Prism	–	–	√
Milling Features			
Facial Hole	–	√	√
Facial Pocket	–	√	√
Facial Pocket-Island	–	√	√
Facial Path	–	√	√
Facial Step	–	√	√
Lateral Hole	–	–	√
Lateral Pocket	–	–	√
Lateral Pocket-Island	–	–	√
Lateral Path	–	–	√
Lateral Step	–	–	√
Circumferential Hole	–	√	√
Circumferential Pocket	–	√	√
Circumferential Pocket-Island	–	√	√
Circumferential Path	–	√	√

3.7 Process Planning and Cutting Tool Selection

Process planing represents the machining sequence of the previously designed features. In developing these sequence user is free to select or change the priority of features. The developed process planning module follows a design procedure which performs machining processes automatically. While sequencing milling operations; program follows an algorithm which orders operations with respect to its work plane so as to minimize the change of cutting tool and machining time. The sequences of these processes are in the following order.

3.7.1 Process Sequence

1. Rough face turning.
2. Rough external(outer) turning.
3. Rough internal(inner) turning.
4. Finish external(facial and outer) turning.
5. Finish internal(Inner) turning.
6. External grooving,threading and prism cutting.
7. Internal grooving and thread cutting.
8. Machining of lateral milling features in the order of user definition.
9. Machining of circumferential milling features in the order of user definition.
10. Machining of facial milling features in the order of user definition.

3.7.2 Cutting Tool Selection

The main aim is to select cutting tools for each operation or feature. Previously written version of the program is able to select one cutting tool per feature. Therefore; the code generation for each process where more than one tool is needed were a problem for user.

The developed algorithm allows users to create a list of selected tools together with all the necessary parameters for machining the related operation. (Figure 3.39) Besides it allows selecting cutting tools infinitely so the number of defined cutting tools can be changed for each operation or feature. In order to define a cutting tool for a previously designed workpiece, the parameters used in classification have to be understand carefully.

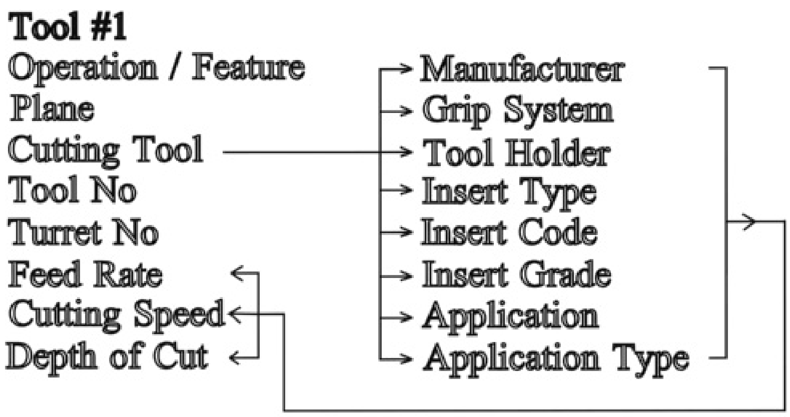


Figure 3.39: Classification of tool definition algorithm.

Operation/Feature parameter states the related operation or feature for the user to define the desired cutting tool. Operation word is declared as combination of features. The program has rough and finish turning operations which contains turning features of cylinder, taper, convex and concave combined with the pre-machined forms of thread and prism features. Feature word is declared for features which has to be machined after rough and finish turning.

Plane parameter defines the cutting plane for the previously defined operation or feature. For turning related operations or features; this parameter can be external or internal. For milling features this parameter can be facial, lateral or circumferential.

Cutting tool parameter can get various choices. For rough external turning it can be a left hand, right hand, natural or even a groove turning tool. On the other hand rough internal turning operation can be machined by using a drill followed by a left hand, right hand boring bar or a grooving tool. So at least two cutting tools must be selected for rough internal cutting where one is enough for cutting some geometries on rough external turning. All associated cutting tool types with related features or operations are stated in Table 3.2.

After selecting the proper cutting tool to machine the defined operation, some advanced configurations have to be considered. In order to define the machining parameters and some important tool dimensions necessary to generate the NC code, following parameters listed must be entered properly.

1 : Manufacturer of cutting tool.

2 : Grip system of tool holder.

3 : Model and dimensions of tool holder.

4 : Application for the tool to be used.

5 : Insert class for the tool holder.

6 : Insert code and dimensions of insert.

7 : Type of application.

8 : Grade of insert.

The dimensions of tool holder are stored with respect to the selected grip system and tool holder model. On the other side the dimensions of insert are stored with respect to the selected insert class and insert code.

Table 3.2: Associated cutting tool types with features.

	Cutting Tool Type
Turning Features	
Outer Turning Features (Cylinder, Taper, Convex, Concave)	Left, Right, Natural H. Turning Tool Left, Right H. Grooving Tool
Inner Turning Features (Cylinder, Taper, Convex, Concave)	Left, Right H. Boring Bar Drill, Left, Right H. Grooving Bar
Thread	Left, Right H. Thread Turning Tool & Bar
Groove	Left, Right H. Groove Turning Tool & Bar
Prism	Turning Tool, End Mill
Milling Features	
Hole	Center Drill, Drill, Reamer End Mill, Countersink Cutter, Tap
Pocket	End Mill
Pocket-Island	End Mill
Path	End Mill, Ball End Mill, Dovetail, T-Slot
Step	End Mill

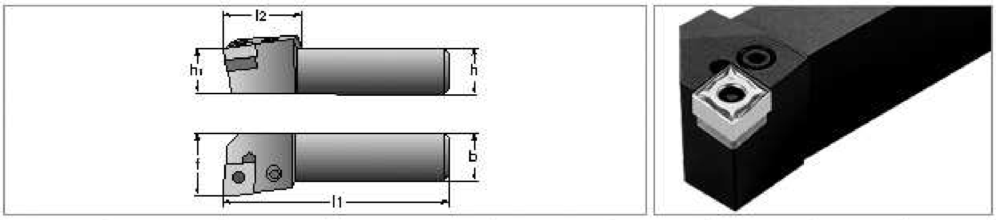
Tool No refers to the cutting tool number stated in machine library and ***Turret No*** refers to the attached turret number of the selected cutting tool.

Machining parameters such as ***Feed Rate***, ***Cutting Speed*** and ***Depth of Cut*** are entered with respect to the selected parameters of cutting tool therefore once configured and recorded to the database; future selections of same cutting tool will reference the proper machining parameters automatically. Besides manually entry can be done over automatically selected values.

An example of cutting tool definition for rough external turning operation is given in Table 3.3 with Figures 3.40 and 3.41 for illustration.

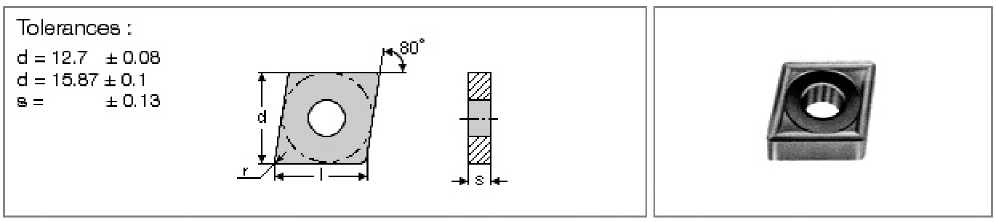
Table 3.3: An example of cutting tool definition.

	Tool#1		**Tool#1**
Operation / Feature	Rough Turning	**Plane**	External
Cutting Tool	Left Hand Turning T.	**Manufacturer**	ISCAR
Grip System	PCLNR/L	**Tool Holder**	2020K-12
Tool No	1	**Turret No**	1
Insert	CNMG	**Insert Code**	160616
Application	Rough Machining	**Insert Grade**	IC9025
Application Type	Roughing	**Feed Rate**	0.3 mm/rev
Cutting Speed	150 m/min	**Depth of Cut**	2 mm



h	h1	b	l1	l2	f	Ga°	Gr°	R/L
20.0	20.0	20.0	125.00	30.0	25.0	-6	-6	L

Figure 3.40: PCLN/R 2020K-12 tool holder.



l	d	S	r
15.90	15.870	6.35	1.60

Figure 3.41: CNMG 160616 IC9025 insert.

3.8 NC code and Tool Path Generation

3.8.1 Numerical Control Programming

A programming language is a system of symbols, codes, and rules that describes the manner in which program instructions can be written.

3.8.1.1 Sequence Number (N-Word)

A block normally starts with a sequence number that identifies the block within the part program. Most control systems use a four-digit sequence number allowing step numbers up to N9999. The numbers are usually advanced by fives or tens in order to leave spaces for additional blocks to be inserted later if required.

3.8.1.2 Preparatory Word (G-Word)

A preparatory word consists of a two digit code with a letter which addresses G word. The preparatory word is placed at the beginning of a block, normally following the sequence number. The G word indicates to the control system how to interpret the remainder of the block. The majority of G codes are designated as modal. Once used, the code remains in effect for the succeeding blocks unless it is specifically cancelled. The list of standardized G-code addresses together with detailed functions for using with Hitachi-Seiki turning center are given in Appendix C. The values shown in Table C.1 are taken from users manual of the machine [13].

Dimension words contain numerical data that indicate either a distance or a position. The dimension units are selected by using G70 inch or G71 metric programming code. The dimension words immediately follow a G-word in a block and on multiaxis machines should be placed in the following order: X, Y, Z, U, V, W and etc..

Absolute programming is a method of defining the coordinate locations of points based on the movement of the cutter with respect to machine zero point. Where as incremental programming is a method of identifying the coordinates of a particular location in terms of the distance of the new location with respect to the current location.

3.8.1.3 Miscellaneous Word (M-Word)

Miscellaneous functions are on-off type commands which are used to control actions such as starting and stopping off motors, turning coolant on-off, changing tools and, clamping and unclamping parts. M-functions are made up of the letter M followed by a two digit code. Table C.2 lists standardized M-codes together with detailed functions, which are used for the developed program to generate NC code using Hitachi-Seiki turning center. The values shown in Table C.2 are taken from handbook of the machine.

3.8.1.4 Feed Function (F-Word)

F-word is stand for feed rate value that depends on the system of units and feed mode in use. Depending on whether G20 or G21, which is used to indicate inch or metric programming and whether G98 or G99 which is used to specify feed rate as expressed in inches per minute or revolution per minute.

The feed function is modal, so it stays in effect until it is changed by setting a new feed rate. In a block, the feed function is placed immediately following the last dimension word of the axes to which it applies.

In turning operations, the spindle speed increases as the working diameter decreases. In order to prevent the spindle speed from increasing beyond a maximum value, the spindle function (S-word) is used to specify the maximum allowable spindle speed before using G50 command. if the spindle speed is changed after the G96 is used, the feed rate is also changed accordingly. If G97 is used to set a constant feed per unit of time, spindle speed changes do not vary feed rate.

3.8.1.5 Spindle Function (S-Word)

S word specifies the rotation speed of spindle. The spindle function is programmed by the address S followed by the number of digits specified in the format detail. In turning, constant spindle speed is applied for threading cycles and for machining parts in which the diameter remains constant. On the other hand constant surface speed is applied for facing and other cutting operations in which the diameter of the workpiece changes. The spindle speed is set to an initial value specified by the S-word and then automatically adjusted as the diameter changes in order to maintain a constant surface speed.

3.8.1.6 Tool Function (T-Word)

The T-word calls out the tool to be selected on a machining center or a lathe having an automatic tool changer or an indexing turret. This word also specifies the proper turret face on a lathe. The word usually is accompanied by several numbers where the first pair of two refers to the turret number and the second pair of two refers to the tool number following by an offset number.

3.8.2 Rough Outer Turning

NC code and tool path generation of rough outer turning operation consist of facing and external turning operations shown in Figure 3.42. As can be seen from the figure a clearance value for facial and side surfaces is given for finishing operation.

Facing operation is done if the blank material length is larger than part material length. G00 and G01 codes are used for cutting vertical pieces divided by a given depth of cut to the start of the face clearance value for finish pass. Figure 3.43 represents the tool path generation for facial cutting operation. Rapid movement is done before (1) and after (3,4) the cutting process (2).

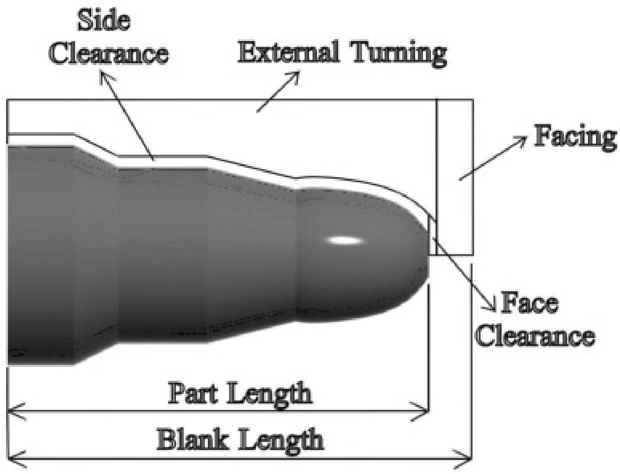


Figure 3.42: Rough outer turning representation.

External turning operation is done for roughing the surface with a given depth of cut till the start of the side clearance value for finish pass. Figure 3.44 represents the tool path generation for external turning operation. Rapid movement is done before (1) and after (3,4) the cutting process (2).

Finally when the external roughing operation is finished, the excess material on the surface of the side clearance can be removed with a single pass to form an offset surface. G01, G02 and G03 codes can be used to form the outer geometry of the workpiece with an offset value. (Figure 3.45)

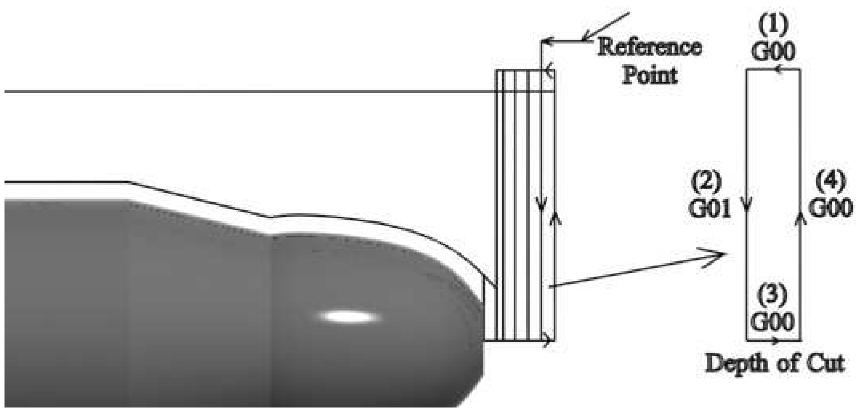


Figure 3.43: Tool path layout for facing.

The developed program is capable of generating NC code for geometries in which the outer diameter is decreasing from spindle side to turret side. Therefore geometries involving increasing diameter or combination of both cannot be machined with the current algorithm.

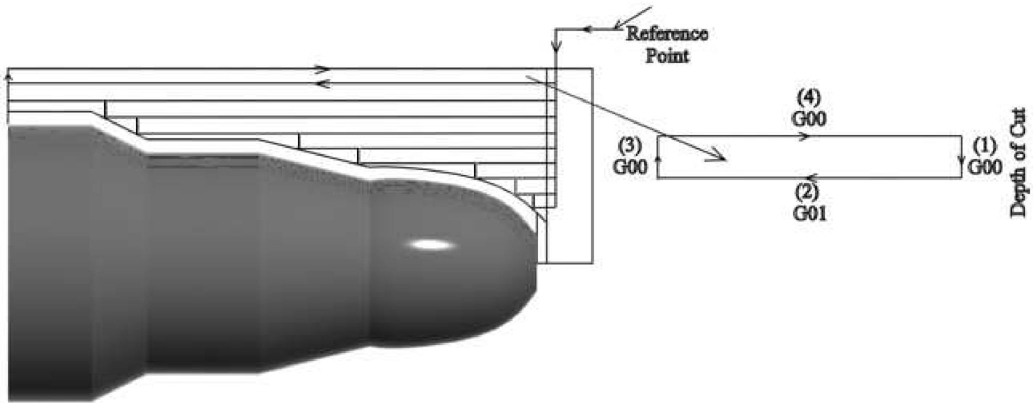


Figure 3.44: Tool path layout for external roughing.

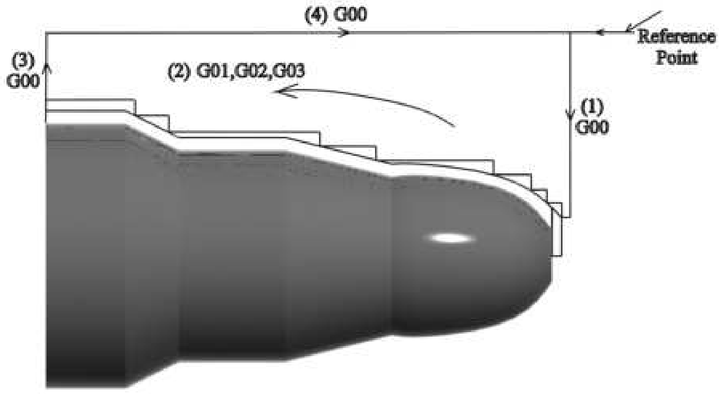


Figure 3.45: Final cut for external roughing.

3.8.3 Finish Outer Turning

Finishing operation is made to generate a good surface quality. Therefore a finish cut can be performed with respect to the given clearance values. The operation is done by rapid positioning (1) , cutting (2) and releasing (3,4). G01, G02 and G03 codes can be used to form the outer profile of the workpiece if necessary. (Figure 3.46)

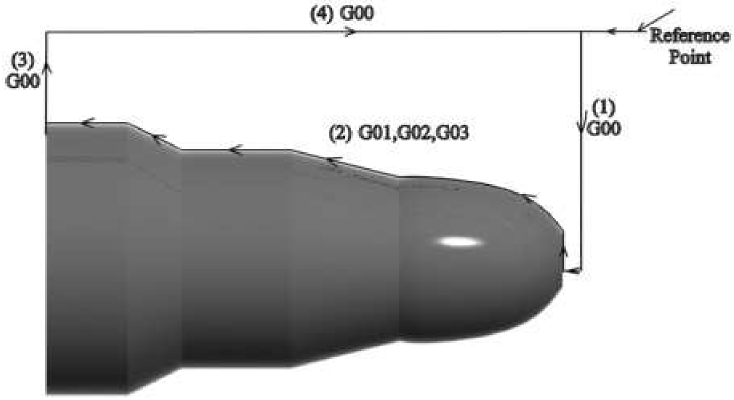


Figure 3.46: Tool path layout for external finishing.

3.8.4 Rough Inner Turning

NC code and tool path generation of rough inner turning operation consist of drilling and boring operations shown in Figure 3.47.

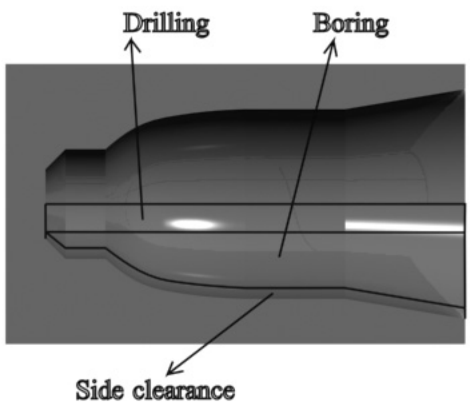


Figure 3.47: Rough inner turning representation.

Drilling operation is done prior to boring process to open an empty space for boring bar to move inside the workpiece. G83 code is used for drilling the minimum diameter of the inner profile. A canned cycle for drilling is handled till G80 code which cancels the operation is executed. Drilling is performed by drill diameter value for each pass. (Figure 3.48)

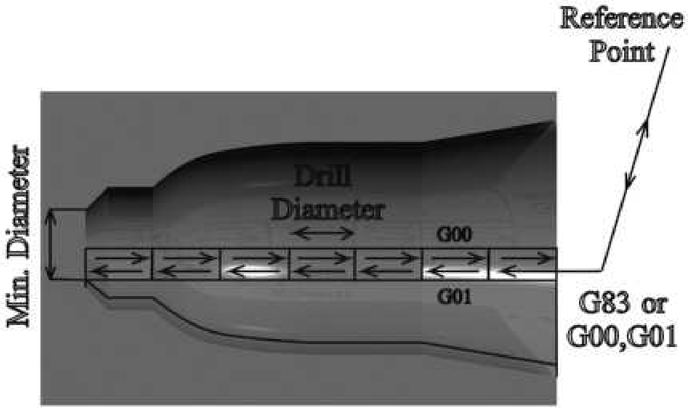


Figure 3.48: Tool path layout for drilling.

Boring operation is performed for roughing the surface with a given depth of cut from the start of the drilled diameter to the side clearance value for finish pass. (Figure 3.50) Rapid movement is done before (1) and after (3,4) the cutting process (2).

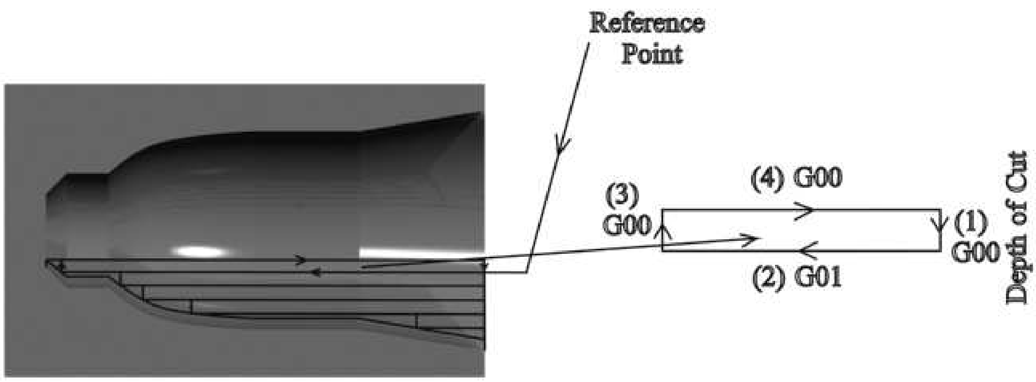


Figure 3.49: Tool path layout for internal roughing.

After the boring operation is finished, the excess material on the surface inside the side clearance can be removed with a single pass to form an offset surface. G01, G02 and G03 codes can be used for cutting process. (Figure 3.50)

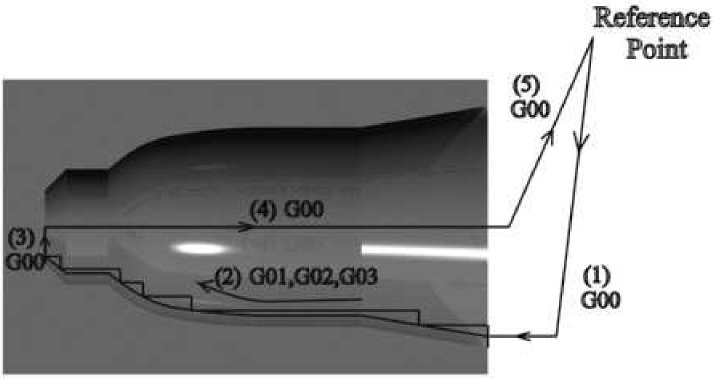


Figure 3.50: Final cut for external roughing.

3.8.5 Finish Inner Turning

Finally a finish cut is done with respect to the given clearance value. The operation is done by rapid positioning (1) , cutting (2) and releasing (3,4). G01, G02 and G03 codes can be used to form the inner profile of the workpiece. (Figure 3.51)

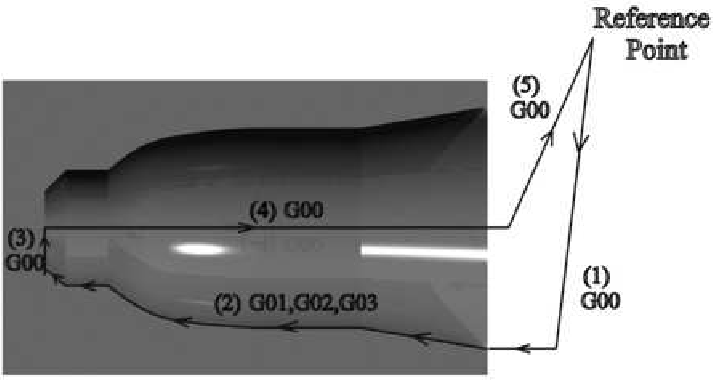


Figure 3.51: Tool path layout for internal finishing.

3.8.6 Grooving

Grooving operation can be made on external, internal and facial surfaces of the workpiece. NC code and tool path generation on external profile is done using G00 and G01 codes with a given depth of cut value seen in Figure 3.52. After machining the groove, a finish pass is applied to provide a good surface quality.

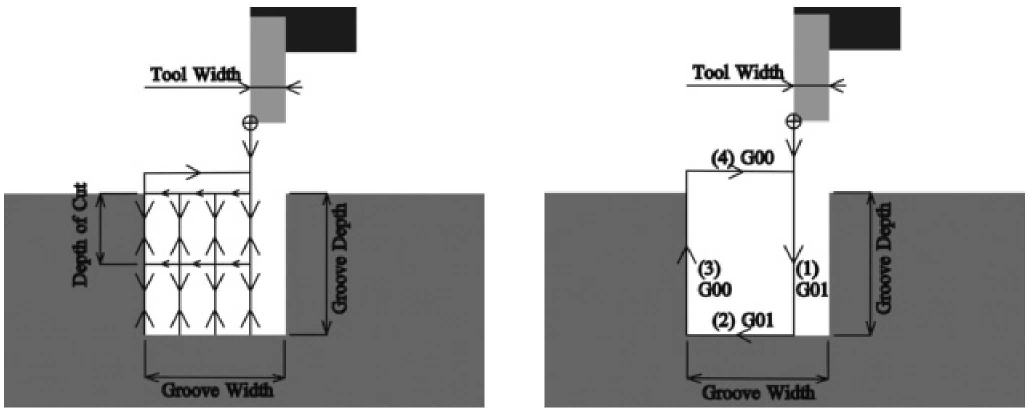


Figure 3.52: Tool path layout for external grooving.

The code generation principle is same for inner profile except the reversed feed direction and facial grooving can also be applied using G00 and G01 codes but this time the cutting algorithm is rotated 90 degrees clockwise. (Figure 3.53)

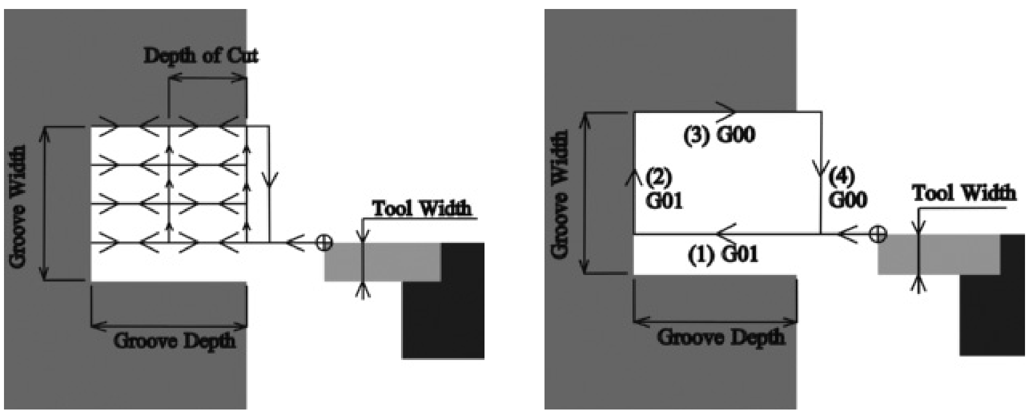


Figure 3.53: Tool path layout for facial grooving.

3.8.7 Threading

G32 command enables threading and tapping on external and internal profiles. Equation 3.1 must be observed in threading when using G32 command where N is the spindle speed in mm and P is the thread pitch in rpm.

$$N \leq \frac{10000}{P} \tag{3.1}$$

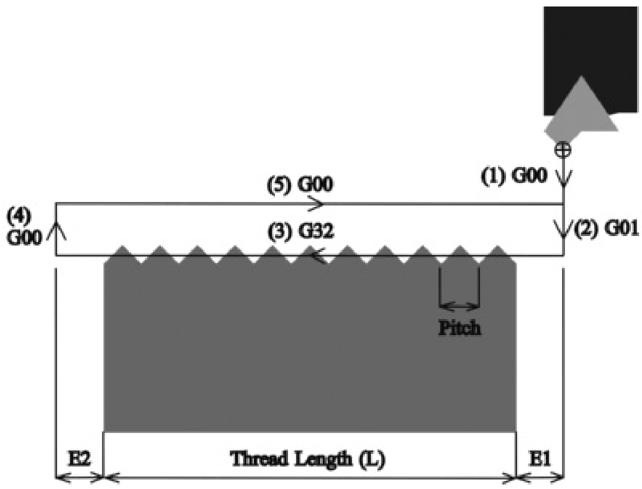


Figure 3.54: Tool path layout for external threading.

When obtaining an effective thread length, a minimum length of "Thread Length + $E_1 + E_2$" is required to prevent shorter pitches due to acceleration and deceleration of the cutting tool. To obtain E_1 and E_2, spindle speed and thread pitch is multiplied by using Eq. 3.2 and Eq. 3.3. Moreover the number of cutting times can be calculated by using Eq. 3.4 where NE is the number of cut.

$$E_1(mm) = 0.0015 \times N(rpm) \times L(mm) \tag{3.2}$$

$$E_2(mm) = 0.00042 \times N(rpm) \times L(mm) \tag{3.3}$$

$$NE = 3.3 \times P + 2.5 \tag{3.4}$$

3.8.8 Milling Planes

Code generation for milling operation differ for the plane to work on. Machining on facial plane consists of using Z-axis rotating tools for planar milling and drilling operations which includes hole, pocket, pocket-island, path and step machining. Z-axis is used for the feed direction together with X, Y and C-axis motions. Moreover G17 code have to be used prior to using circular interpolation commands such as G02 and G03. (Figure 3.55)

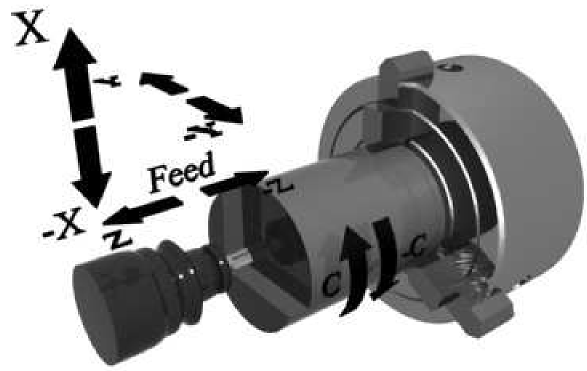


Figure 3.55: NC coordinate system for machining on facial plane.

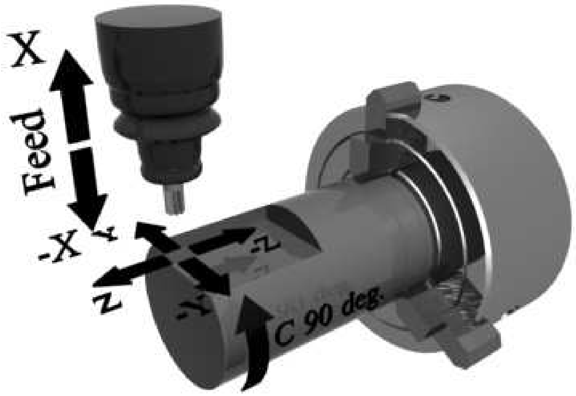


Figure 3.56: NC coordinate system for machining on lateral plane.

The same operations stated above can be handled on lateral plane by using X-axis rotating tools for feed direction and movement of Y, Z and C axes. G19 code have to be used prior to using circular interpolation commands such as G02 and G03 when machining on lateral and circumferential plane.(Figure 3.56)

Machining on circumferential plane consist of using X-axis rotating tools for circular milling and hole operations which includes pocket, pocket-island, path and hole machining with the use of X-axis for feed direction and Z, C-axis motions. (Figure 3.57)

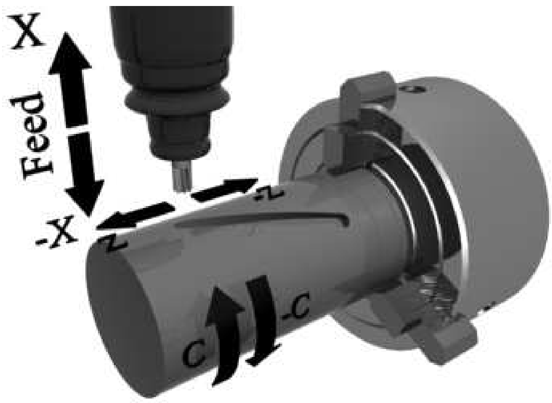


Figure 3.57: NC coordinate system for machining on circumferential plane.

3.8.9 Prism and Step Milling

Tool path generation is performed on facial and lateral surfaces of step feature with the use of G00 and G01 codes. The algorithm used for machining the external profile of prism feature is same as machining radially patterned step features with respect to the circumferential plane formed by Z-axis.

Starting from the diameter of the workpiece; the cutting tool feeds the facial surface with a given depth of cut and travels horizontally through the width value calculated from Equation 3.5 for each pass and vertically with the radius value of the cutting tool. (Figure 3.58) Equation 3.6 shows the calculation of Number of Pass (NP) value for facial plane where CTR represents cutting tool radius.

$$Width = \sqrt{Radius^2 + Height^2} \tag{3.5}$$

$$NP = \frac{Radius - Height}{CTR} \tag{3.6}$$

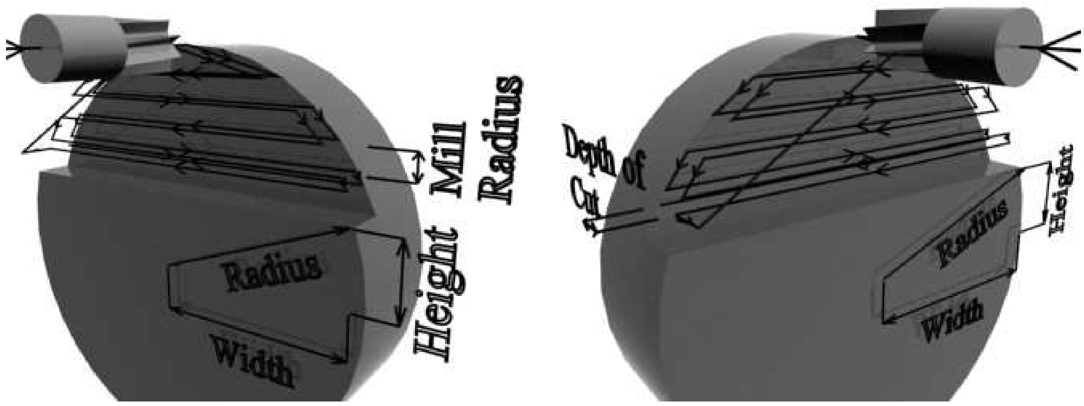


Figure 3.58: Tool path layout for facial step milling.

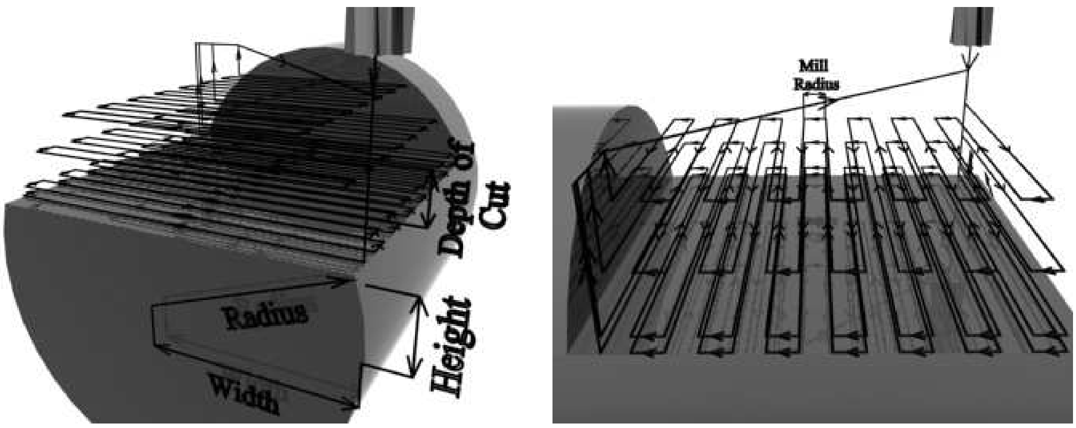


Figure 3.59: Tool path layout for prism and lateral step milling.

Starting from the diameter of the workpiece; the cutting tool feeds the lateral surface with a given depth of cut and travels horizontally through the width value for each depth of cut and vertically with the radius value of the cutting tool through the length of the feature. (Figure 3.59) Equation 3.7 shows the calculation of Number of Pass (NP) value for lateral plane where DOC represents depth of cut of each pass.

$$NP = \frac{Radius - Height}{DOC} \tag{3.7}$$

3.8.10 Pocket Milling

Tool path generation for pocket milling is performed for rectangular and circular profiles. G00 and G01 codes are used for generating tool path on facial, lateral and circumferential planes for rectangular pocket milling. Starting from the higher left corner; for each pass the cutting tool feeds the surface with a given depth of cut and travels horizontally through the length of the pocket and vertically with the radius value of the cutting tool through the width of the pocket. (Figure 3.60)

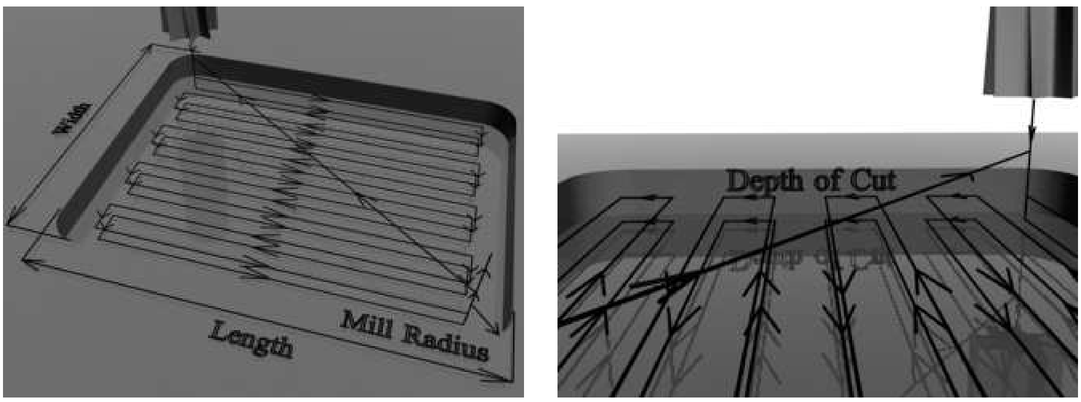


Figure 3.60: Tool path layout for rectangular pocket milling.

On the other hand; G00, G01, G02 codes are used for generating tool path on facial and lateral planes for circular pocket milling. Starting from the center of pocket; for each pass the cutting tool feeds the surface with a given depth of cut and travels horizontally with a value of cutting tool radius and circular interpolating through the circumference of circle formed by each horizontal movement of the tool through outwards of the pocket diameter. (Figure 3.61)

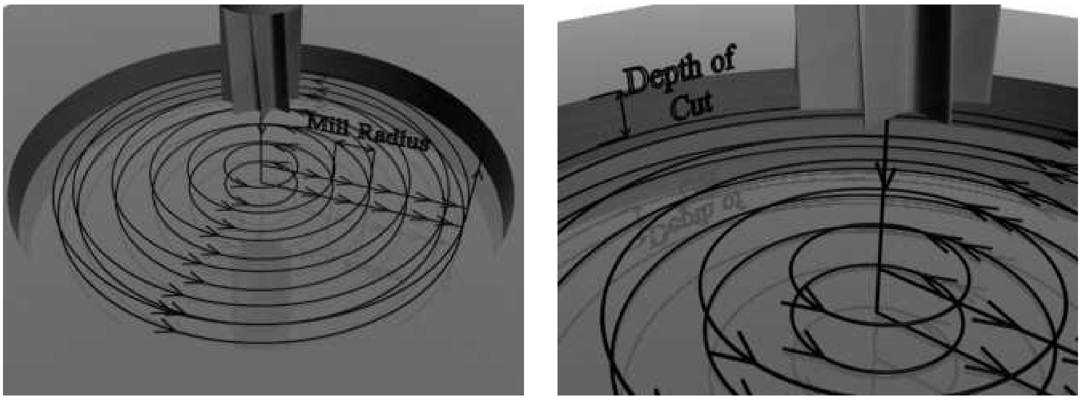


Figure 3.61: Tool path layout for circular pocket milling.

3.8.11 Path Milling

Tool path generation for path milling is performed for linear and circular profiles with the use of G00 and G01 or G02 codes shown in Figure 3.62.

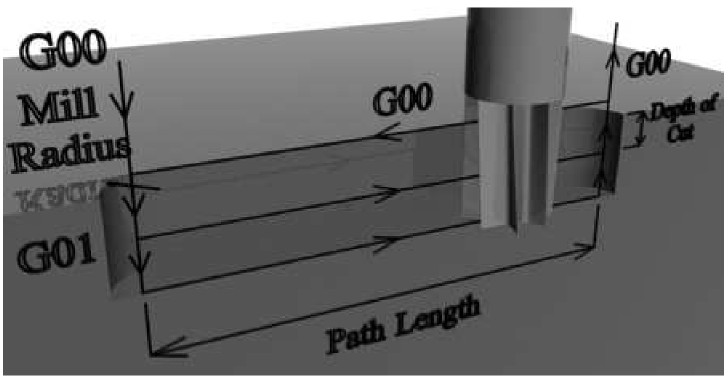


Figure 3.62: Tool path layout for path milling.

The NC code is generated by feeding the cutting tool to the start point with a given depth of cut and continued cutting through the end point of path. After the cutting process; the cutting tool rapidly release and position on start point in order to cut the next pass.

Principle of generating tool path for circular path is same as in linear path except G02 code has to be used with a predefined path radius value.

3.8.12 Hole Milling

Hole machining is combination of several operations which are; center-drilling, drilling, reaming, threading, counterboring and countersinking. A center-drilling operation have to be performed prior to the drilling operation to center the hole without an offset error and drilling operation must be done prior to reaming, threading or counterboring and countersinking.

For machines which are not capable of using canned cycles; G01 and G32 codes are used for linear interpolation and tapping process. On the other hand, G83, G84 and G85 canned cycles can be used for facial; G87, G88 and G89 canned cycles can be used for lateral machining process. (Figure 3.63)

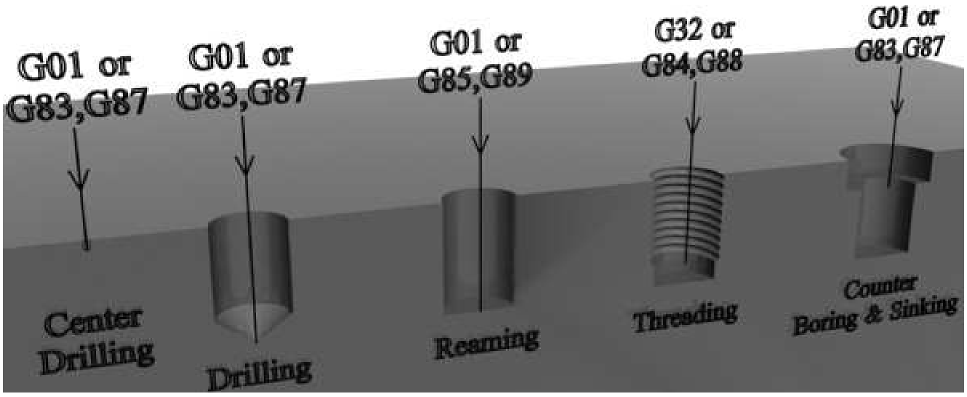


Figure 3.63: Tool path layout for hole milling.

CHAPTER 4

CASE STUDIES

This section contains three case studies shown in Figure 4.1 which are explained in detailed information. Each case study contains various sections starting with a setup information for the workpiece, blank geometry definition, part geometry definition and tool definition in order to visualize the whole design process step by step with the given technical drawings. It should be useful to look through the user manual of program given in Appendix A prior to study these tutorials.

Figure 4.1: Case Studies.

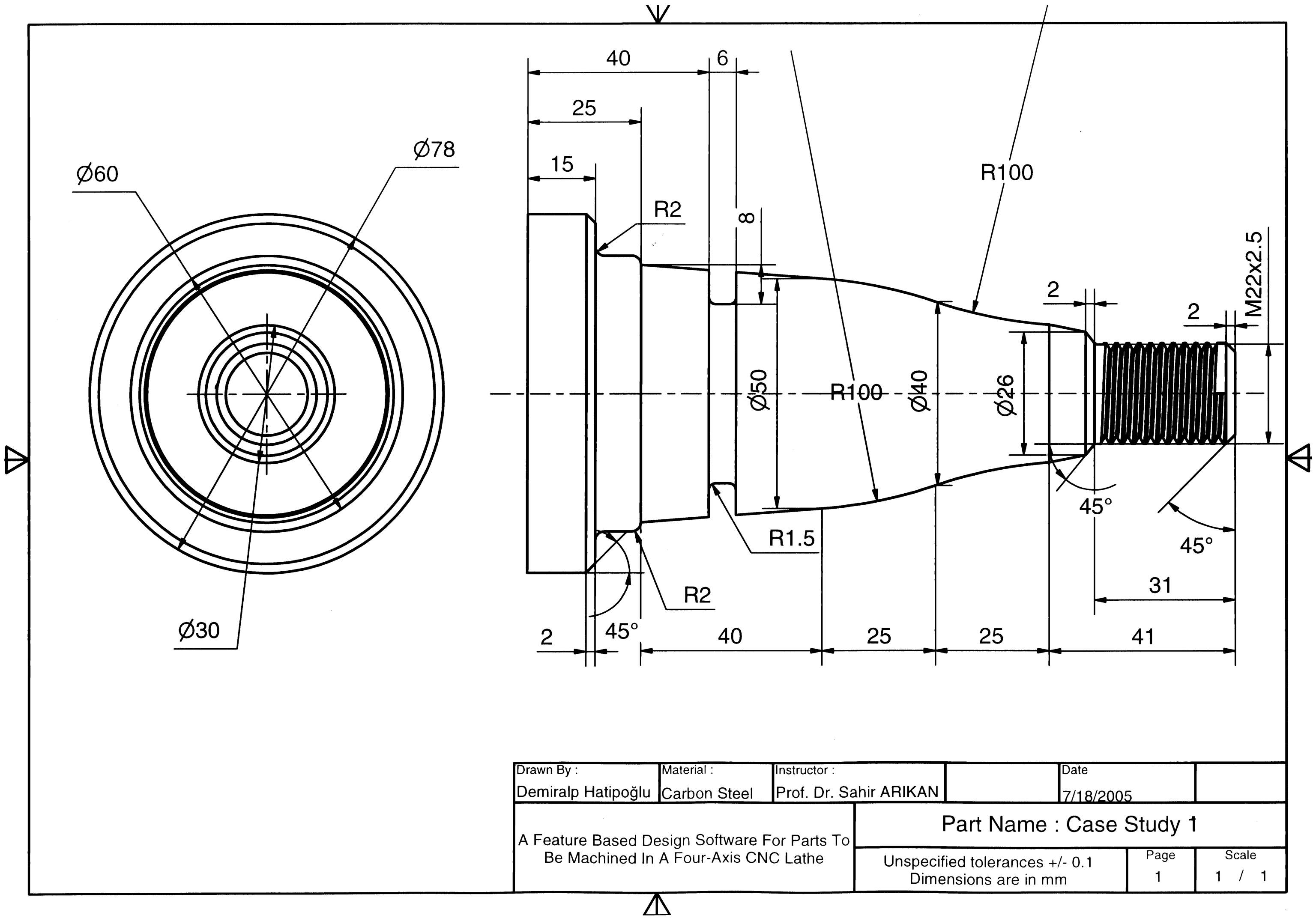


Figure 4.2: Drawing of Case Study 1.

4.1 Case Study I

This part demonstrates turning operations and composed of turning features which are cylinder, taper, convex, concave, thread and groove.(Figure 4.3) The detailed technical drawing of the part is given in Figure 4.2.

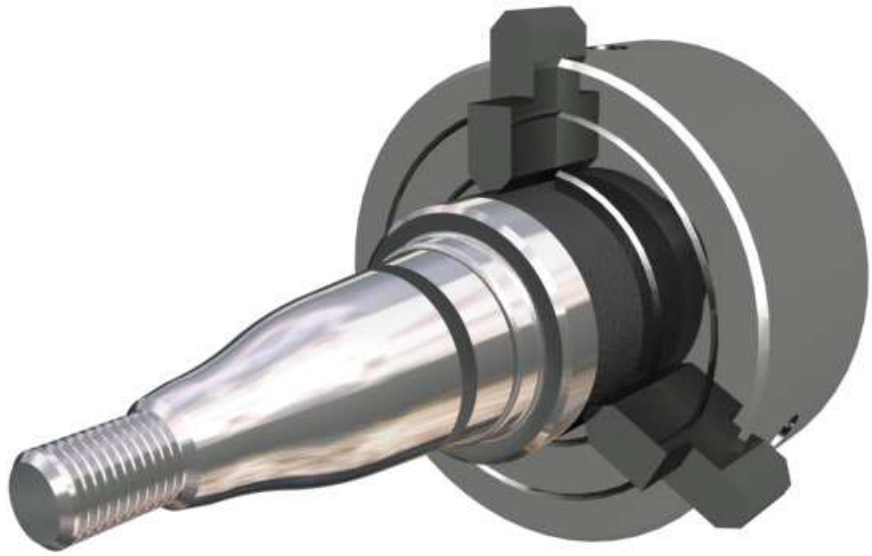


Figure 4.3: Case Study 1.

4.1.1 Setup Information

Blank Material :

Length : 160 mm , Diameter : 85 mm

Part Material :

Length : 156 mm , Diameter : 78 mm

Part Program Number : 0001

Workpiece Material : Carbon Steel

Reference Point Coordinates :

X = 100 mm , Y = 0 mm , Z = 15 mm

4.1.2 Blank Geometry Definition

Cylinder_Feature_1 , Outer Profile :

Diameter = 85 mm , Length = 160 mm

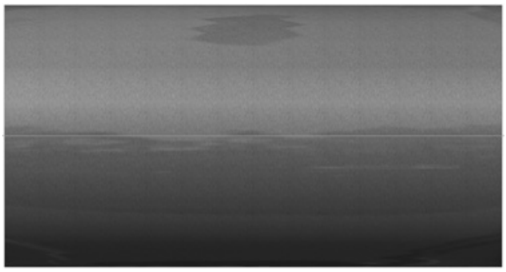


Figure 4.4: Cylinder_Feature_1

4.1.3 Part Geometry Definition

Cylinder_Feature_1 , Outer Profile :

Diameter = 78 mm , Length = 15 mm

Turret Side = Chamfer (Inwards) , Length1 = 2 mm , Length2 = 2mm

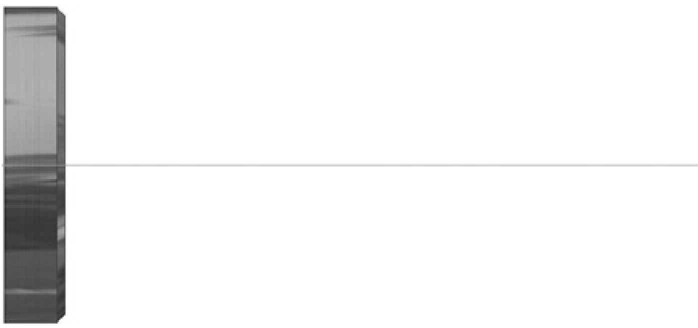


Figure 4.5: Cylinder_Feature_1

Cylinder_Feature_2 , Outer Profile :

Diameter = 60 mm , Length = 10 mm

Spindle Side = Fillet (Outwards) , Radius = 2 mm

Turret Side = Fillet (Inwards) , Radius = 2 mm

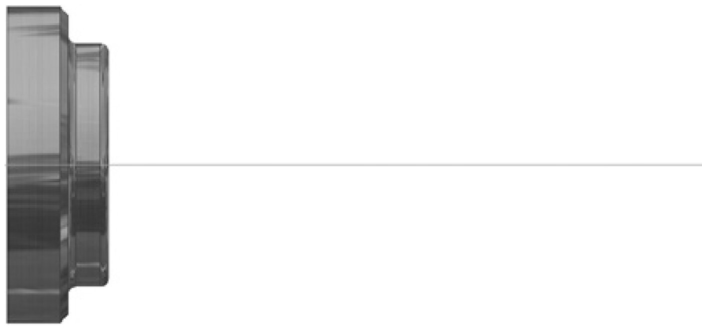


Figure 4.6: Cylinder_Feature_2

Taper_Feature_1 , Outer Profile :

Diameter1 = 56 mm , Diameter2 = 50 mm , Length = 40 mm

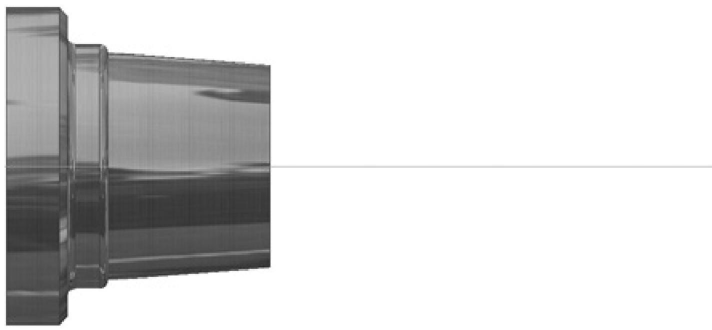


Figure 4.7: Taper_Feature_1

Convex_Feature_1 , Outer Profile :

Diameter1 = 50 mm , Diameter2 = 40 mm , Length = 25 mm , Radius = 100 mm

Concave_Feature_1 , Outer Profile :

Diameter1 = 40 mm , Diameter2 = 30 mm , Length = 25 mm , Radius = 100 mm

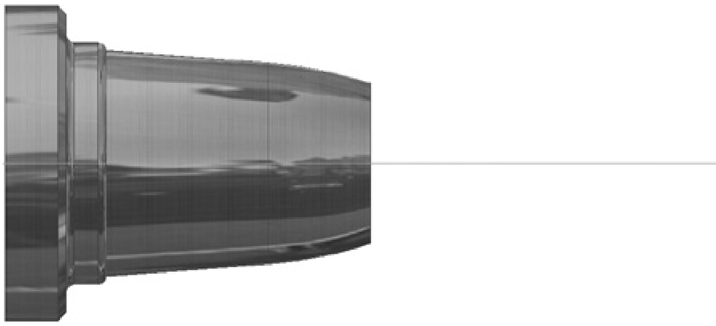


Figure 4.8: Convex_Feature_1

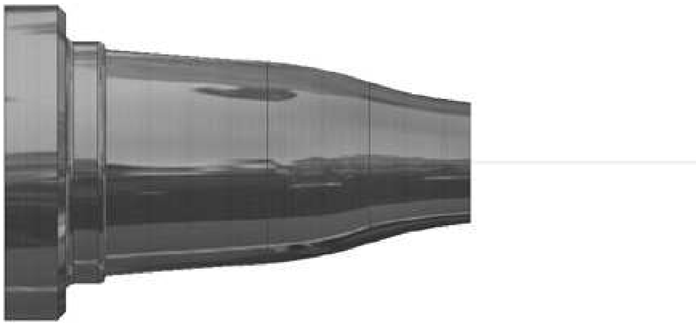


Figure 4.9: Concave_Feature_1

Taper_Feature_2 , Outer Profile :

Diameter1 = 30 mm , Diameter2 = 26 mm , Length = 10 mm

Turret Side = Chamfer (Inwards) , Length1 = 2 mm , Length2 = 2 mm

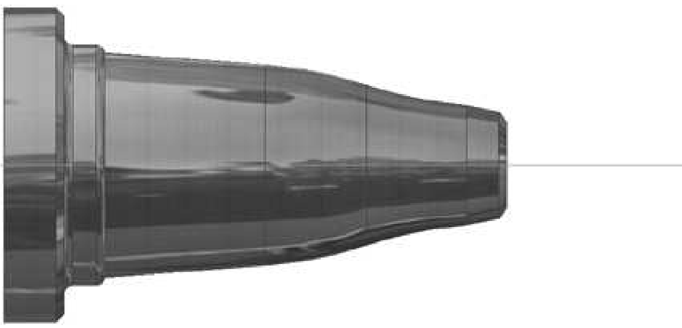


Figure 4.10: Taper_Feature_2

Thread_Feature_1 , Outer Profile :

Diameter = 22 mm , Length = 31 mm , Standart = Iso Metric , Type = M22 x 2.5

Turret Side = Chamfer (Inwards) , Length1 = 2 mm , Length2 = 2 mm

Right Handed = Selected

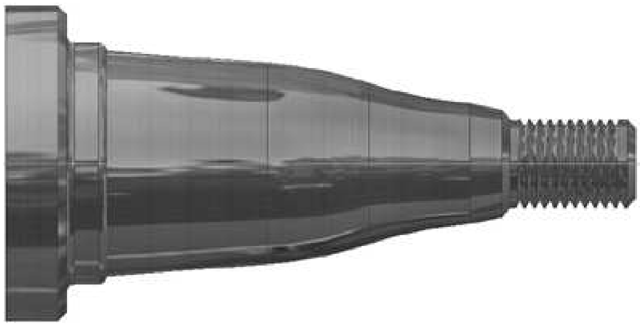


Figure 4.11: Thread_Feature_1

Groove_Feature_1 , Outer Profile :

Diameter = 55 mm , Length = 40 mm , Width = 6 mm , Depth = 8 mm

Groove Radius = 1.5 mm , Round Groove = Selected

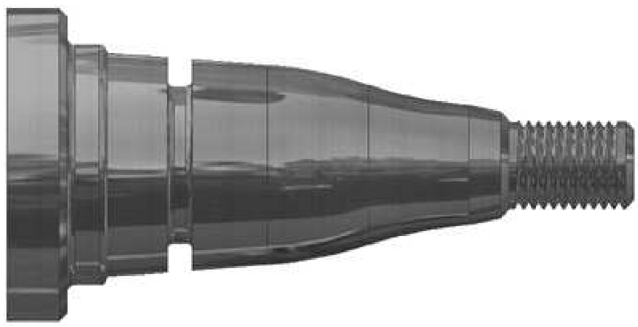


Figure 4.12: Groove_Feature_1

4.1.4 Tool Definition

Clearance values for finish cut :

Side = 1 mm , Face = 1 mm

Tool#1 :

Operation/Feature = Rough Turning , Plane = External

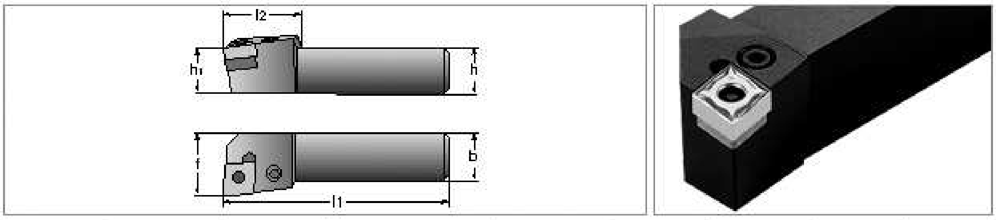
Cutting Tool = Left Hand Turning Tool , Manufacturer = ISCAR

Grip System = PCLNR/L , Tool Holder = 2020K-12 , Tool No = 1 , Turret No = 1

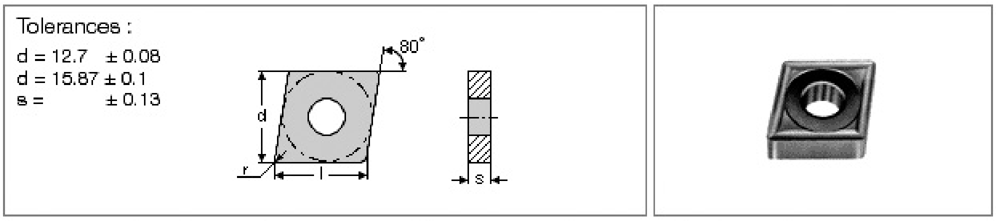
Insert = CNMG , Code = 160616 , Grade = IC9025

Application = Rough Machining , Type = Roughing

Feed Rate = 0.3 mm/rev , Cutting Speed = 150 m/min , Depth of Cut = 2 mm



h	h1	b	l1	l2	f	Ga°	Gr°	R/L
20.0	20.0	20.0	125.00	30.0	25.0	-6	-6	L



l	d	S	r
15.90	15.870	6.35	1.60

Figure 4.13: Holder and insert dimensions of Tool#1.

Tool#2 :

Operation/Feature = Finish Turning , Plane = External

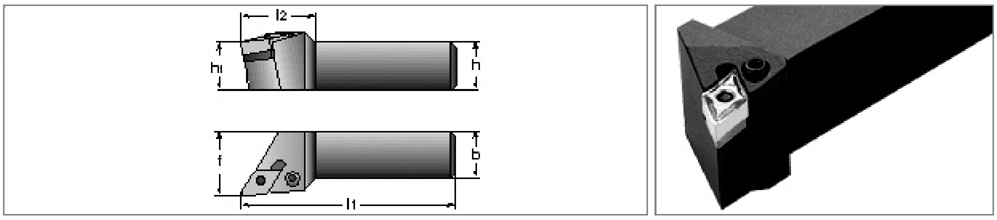
Cutting Tool = Left Hand Turning Tool , Manufacturer = ISCAR

Grip System = PDJNR/L , Tool Holder = 2020K-11 , Tool No = 2 , Turret No = 3

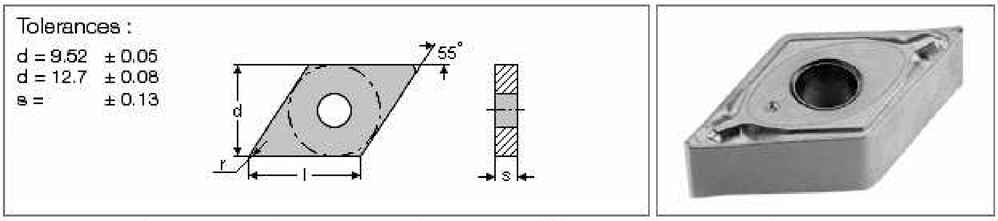
Insert = DNMG , Code = 110402-NF , Grade = IC9025

Application = Finish Machining , Type = Finishing

Feed Rate = 0.16 mm/rev , Cutting Speed = 250 m/min , Depth of Cut = 1 mm

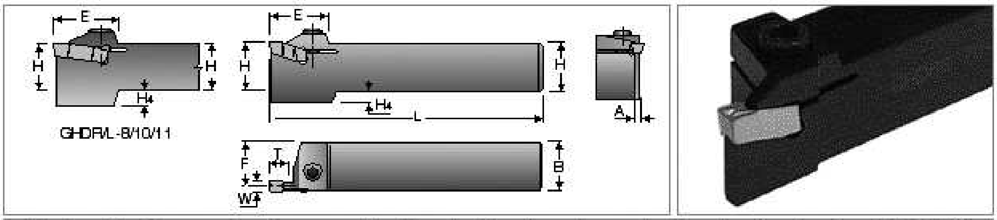


h	h1	b	l1	l2	f	Ga°	Gr°	R/L
20.0	20.0	20.0	125.00	30.0	25.0	-6	-7	L

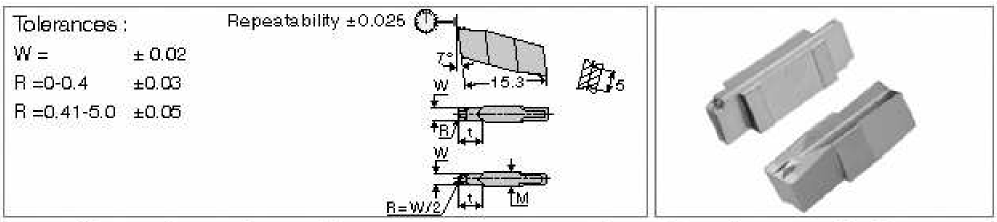


l	d	S	r
11.63	9.520	4.76	0.20

Figure 4.14: Holder and insert dimensions of Tool#2.



W min	W max	Tmax-r	H	B	L	F	A	E	H4	R/L
2.80	4.00	9.00	20.0	20.0	120.00	18.8	2.40	26.0	-	L



W	R	Tmax-r	M	D1 min
3.00	1.50	13.00	2.4	160.0

Figure 4.15: Holder and insert dimensions of Tool#3.

Tool#3 :

Operation/Feature = Groove_Feature_1 , Plane = External

Cutting Tool = Left Hand Grooving Tool , Manufacturer = ISCAR

Grip System = GHDR/L , Tool Holder = 20-3 , Tool No = 3 , Turret No = 5

Insert = GIP , Code = 3.00E-1.50 , Grade = IC635

Application = External Turning and Grooving , Type = Grooving

Feed Rate = 0.085 mm/rev , Cutting Speed = 70 m/min , Depth of Cut = 1 mm

Tool#4 :

Operation/Feature = Thread_Feature_1 , Plane = External

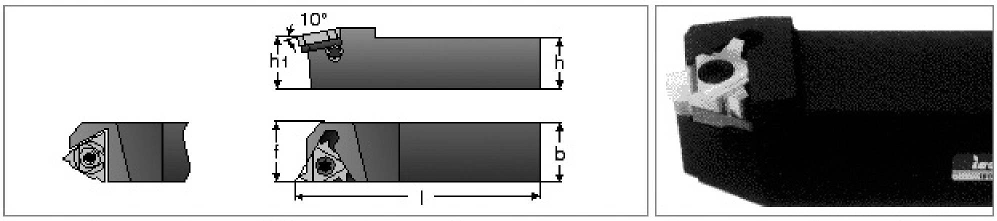
Cutting Tool = Left Hand Threading Tool , Manufacturer = ISCAR

Grip System = SER/L , Tool Holder = 2020-K16 , Tool No = 4 , Turret No = 7

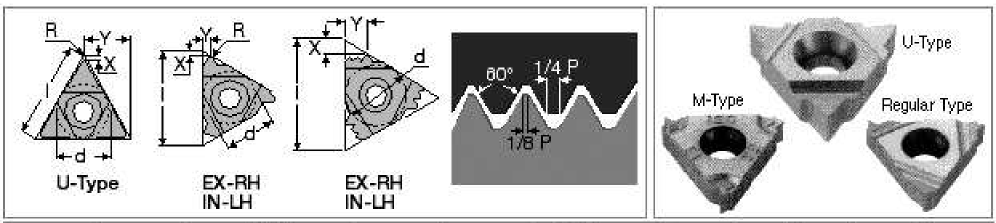
Insert = 16ER/L , Code = 2.5 ISO , Grade = IC228

Application = Iso Metric Full Profile , Type = External Threading

Feed Rate = 2.5 mm/rev , Cutting Speed = 94.5 m/min , Depth of Cut = 0.53 mm



h	h1	b	L	f
20.0	20.0	20.0	125.00	20.0



d	Pitch	l	Z	R	X	Y
9.525	2.50	16.00	1	0.31	1.1	1.50

Figure 4.16: Holder and insert dimensions of Tool#4.

Note : Illustrations for the design process are only given for *Case Study I* therefore parameters for part features in the remaining studies must be entered in a row same as stated in text. In addition to that cutting tool dimensions can be achieved from the web site [3] for the following studies.

4.2 Case Study II

Part shown in Figure 4.17 demonstrates turning operations on external and internal profiles and contains milling operation involving machining on lateral plane. Cylinder, taper, concave, prism, thread, groove and step features are used. (Figure 4.19) The detailed technical drawing of the part is given in Figure 4.18.

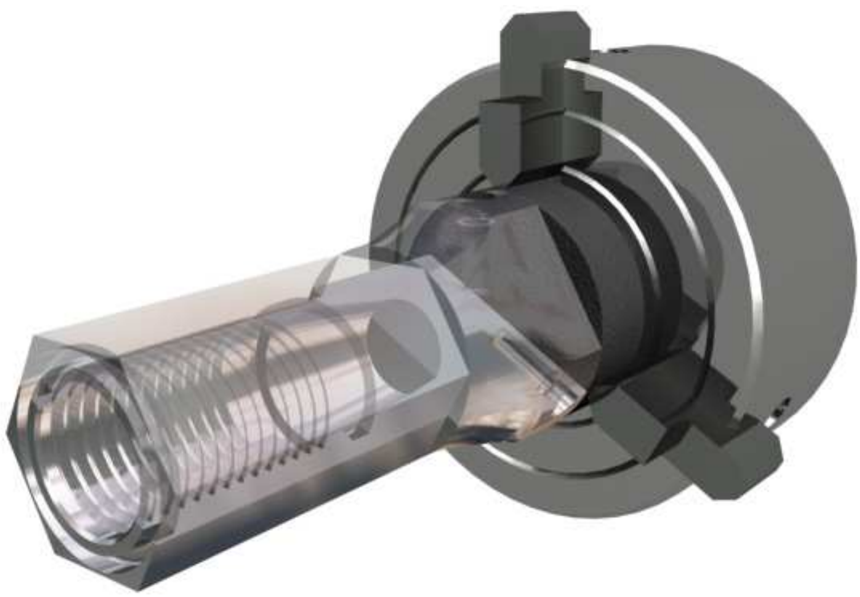


Figure 4.17: Case Study 2.

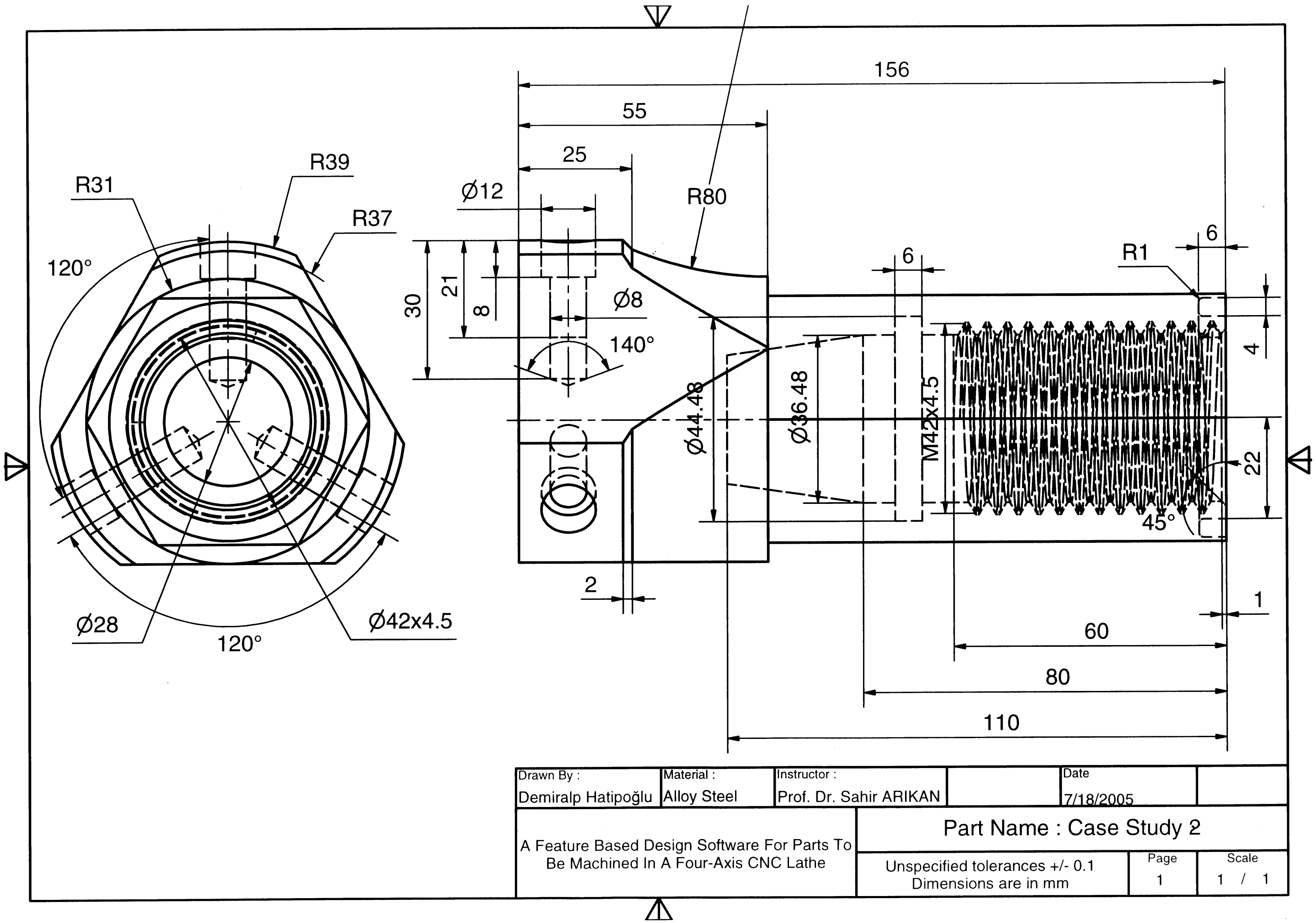


Figure 4.18: Drawing of Case Study 2.

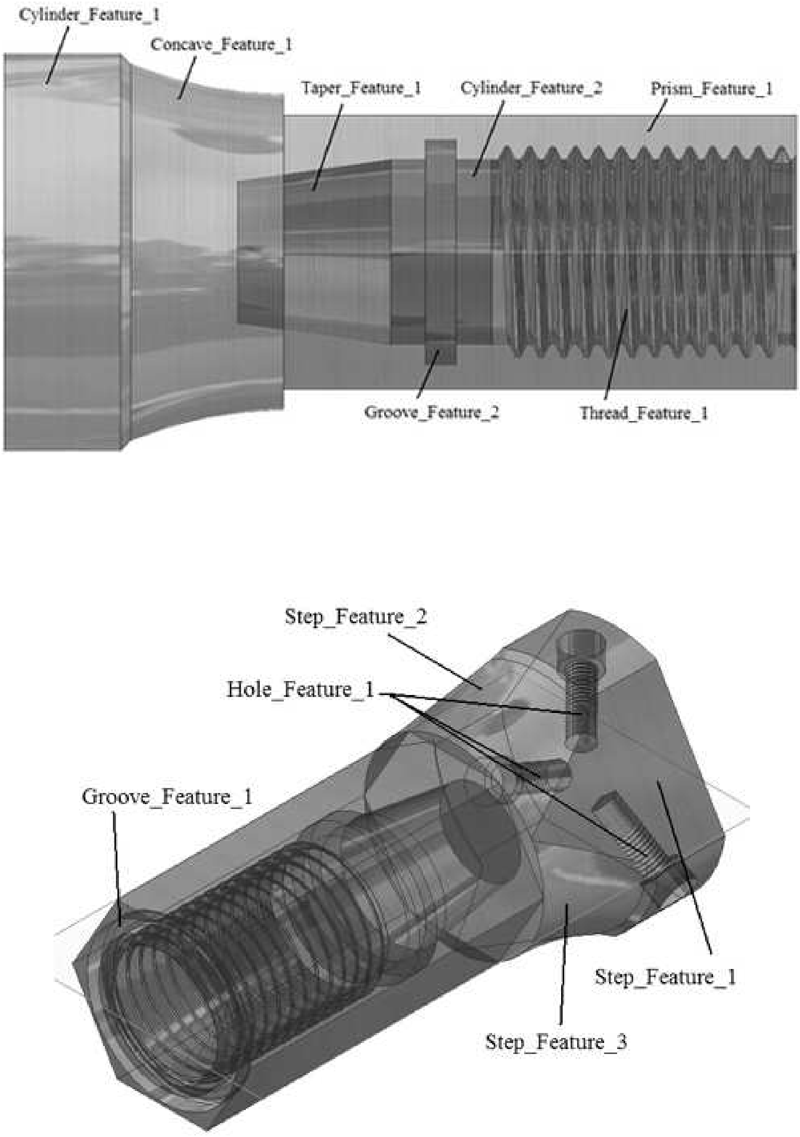


Figure 4.19: Features used for Case Study 2.

4.2.1 Setup Information

Blank Material :

Length : 160 mm , Diameter : 85 mm

Part Material :

Length : 156 mm , Diameter : 78 mm

Part Program Number : 0002

Workpiece Material : Alloy Steel

Reference Point Coordinates :

X = 100 mm , Y = 0 mm , Z = 15 mm

4.2.2 Blank Geometry Definition

Cylinder_Feature_1 , Outer Profile :

Diameter = 85 mm , Length = 160 mm

4.2.3 Part Geometry Definition

Cylinder_Feature_1 , Outer Profile :

Diameter = 78 mm , Length = 25 mm

Turret Side = Chamfer (Inwards) , Length1 = 2 mm , Length2 = 2mm

Concave_Feature_1 , Outer Profile :

Diameter1 = 74 mm , Diameter2 = 62 mm , Length = 30 mm , Radius = 80 mm

Prism_Feature_1 , Outer Profile :

Diameter = 62 mm , Length = 101 mm , Number of Sides = 6

Rotation Angle = 0 deg , Inscribed in Circle = Selected

Groove_Feature_1 , Outer Profile :

Radius = 22 mm , Width = 4 mm , Depth = 6 mm

Groove Radius = 1 mm , Round Groove = Selected , Face Groove = Selected

Thread_Feature_1 , Inner Profile :

Diameter = 42 mm , Length = 60 mm , Standard = Iso Metric , Type = M42 x 4.5

Turret Side = Chamfer (Outwards) , Length1 = 1 mm , Length2 = 1 mm

Right Handed = Selected

Cylinder_Feature_2 , Inner Profile :

Diameter = 36.48 mm , Length = 20 mm

Taper_Feature_1 , Inner Profile :

Diameter1 = 28 mm , Diameter2 = 36.48 mm , Length = 30 mm

Groove_Feature_2 , Inner Profile :

Diameter = 36.48 mm , Length = 83 mm , Width = 6 mm , Depth = 4 mm

Rectangular Groove = Selected

Step_Feature_1 , Lateral Plane :

CenterX = 39 mm , AngleXZ = 30 deg , Length = 55 mm , Depth = 8 mm

Spindle Dist = 27.5 mm

Step_Feature_2 , Lateral Plane :

CenterX = 39 mm , AngleXZ = 150 deg , Length = 55 mm , Depth = 8 mm

Spindle Dist = 27.5 mm

Step_Feature_3 , Lateral Plane :

CenterX = 39 mm , AngleXZ = 270 deg , Length = 55 mm , Depth = 8 mm

Spindle Dist = 27.5 mm

Hole_Feature_1 , Circumferential Plane :

Hole Diameter = 8 mm , Hole Depth = 30 mm , Hole Pattern = Radial

Hole Style = Counterbore , Bore Diameter = 12 mm , Bore Depth = 8 mm

Thread Diameter = M8 , Thread Depth = 21 mm

Bottom Style = Angled , Tip Angle = 140 deg

CenterX = 39 mm , AngleXZ = 90 deg , Distance to Spindle Side = 11 mm

Angle to Fill = 240 deg , Number of Holes = 3 , Number of Rows = 1

4.2.4 Tool Definition

Clearance values for finish cut :

Side = 1 mm , Face = 1 mm

Tool#1 :

Operation/Feature = Rough Turning , Plane = External

Cutting Tool = Left Hand Turning Tool , Manufacturer = ISCAR

Grip System = PCLNR/L , Tool Holder = 2020K-12 , Tool No = 1 , Turret No = 1

Insert = CNMM , Code = 120408-NM , Grade = IC9015

Application = Rough Machining , Type = Interrupted

Feed Rate = 0.375 mm/rev , Cutting Speed = 250 m/min , Depth of Cut = 2 mm

Tool#2 :

Operation/Feature = Finish Turning , Plane = External

Cutting Tool = Left Hand Turning Tool , Manufacturer = ISCAR

Grip System = PDJNR/L , Tool Holder = 2020K-11 , Tool No = 2 , Turret No = 3

Insert = DNMG , Code = 110402-NF , Grade = IC9015

Application = Finish Machining , Type = Finishing

Feed Rate = 0.16 mm/rev , Cutting Speed = 325 m/min , Depth of Cut = 1 mm

Tool#3 :

Operation/Feature = Rough Turning , Plane = Internal

Cutting Tool = XY_Plane_Drill , Manufacturer = ISCAR

Grip System = DZ , Tool Holder = 028-112-32-05 , Tool No = 3 , Turret No = 2

Insert = WOLH , Code = 05T304-GF , Grade = IC908

Application = Drilling , Type = N/A

Feed Rate = 0.07 mm/rev , Cutting Speed = 185 m/min , Depth of Cut = 0 mm

Tool#4 :

Operation/Feature = Rough Turning , Plane = Internal

Cutting Tool = Left Hand Boring Bar , Manufacturer = ISCAR

Grip System = A/S-PDUNR/L , Tool Holder = A20Q-11 , Tool No = 4

Turret No = 5 , Insert = DNMM , Code = 150612-NM , Grade = IC9025

Application = Rough Machining , Type = Interrupted

Feed Rate = 0.325 mm/rev , Cutting Speed = 200 m/min , Depth of Cut = 2 mm

Tool#5 :

Operation/Feature = Finish Turning , Plane = Internal

Cutting Tool = Left Hand Boring Bar , Manufacturer = ISCAR

Grip System = A/S-PDUNR/L , Tool Holder = A20Q-11 , Tool No = 5

Turret No = 7 , Insert = DNMG , Code = 110402-NF , Grade = IC9015

Application = Finish Machining , Type = Finishing

Feed Rate = 0.16 mm/rev , Cutting Speed = 325 m/min , Depth of Cut = 1 mm

Tool#6 :

Operation/Feature = Groove_Feature_1 , Plane = External

Cutting Tool = Left Hand Grooving Tool , Manufacturer = ISCAR

Grip System = HFHR/L , Tool Holder = 25-48-4T25 , Tool No = 6 , Turret No = 1

Insert = HFPR/L , Code = 4004 , Grade = IC9025

Application = Face Grooving , Type = Grooving

Feed Rate = 0.08 mm/rev , Cutting Speed = 92.5 m/min , Depth of Cut = 1.5 mm

Tool#7 :

Operation/Feature = Groove_Feature_2 , Plane = Internal

Cutting Tool = Left Hand Grooving Bar , Manufacturer = ISCAR

Grip System = GHIR/L , Tool Holder = 20-4 , Tool No = 7 , Turret No = 3

Insert = GIPI , Code = 3.00E-0.40 , Grade = IC354

Application = Internal Turning and Grooving , Type = Grooving

Feed Rate = 0.0925 mm/rev , Cutting Speed = 70 m/min , Depth of Cut = 1.2 mm

Tool#8 :

Operation/Feature = Thread_Feature_1 , Plane = Internal

Cutting Tool = Left Hand Threading Bar , Manufacturer = ISCAR

Grip System = SIR/L , Tool Holder = 0025-R22 , Tool No = 8 , Turret No = 5

Insert = 22IR/L , Code = 4.5 ISO , Grade = IC228

Application = Iso Metric Full Profile , Type = Internal Threading

Feed Rate = 4.5 mm/rev , Cutting Speed = 62 m/min , Depth of Cut = 0.53 mm

Tool#9 :

Operation/Feature = Prism_Feature_1 , Plane = External

Cutting Tool = YZ_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-4 , Tool Holder = 100A22-4C10 , Tool No = 9

Turret No = 4 , Insert = 100A22-4C10 , Code = N/A , Grade = IC300

Application = Shouldering , Type = Roughing , Feed Rate = 0.1695 mm/rev

Cutting Speed = 137.5 m/min , Depth of Cut = 2.5 mm

Tool#10,11,12 :

Operation/Feature = Step_Feature_1,2,3 , Plane = Lateral

Cutting Tool = YZ_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-4 , Tool Holder = 100A22-4C10 , Tool No = 9

Turret No = 4 , Insert = 100A22-4C10 , Code = N/A , Grade = IC300

Application = Shouldering , Type = Roughing , Feed Rate = 0.1695 mm/rev

Cutting Speed = 137.5 m/min , Depth of Cut = 2.5 mm

Tool#13 :

Operation/Feature = Hole_Feature_1 , Plane = Circumferential

Cutting Tool = YZ_Plane_Drill , Manufacturer = ISCAR

Grip System = SCD-AG5 , Tool Holder = 080-043-080-AG5 , Tool No = 10

Turret No = 6 , Insert = 080-043-080-AG5 , Code = N/A , Grade = IC908

Application = Drilling , Type = N/A

Feed Rate = 0.2 mm/rev , Cutting Speed = 105 m/min , Depth of Cut = 0 mm

Tool#14 :

Operation/Feature = Hole_Feature_1 , Plane = Circumferential

Cutting Tool = YZ_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-3 , Tool Holder = 120A25-3C12 , Tool No = 11

Turret No = 8 , Insert = 120A25-3C12 , Code = N/A , Grade = IC300

Application = Slotting , Type = Roughing , Feed Rate = 0.1745 mm/rev

Cutting Speed = 137.5 m/min , Depth of Cut = 4 mm

Tool#15 :

Operation/Feature = Hole_Feature_1 , Plane = Circumferential

Cutting Tool = YZ_Plane_Tap , Manufacturer = ISCAR

Grip System = TP5-6H-CCN-HEV , Tool Holder = M8x1.25 , Tool No = 12

Turret No = 10 , Insert = TP5-6H-CCN-HEV , Code = M8x1.25 , Grade = H.S.S.

Application = Deep Blind Hole Tapping , Type = N/A

Feed Rate = 1.25 mm/rev , Cutting Speed = 6 m/min , Depth of Cut = 0 mm

4.3 Case Study III

Part shown in Figure 4.20 demonstrates advanced milling operations on facial, lateral and circumferential planes. Cylinder, taper, step, hole, path and pocket features are used. (Figure 4.23) The detailed technical drawings of the part are given in Figures 4.21 and 4.22.

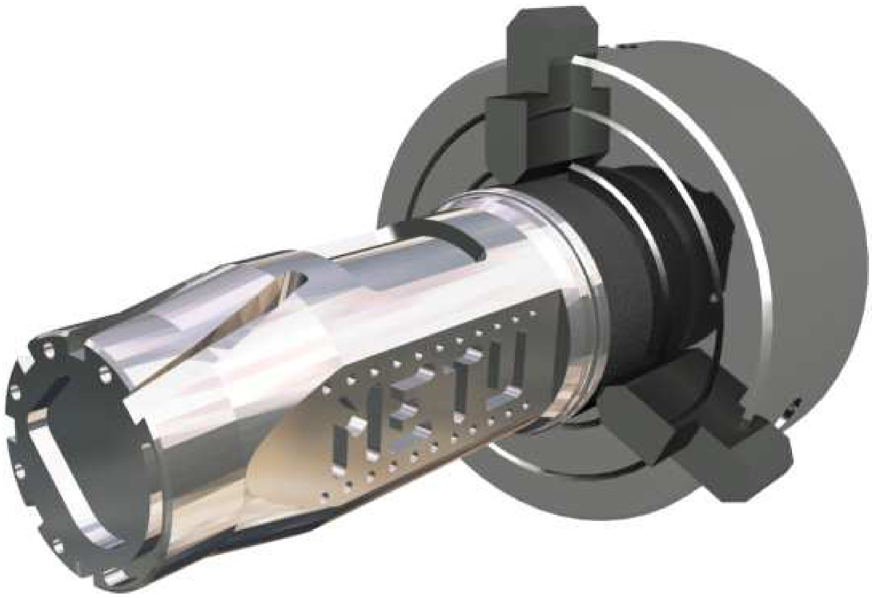


Figure 4.20: Case Study 3.

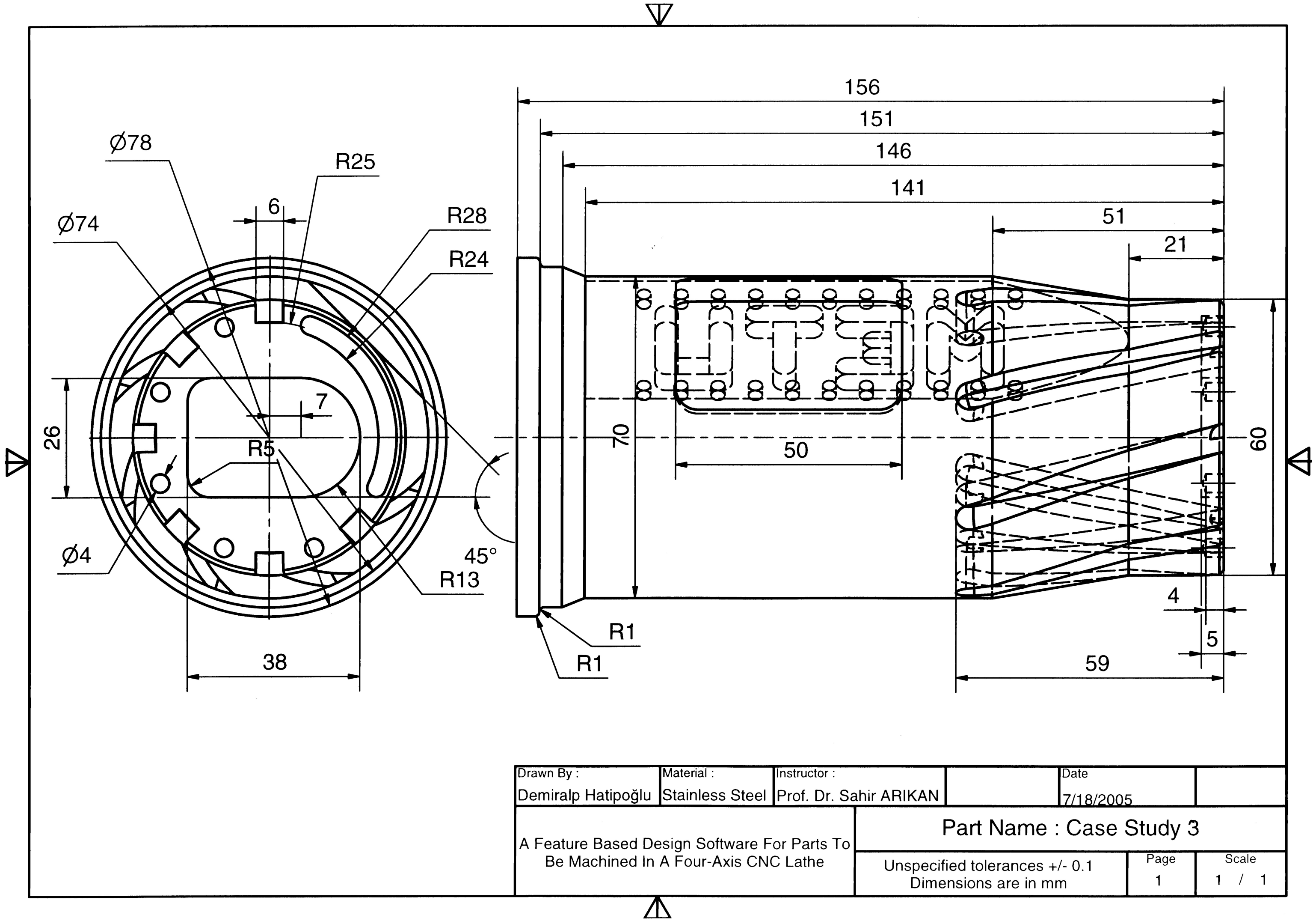


Figure 4.21: Drawing of Case Study 3.

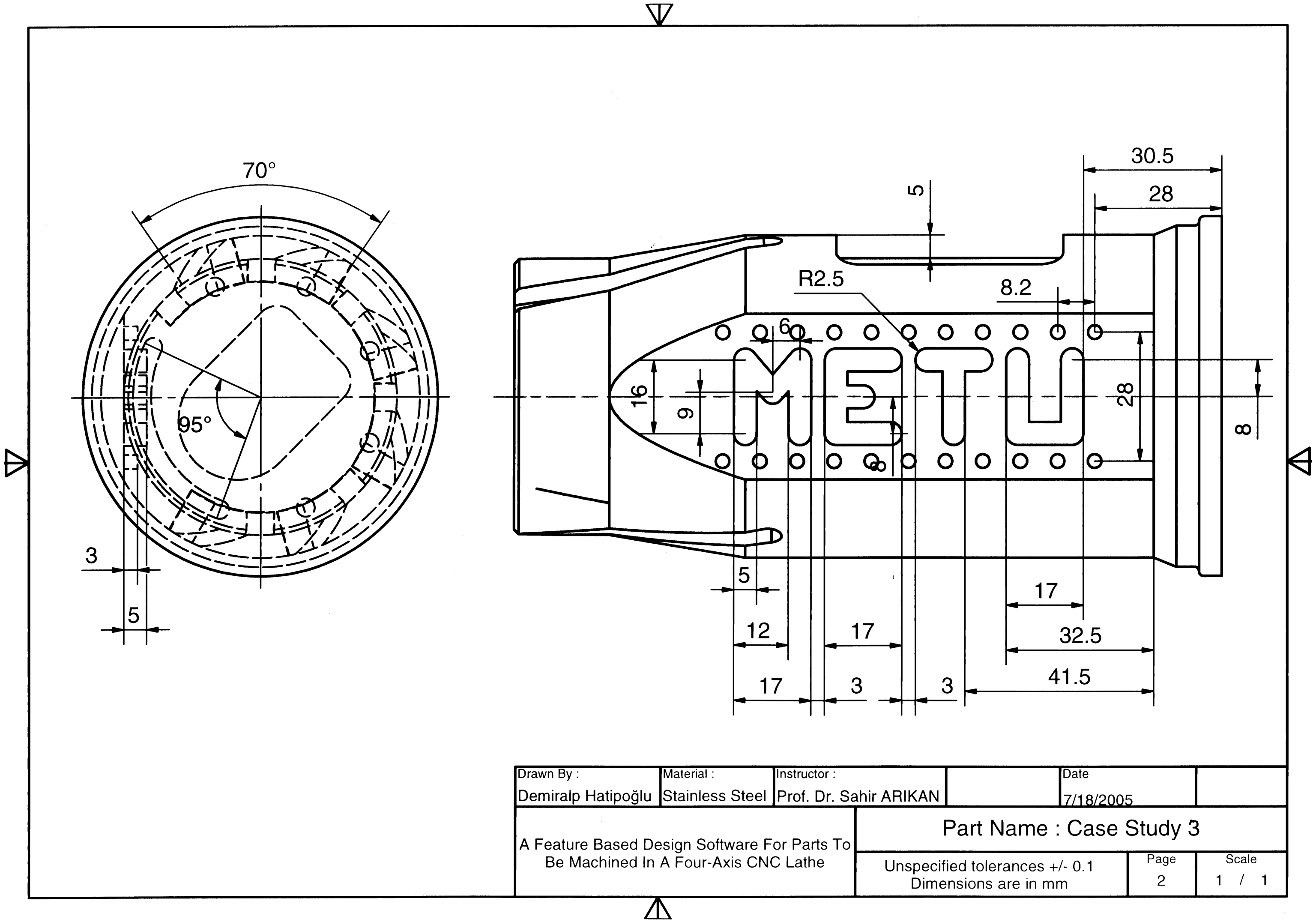
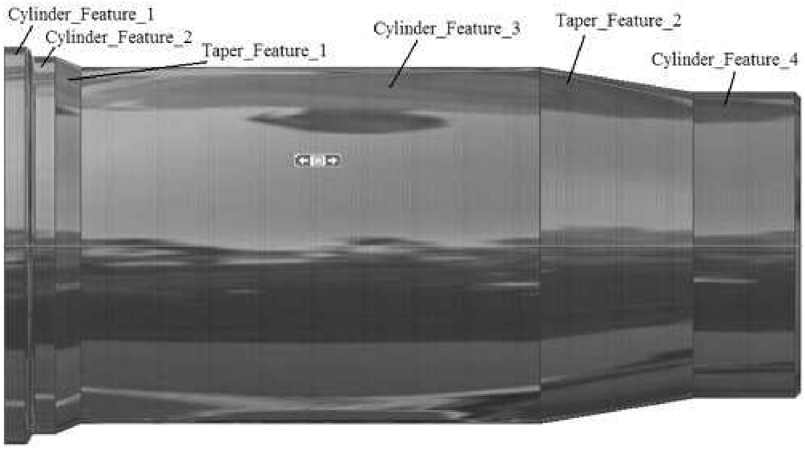


Figure 4.22: (continued) Drawing of Case Study 3.



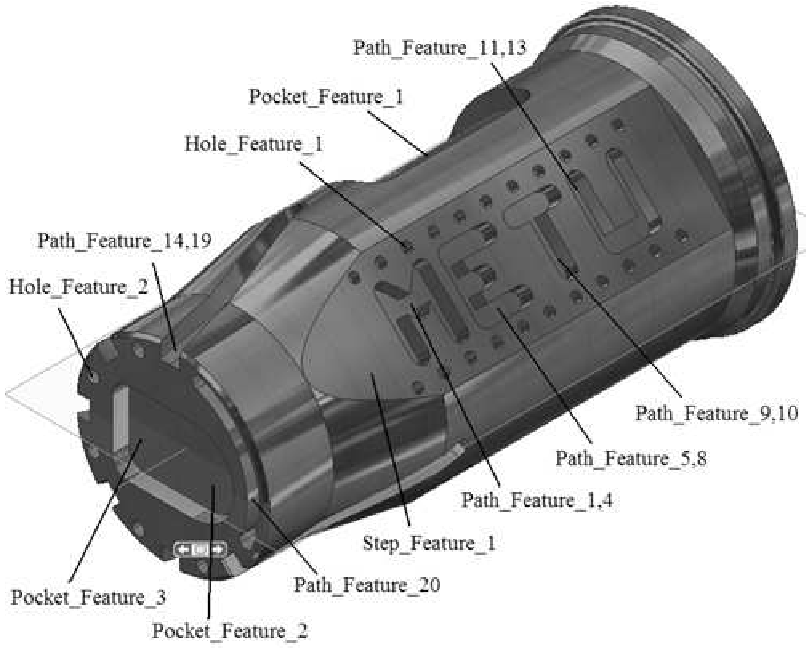


Figure 4.23: Features used for Case Study 3.

4.3.1 Setup Information

Blank Material :

Length : 160 mm , Diameter : 85 mm

Part Material :

Length : 156 mm , Diameter : 78 mm

Part Program Number : 0003

Workpiece Material : Stainless Steel

Reference Point Coordinates :

X = 100 mm , Y = 0 mm , Z = 15 mm

4.3.2 Blank Geometry Definition

Cylinder_Feature_1 , Outer Profile :

Diameter = 85 mm , Length = 160 mm

4.3.3 Part Geometry Definition

Cylinder_Feature_1 , Outer Profile :

Diameter = 78 mm , Length = 5 mm

Turret Side = Fillet (Inwards) , Length1 = 1 mm , Length2 = 1mm

Cylinder_Feature_2 , Outer Profile :

Diameter = 74 mm , Length = 5 mm

Spindle Side = Fillet (Outwards) , Length1 = 1 mm , Length2 = 1mm

Taper_Feature_1 , Outer Profile :

Diameter1 = 74 mm , Diameter2 = 70 mm , Length = 5 mm

Cylinder_Feature_3 , Outer Profile :

Diameter = 70 mm , Length = 90 mm

Taper_Feature_2 , Outer Profile :

Diameter1 = 70 mm , Diameter2 = 60 mm , Length = 30 mm

Cylinder_Feature_4 , Outer Profile :

Diameter = 60 mm , Length = 21 mm

Turret Side = Chamfer (Inwards) , Length1 = 1 mm , Length2 = 1mm

Step_Feature_1 , Lateral Plane :

CenterX = 35 mm , AngleXZ = 45 deg , Length = 120 mm , Depth = 5 mm

Spindle Dist = 75 mm

Path_Feature_1 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = -8 mm , EndY = 8 mm

StartZ = 105 mm , EndZ = 105 mm , Angle to XZ = 45 deg

Path_Feature_2 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = 8 mm , EndY = 1 mm

StartZ = 105 mm , EndZ = 99 mm , Angle to XZ = 45 deg

Path_Feature_3 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = 1 mm , EndY = 8 mm

StartZ = 99 mm , EndZ = 93 mm , Angle to XZ = 45 deg

Path_Feature_4 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = 8 mm , EndY = -8 mm

StartZ = 93 mm , EndZ = 93 mm , Angle to XZ = 45 deg

Path_Feature_5 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = -8 mm , EndY = 8 mm

StartZ = 85 mm , EndZ = 85 mm , Angle to XZ = 45 deg

Path_Feature_6 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = 8 mm , EndY = 8 mm

StartZ = 85 mm , EndZ = 73 mm , Angle to XZ = 45 deg

Path_Feature_7 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = 0 mm , EndY = 0 mm

StartZ = 85 mm , EndZ = 79 mm , Angle to XZ = 45 deg

Path_Feature_8 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = -8 mm , EndY = -8 mm

StartZ = 85 mm , EndZ = 73 mm , Angle to XZ = 45 deg

Path_Feature_9 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = 8 mm , EndY = 8 mm

StartZ = 65 mm , EndZ = 53 mm , Angle to XZ = 45 deg

Path_Feature_10 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = 8 mm , EndY = -8 mm

StartZ = 59 mm , EndZ = 59 mm , Angle to XZ = 45 deg

Path_Feature_11 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = 8 mm , EndY = -8 mm

StartZ = 45 mm , EndZ = 45 mm , Angle to XZ = 45 deg

Path_Feature_12 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = -8 mm , EndY = -8 mm

StartZ = 45 mm , EndZ = 33 mm , Angle to XZ = 45 deg

Path_Feature_13 , Lateral Plane :

Groove Profile = Straight , Width = 5 mm , Depth = 5 mm

Path Type = Linear , CenterX = 30 mm , StartY = -8 mm , EndY = 8 mm

StartZ = 33 mm , EndZ = 33 mm , Angle to XZ = 45 deg

Hole_Feature_1 , Lateral Plane :

Hole Diameter = 3 mm , Hole Depth = 3 mm , Hole Pattern = Rectangular

Hole Style = Plain Hole , Bottom Style = Flat Bottom

CenterX = 30 mm , CenterY = 14 mm , Distance to Spindle Side = 110 mm

Angle to XZ plane = 45 deg , Start Angle = -90 deg

Number of Rows = 11 , Number of Columns = 2

Distance Between Rows = 8.2 , Distance Between Columns = 28

Path_Feature_14 , Circumferential Plane :

Groove Profile = Straight , Width = 6 mm , Depth = 10 mm

Path Type = Linear , CenterX = 35 mm , StartZ = 156 mm , EndZ = 100 mm

Start Angle = 90 deg , End Angle = 120 deg

Path_Feature_15 , Circumferential Plane :

Groove Profile = Straight , Width = 6 mm , Depth = 10 mm

Path Type = Linear , CenterX = 35 mm , StartZ = 156 mm , EndZ = 100 mm

Start Angle = 135 deg , End Angle = 165 deg

Path_Feature_16 , Circumferential Plane :

Groove Profile = Straight , Width = 6 mm , Depth = 10 mm

Path Type = Linear , CenterX = 35 mm , StartZ = 156 mm , EndZ = 100 mm

Start Angle = 180 deg , End Angle = 210 deg

Path_Feature_17 , Circumferential Plane :

Groove Profile = Straight , Width = 6 mm , Depth = 10 mm

Path Type = Linear , CenterX = 35 mm , StartZ = 156 mm , EndZ = 100 mm

Start Angle = 225 deg , End Angle = 255 deg

Path_Feature_18 , Circumferential Plane :

Groove Profile = Straight , Width = 6 mm , Depth = 10 mm

Path Type = Linear , CenterX = 35 mm , StartZ = 156 mm , EndZ = 100 mm

Start Angle = 270 deg , End Angle = 300 deg

Path_Feature_19 , Circumferential Plane :

Groove Profile = Straight , Width = 6 mm , Depth = 10 mm

Path Type = Linear , CenterX = 35 mm , StartZ = 156 mm , EndZ = 100 mm

Start Angle = 315 deg , End Angle = 345 deg

Pocket_Feature_1 , Circumferential Plane :

Pocket Type = Rectangular , Distance = 30 mm , Included Angle = 70 deg

Spindle Distance = 60 mm , Radius = 5 mm , Depth = 5 mm , Length = 50 mm

Angle to XZ plane = 100 deg

Hole_Feature_2 , Facial Plane :

Hole Diameter = 4 mm , Hole Depth = 4 mm , Hole Pattern = Radial

Hole Style = Plain Hole , Bottom Style = Angled Bottom , Tip Angle = 140 deg

CenterX = 0 mm , CenterY = 0 mm , Distance to Spindle Side = 156 mm

Start Angle = 112.5 deg , Pitch Diameter = 26 mm , Angle to Fill = 225 deg

Number of Holes = 5

Pocket_Feature_2 , Facial Plane :

Pocket Type = Circular , OffsetX = 7 mm , OffsetY = 0 mm

Spindle Distance = 156 mm , Diameter = 26 mm , Depth = 5 mm

Pocket_Feature_3 , Facial Plane :

Pocket Type = Rectangular , OffsetX = -3 mm , OffsetY = 0 mm , Depth = 5 mm

Spindle Distance = 156 mm , Radius = 5 mm , Length = 30 mm , Width = 26 mm

Path_Feature_20 , Facial Plane :

Groove Profile = Straight , Width = 4 mm , Depth = 5 mm

Path Type = Circular , CenterX = 0 mm , CenterY = 0 mm , Radius = 26 mm

Start Angle = 335 deg , End Angle = 70 deg , Spindle Distance = 156 mm

4.3.4 Tool Definition

Clearance values for finish cut :

Side = 1 mm , Face = 1 mm

Tool#1 :

Operation/Feature = Rough Turning , Plane = External

Cutting Tool = Left Hand Turning Tool , Manufacturer = ISCAR

Grip System = PCLNR/L , Tool Holder = 2020K-12 , Tool No = 1 , Turret No = 1

Insert = CNMG , Code = 160616 , Grade = IC9025

Application = Rough Machining , Type = Roughing

Feed Rate = 0.45 mm/rev , Cutting Speed = 160 m/min , Depth of Cut = 2 mm

Tool#2 :

Operation/Feature = Finish Turning , Plane = External

Cutting Tool = Left Hand Turning Tool , Manufacturer = ISCAR

Grip System = PDJNR/L , Tool Holder = 2020K-11 , Tool No = 2 , Turret No = 3

Insert = DNMG , Code = 110402-NF , Grade = IC9025

Application = Finish Machining , Type = Finishing

Feed Rate = 0.16 mm/rev , Cutting Speed = 200 m/min , Depth of Cut = 1 mm

Tool#3 :

Operation/Feature = Step_Feature_1 , Plane = Lateral

Cutting Tool = YZ_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-4 , Tool Holder = 100A22-4C10 , Tool No = 3

Turret No = 2 , Insert = 100A22-4C10 , Code = N/A , Grade = IC300

Application = Shouldering , Type = Roughing , Feed Rate = 0.1395 mm/rev

Cutting Speed = 88 m/min , Depth of Cut = 3 mm

Tool#4 to 16 :

Operation/Feature = Path_Feature_1 to 13 , Plane = Lateral

Cutting Tool = YZ_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-3 , Tool Holder = 050A14-3C05 , Tool No = 4

Turret No = 4 , Insert = 050A14-3C05 , Code = N/A , Grade = IC300

Application = Slotting , Type = Roughing , Feed Rate = 0.067 mm/rev

Cutting Speed = 88 m/min , Depth of Cut = 2.5 mm

Tool#17 to 22 :

Operation/Feature = Path_Feature_14 to 19 , Plane = Circumferential

Cutting Tool = YZ_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-3 , Tool Holder = 060A16-3C06 , Tool No = 5

Turret No = 6 , Insert = 060A16-3C06 , Code = N/A , Grade = IC300

Application = Slotting , Type = Roughing , Feed Rate = 0.0808 mm/rev

Cutting Speed = 88 m/min , Depth of Cut = 3 mm

Tool#23 :

Operation/Feature = Path_Feature_20 , Plane = Facial

Cutting Tool = XY_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-3 , Tool Holder = 040A12-3C04 , Tool No = 6

Turret No = 8 , Insert = 040A12-3C04 , Code = N/A , Grade = IC300

Application = Slotting , Type = Roughing , Feed Rate = 0.0535 mm/rev

Cutting Speed = 88 m/min , Depth of Cut = 2 mm

Tool#24 :

Operation/Feature = Pocket_Feature_1 , Plane = Circumferential

Cutting Tool = YZ_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-3 , Tool Holder = 100A22-3C10 , Tool No = 7

Turret No = 2 , Insert = 100A22-3C10 , Code = N/A , Grade = IC300

Application = Slotting , Type = Roughing , Feed Rate = 0.134 mm/rev

Cutting Speed = 88 m/min , Depth of Cut = 3 mm

Tool#25,26 :

Operation/Feature = Pocket_Feature_2,3 , Plane = Facial

Cutting Tool = XY_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-3 , Tool Holder = 100A22-3C10 , Tool No = 8

Turret No = 10 , Insert = 100A22-3C10 , Code = N/A , Grade = IC300

Application = Slotting , Type = Roughing , Feed Rate = 0.134 mm/rev

Cutting Speed = 88 m/min , Depth of Cut = 3 mm

Tool#27 :

Operation/Feature = Hole_Feature_1 , Plane = Lateral

Cutting Tool = YZ_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-3 , Tool Holder = 030A10-3C03 , Tool No = 9

Turret No = 4 , Insert = 030A10-3C03 , Code = N/A , Grade = IC300

Application = Slotting , Type = Roughing , Feed Rate = 0.4035 mm/rev

Cutting Speed = 88 m/min , Depth of Cut = 1.5 mm

Tool#28 :

Operation/Feature = Hole_Feature_2 , Plane = Facial

Cutting Tool = XY_Plane_End Mill , Manufacturer = ISCAR

Grip System = ECC-A-3 , Tool Holder = 040A12-3C04 , Tool No = 10

Turret No = 10 , Insert = 040A12-3C04 , Code = N/A , Grade = IC300

Application = Slotting , Type = Roughing , Feed Rate = 0.535 mm/rev

Cutting Speed = 88 m/min , Depth of Cut = 2 mm

Table 4.1: Generated NC code for Case Study I.

O0001	N160 X89.000
N10 G21	N170 G28 U0 V0 W0
N20 G50 S3600	N180 G00 X85.000 Z3.000
N30 G28 U0 V0 W0	N190 X81.000
N40 T010100	N200 G01 Z-156.000 F0.3
N50 G99	N210 G00 X85.000
N60 G96 S150 M08	N220 Z3.000
N70 M03	N230 G00 X77.000
N80 G00 X89.000 Z6.000	N240 G01 Z-139.500 F0.3
N90 G01 Z2.000 F0.3	N250 G00 X81.000
N100 X0.0	N260 Z3.000
N110 G00 Z4.000	N270 G00 X73.000
N120 X89.000	N280 G01 Z-139.000 F0.3
N130 G01 Z1.000 F0.3	N290 G00 X77.000
N140 X0.0	N300 Z3.000
N150 G00 Z6.000	N310 G00 X69.000

Table 4.2: (continued) Generated NC code for Case Study I.

N320 G01 Z-139.000 F0.3	N580 Z3.000
N330 G00 X73.000	N590 G00 X41.000
N340 Z3.000	N600 G01 Z-62.477 F0.3
N350 G00 X65.000	N610 G00 X45.000
N360 G01 Z-138.936 F0.3	N620 Z3.000
N370 G00 X69.000	N630 G00 X37.000
N380 Z3.000	N640 G01 Z-55.305 F0.3
N390 G00 X61.000	N650 G00 X41.000
N400 G01 Z-129.677 F0.3	N660 Z3.000
N410 G00 X65.000	N670 G00 X33.000
N420 Z3.000	N680 G01 Z-44.207 F0.3
N430 G00 X57.000	N690 G00 X37.000
N440 G01 Z-122.333 F0.3	N700 Z3.000
N450 G00 X61.000	N710 G00 X29.000
N460 Z3.000	N720 G01 Z-31.500 F0.3
N470 G00 X53.000	N730 G00 X33.000
N480 G01 Z-95.667 F0.3	N740 Z3.000
N490 G00 X57.000	N750 G00 X25.000
N500 Z3.000	N760 G01 Z-29.410 F0.3
N510 G00 X49.000	N770 G00 X29.000
N520 G01 Z-77.368 F0.3	N780 Z3.000
N530 G00 X53.000	N790 G00 X21.000
N540 Z3.000	N800 G01 Z1.500 F0.3
N550 G00 X45.000	N810 G00 X25.000
N560 G01 Z-68.852 F0.3	N820 Z3.000
N570 G00 X49.000	N830 G00 X85.000

Table 4.3: (continued) Generated NC code for Case Study I.

N840 X0.000	N1100 X22.000 Z-2.000
N850 G01 Z1.000 F0.3	N1110 X22.000 Z-31.000
N860 X20.000 F0.3	N1120 X26.784 Z-32.961
N870 X24.000 Z-1.000	N1130 X30.000 Z-41.000
N880 X24.000 Z-30.000	N1140 G02 X40.000 Z-66.000 R100.000
N890 X28.784 Z-31.961	N1150 G03 X50.000 Z-91.000 R100.000
N900 X32.000 Z-40.000	N1160 G01 X56.000 Z-131.000
N910 G02 X42.000 Z-65.000 R100.000	N1170 G03 X60.000 Z-133.000 R2.000
N920 G03 X52.000 Z-90.000 R100.000	N1180 G01 Z-139.000
N930 G01 X58.000 Z-130.000	N1190 G02 X64.000 Z-141.000 R2.000
N940 G03 X62.000 Z-132.000 R2.000	N1200 G01 X74.000 F0.16
N950 G01 Z-138.000	N1210 X78.000 Z-143.000
N960 G02 X66.000 Z-140.000 R2.000	N1220 Z-156.000
N970 G01 X76.000 F0.3	N1230 G00 X85.000
N980 X80.000 Z-142.000	N1240 Z0.000
N990 Z-155.000	N1250 G28 U0 V0 W0 M09
N1000 Z-156.000	N1260 T050300
N1010 G00 X85.000	N1270 G96 S70 M08
N1020 Z0.000	N1280 G00 X89.000 Z6.000
N1030 G28 U0 V0 W0 M09	N1290 Z-113.000
N1040 T030200	N1300 X59.000
N1050 G96 S250 M08	N1310 G01 X53.000 F0.085
N1060 G00 X85.000 Z3.000	N1320 G00 X59.000
N1070 X0.000	N1330 Z-116.000
N1080 G01 Z0.000 F0.16	N1340 G01 X53.000 F0.085
N1090 X18.000 F0.16	N1350 G00 X59.000

Table 4.4: (continued) Generated NC code for Case Study I.

N1360 Z-113.000	N1620 G00 X49.000
N1370 G01 X51.000 F0.085	N1630 Z-116.000
N1380 G00 X57.000	N1640 G01 X43.000 F0.085
N1390 Z-116.000	N1650 G00 X49.000
N1400 G01 X51.000 F0.085	N1660 Z-113.000
N1410 G00 X57.000	N1670 G01 X41.000 F0.085
N1420 Z-113.000	N1680 G00 X47.000
N1430 G01 X49.000 F0.085	N1690 Z-116.000
N1440 G00 X55.000	N1700 G01 X41.000 F0.085
N1450 Z-116.000	N1710 G00 X47.000
N1460 G01 X49.000 F0.085	N1720 Z-113.000
N1470 G00 X55.000	N1730 G01 X39.000 F0.085
N1480 Z-113.000	N1740 G00 X45.000
N1490 G01 X47.000 F0.085	N1750 Z-116.000
N1500 G00 X53.000	N1760 G01 X39.000 F0.085
N1510 Z-116.000	N1770 G00 X45.000
N1520 G01 X47.000 F0.085	N1780 Z-113.000
N1530 G00 X53.000	N1790 G01 X39.000 F0.085
N1540 Z-113.000	N1800 Z-116.000
N1550 G01 X45.000 F0.085	N1810 G00 X89.000
N1560 G00 X51.000	N1820 G28 U0 V0 W0 M09
N1570 Z-116.000	N1830 T070400
N1580 G01 X45.000 F0.085	N1840 G96 S94.5 M08
N1590 G00 X51.000	N1850 G00 X89.000 Z6.000
N1600 Z-113.000	N1860 Z5.127
N1610 G01 X43.000 F0.085	N1870 X25.861

Table 4.5: (continued) Generated NC code for Case Study I.

N1880 G01 X21.861 F2.5	N2160 Z5.127
N1890 G32 Z-32.436 F2.5	N2170 X24.188
N1900 G00 X26.000	N2180 G01 X20.188 F2.5
N1910 Z5.127	N2190 G32 Z-32.436 F2.5
N1920 X25.582	N2200 G00 X26.000
N1930 G01 X21.582 F2.5	N2210 Z5.127
N1940 G32 Z-32.436 F2.5	N2220 X23.909
N1950 G00 X26.000	N2230 G01 X19.909 F2.5
N1960 Z5.127	N2240 G32 Z-32.436 F2.5
N1970 X25.303	N2250 G00 X26.000
N1980 G01 X21.303 F2.5	N2260 Z5.127
N1990 G32 Z-32.436 F2.5	N2270 X23.630
N2000 G00 X26.000	N2280 G01 X19.630 F2.5
N2010 Z5.127	N2290 G32 Z-32.436 F2.5
N2020 X25.024	N2300 G00 X26.000
N2030 G01 X21.024 F2.5	N2310 Z5.127
N2040 G32 Z-32.436 F2.5	N2320 X23.351
N2050 G00 X26.000	N2330 G01 X19.351 F2.5
N2060 Z5.127	N2340 G32 Z-32.436 F2.5
N2070 X24.745	N2350 G00 X26.000
N2080 G01 X20.745 F2.5	N2360 Z5.127
N2090 G32 Z-32.436 F2.5	N2370 X23.072
N2100 G00 X26.000	N2380 G01 X19.072 F2.5
N2110 Z5.127	N2390 G32 Z-32.436 F2.5
N2120 X24.467	N2400 G00 X26.000
N2130 G01 X20.467 F2.5	N2410 X89.000 Z6.000
N2140 G32 Z-32.436 F2.5	N2420 G28 U0 V0 W0
N2150 G00 X26.000	N2430 M02

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS FOR FUTURE WORK

The aim of the present study was to develop a feature based design software with the use of the pre-developed primitive library and procedure done for parts to be machined in a four-axis CNC lathe.

Sander [23] developed a software capable of designing complex parts by using a primitive library containing features of turning and milling operations. The pre-developed package used AutoCAD R12 for the visualization process. Despite being a great workout for designing and developing parts to be machined using CNC turning centers; lots of improvement had to be done in order to work such a fully operational desktop CAD/CAM package.

Selection of AutoCAD software was previously a good idea to design parts for axisymetric operations. But the developed workout indicates that; AutoCAD capabilities is not as good enough to design features for non-axisymetric operations. To resolve this failure a feature based CAD package which can be capable of design all of the features possible to manufacture with a turning center has to be used. By selecting within a number of famous brands of software developers; Autodesk's INVENTOR software was chosen to perform designing applications.

The previously developed feature codes are reconstructed by using the INVENTOR's application programming interface. The designed parts can also be saved and most importantly be modified through INVENTOR software for using on other applications such as finite element analysis tools.

The previous cutting tool selection and machining parameter definition modules were recreated to allow users to select, create and save cutting tools with proper machinability parameters stated as in cutting tool manufacturers catalogs.

NC code and tool path generation algorithms were created for turning and milling related operations. Moreover the tool path is drawn to simulate the machining process in order to allow user to check and verify the code generated by the developed software.

After all the upgrades and modifications; the developed software became an user friendly desktop CAD/CAM package capable of design and machine a wide range of various different geometries for four-axis CNC turning centers. But there are still things to be done for the program to be fully operational for the future use.

To start with the design process; the case studies shows that pocket, pocket-island and path features should also be radially patterned with respect to the circumferential plane of the workpiece to reduce the overall number of features used by each project. On the other hand there should be custom profiles for pocket and pocket-island features rather than machining only rectangular and circular profiles. Another improvement can be done on path features by allowing user to design custom paths combined with linear and circular profiles again to reduce the overall number of path features used for lateral or facial plane. Moreover new features can be constructed to allow helical paths on facial and lateral plane.

Again some improvements can also be done on NC code and tool path generation algorithms. For rough and finish cutting on turning the outer and inner geometries; the developed package is capable of generating codes for external profiles which are increasing from the turret side to the spindle side and internal profiles which are decreasing from the turret side to the spindle side. These codes must be expanded to allow machining of any kind of geometries for internal and external profiles. On the other hand a fully working pocket-island algorithm must be develop to allow the machinability of designed pocket-island geometries.

A final recommendation can be given on simulation process that; the NC code verification software that Salihler S. [22] has developed in 2001 can be combined to simulate all the machine motions and all possible errors and to visualize the machined workpiece to check for the desired dimensions.

REFERENCES

[1] CATIA NC-Manufacturing. http://www.3ds.com, August 2005.

[2] EdgeCAM. http://www.edgecam.com, August 2005.

[3] Iscar. http://www.iscar.com, August 2005.

[4] Mastercam X. http://www.mastercam.com, August 2005.

[5] Pro/Engineer Complete Machining. http://www.ptc.com, August 2005.

[6] Unigraphix NX. http://www.ugs.com, August 2005.

[7] Arıkan, M.A.S., Totuk, H.O. "Design by Using Machining Operations". *Annals of the CIRP*, 41(1):185–188, 1992.

[8] Boz, O. "Development of CAD/CAM Program for Milling Machines". Master's thesis, Mechanical Engineering Department, Middle East Technical University, Ankara, TURKEY, 1998.

[9] Chang, T.C., Wysk, R.A., Wang,H.P. *Computer Aided Manufacturing*. Prentice Hall, 1991.

[10] Chow J. G., Sakal L.R. "An Integrated Intelligent Machining System for Axisymmetric Parts Using PC-based CAD and CAM Software Packages". *Journal of Intelligent Manufacturing*, 5(2):93–102, 1994.

[11] Delbressine, F.L.M., Hijink, J.A.W. "Discrete Part Design by Taking Manufacturing Restrictions into Account". *Annals of the CIRP*, 40(1):171–174, 1991.

[12] Elkeran A., El-baz M. A. "CNC Manufacturing of Complex Surfaces Based on Solid Modeling". *Lecture Notes in Computer Science*, pages 841–848, 2004.

[13] Hitachi Seiki. Hitachi Seiki-Seicos LII Users Manual, 1989.

[14] İlik, Y. "A Desktop CAD/CAM Package for Rotational Parts". Master's thesis, Mechanical Engineering Department, Middle East Technical University, Ankara, TURKEY, 1995.

[15] ISCAR. ISCAR Mill, Helimill-Heliquad-Helistar-Heliball, 1996.

[16] ISCAR. ISCAR Mill, Slotting and Slitting tools, 1999.

[17] ISCAR. ISCAR Turn Plus, The complete range of cutting tools, 1999.

[18] ISCAR. ISCAR Mill Plus, The complete range of cutting tools, 2000.

[19] Öztürk, F. "Machinability and Tool Databases with Automatic Tool Selection Module for a Desktop Turning CAD/CAM Package". Master's thesis, Mechanical Engineering Department, Middle East Technical University, Ankara, TURKEY, 1998.

[20] Peklenik, J., Sekolonik, R. "Development of the Part Spectrum Database for Computer Integrated Manufaturing Systems (CIMS)". *Annals of the CIRP*, 39(1):471–474, 1990.

[21] Salamons, O., Van Slooten, F. *"Link Between CAD and CAPP"*. PhD thesis, Production and Design Engineering Department, Twente University, Enschede , HOLLAND, 1994.

[22] Salihler, S. "Development of Simulation and Verification Software For CNC Turning Machines". Master's thesis, Mechanical Engineering Department, Middle East Technical University, Ankara, TURKEY, 2001.

[23] Sander, T.Z. "Development of Primitive Library and a Procedure For Design of Parts to be Machined in a Four Axis CNC Lathe". Master's thesis, Mechanical Engineering Department, Middle East Technical University, Ankara, TURKEY, 1999.

[24] Saygın, C. "A Rule Based Approach in Sequencing Machining Operations for Rotational Components". Master's thesis, Mechanical Engineering Department, Middle East Technical University, Ankara, TURKEY, 1992.

[25] Şen, Z., Özçilingir, N. Standart Makine Elemanları Çizelgeleri. DEHA publishing.

[26] Stainslaw, Z. "System Integrated Product Design, CNC Programming and Postprocessing for Three-Axis Lathes". *Journal of Materials Processing Technology*, 109:294–299, 2001.

[27] Su C.J., Sun T.L., Wu C.N. and Mayer R.J. "An Integrated Form-Feature-Based Design System for Manufacturing". *Journal of Intelligent Manufacturing*, 6(5):277–290, 1995.

[28] Susuzlu, T. "NC Verification for Turning Operations". Master's thesis, Mechanical Engineering Department, Middle East Technical University, Ankara, TURKEY, 2001.

[29] Zezhong C.C., Zuomin D., Geoffrey W.V. "Automated Surface Subdivision and Tool Path Generation for 3-1/2 Axis CNC Machining of Sculptured Parts". *Journal of Computers in Industry*, 50:319–331, 2003.

APPENDIX A

USER'S GUIDE

A.1 System Requirements and Installation

The developed software and Autodesk INVENTOR works on Windows 2000 and XP operating systems. Any hardware configuration that is capable of running INVENTOR is adequate. It must be paid attention that INVENTOR software is installed prior to the developed program. The developed program can be installed by running the "Setup.exe" file on SETUP CD. To run SETUP CD;

1 : Insert the SETUP CD into your drive.

2 : Click the "Start" button then click "Run".

3 : Type < *drive* >:\setup.exe , Ex: d:\setup.exe

4 : Follow on screen instructions.

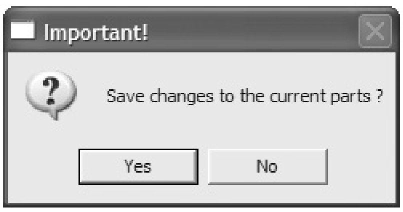
A.2 Introduction

The developed software is an user friendly combination of computer aided design and manufacturing package. Therefore it is quite easy to design complex parts combining features of turning and milling operations with the use of designed procedure and the developed software. The capabilities of four-axis turning centers brings out large amount of flexibility in design progress. The risk of designing a part which is impossible to manufacture is eliminated by using the machinable features.

The developed software is easy to use by including graphical interface with automatically updating pictures and online guidance throughout. It is also simple to understand and intended for the usage of shop level people. In fact the user is not required to have the knowledge to use INVENTOR program since everything needed for design is reached though within the developed software. However basic understanding of INVENTOR may be useful.

A.3 Getting Started and System Overview

The program can be started by clicking the "InventorNC" icon in the "InventorNC" folder of programs group. The program first checks the INVENTOR software is already opened. if not then it also runs inventor. If a document is opened previously then the program asks to save the document for future studies. Whether save or not the previously opened document is closed and program asks to create a document for the new design process. (Figure A.1) For each design process the developed software creates two documents which are "Raw Part_PartNo" and "Machined Part_PartNo". These are the documents of the part before machining and after machining.



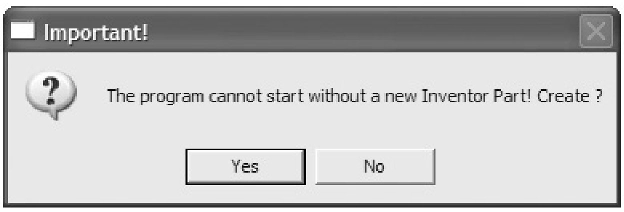


Figure A.1: Document checking.

The title bar can be used to activate the base window or to reposition it on the screen. The file name of the loaded part appears on the main menu in the file name bar. It is possible to switch to INVENTOR anytime by clicking anywhere in the INVENTOR screen. General commands which involves file management operations, setup information, visualization and help documents are placed in the menu bar and commands related to part definition and manufacturing operations can be reached through the command buttons in the base window. (Figure A.2)

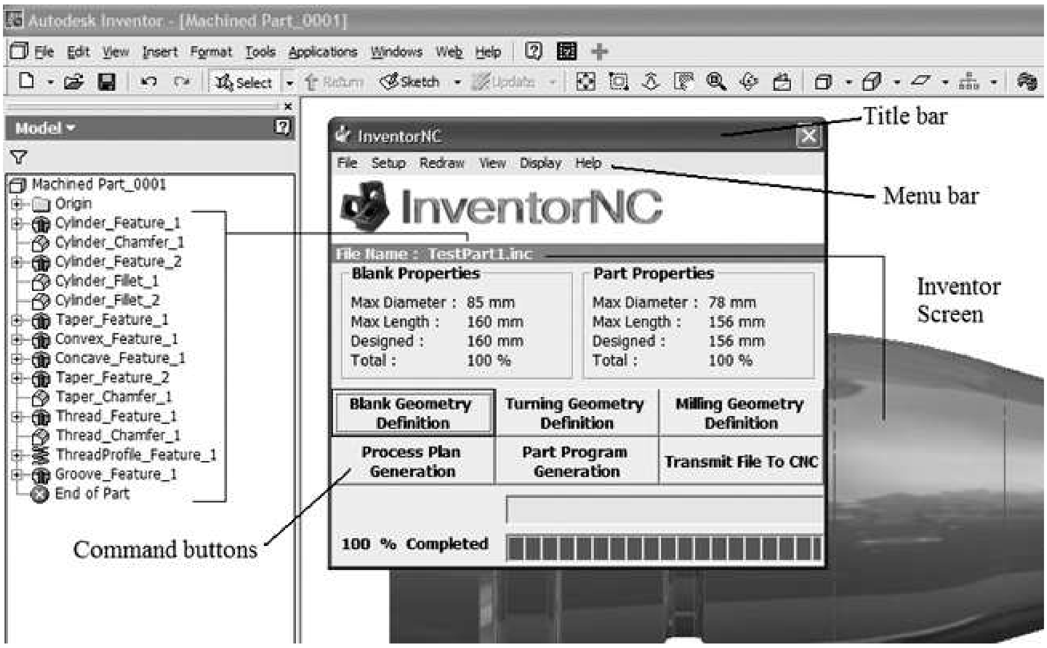


Figure A.2: The main menu.

A.4 Menu Bar Commands

A.4.1 File Menu

File pulldown menu contains several commands which are; new, open, save, save as and exit. (Figure A.3) "New" command clears the data in the memory and opens up a new part document in INVENTOR software. Again the program asks user for saving previous document and after closing previous file re-asks for opening a new part document. Open command contains another pull-down menu for opening a previously created one or one in the last four files opened. "Open any file" command calls a dialogue window (Figure A.3) to get the name of a previously designed part and draws it on INVENTOR screen. "Save" command saves the changes made to the file being used for design process. "Save As" saves the design into a new file which previously named via a dialog similar to the one for "Open any file" command.

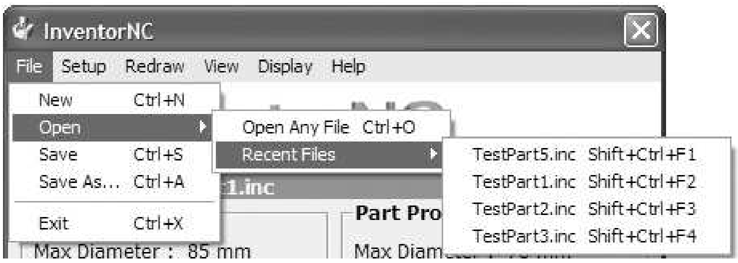


Figure A.3: File pulldown menu.

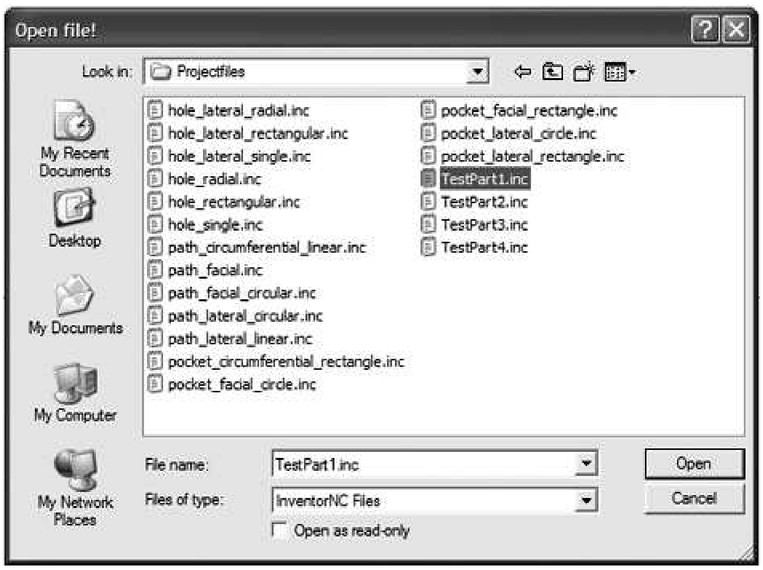


Figure A.4: Open dialogue window.

Finally "Exit" command quits the program after asking the user whether to save the part if there are changes to the part design since the last saved operation.

A.4.2 Setup Menu

Setup command in the main menu opens the Setup information window shown in Figure A.5. Setup information window contains some important parameters which must be configured prior to the design process.

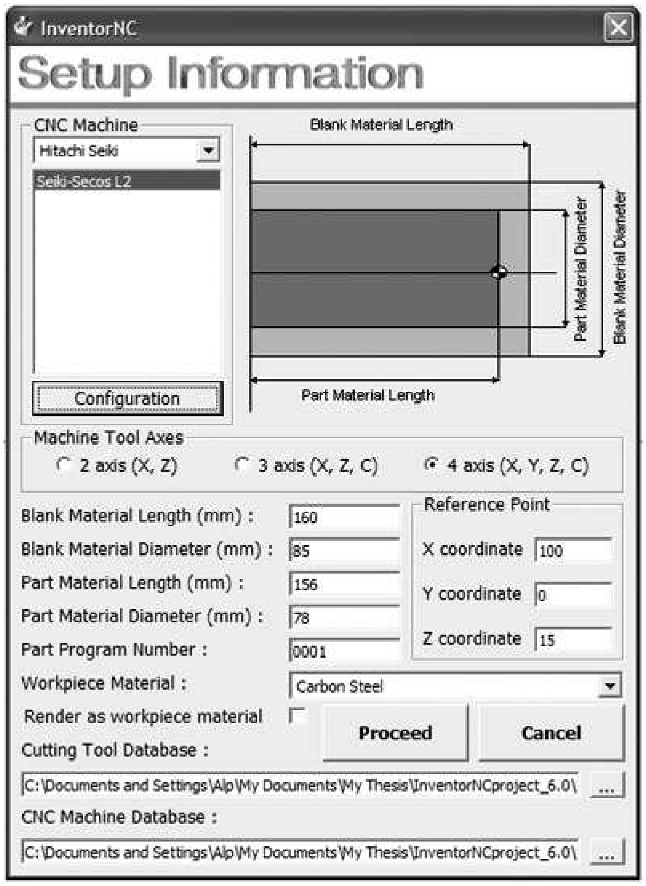


Figure A.5: Setup information window.

CNC machines have control units which can be different from manufacturer. Some manufacturers uses Fanuc system but mostly they have their control systems like in Siemens or Phillips. Therefore Some CNC codes can be different from machine to machine. To prevent wrong interpretations while coding; the manufacturer and model of the machine must be selected in the setup window. By clicking the configuration button the CNC configuration window appears. (Figure A.6)

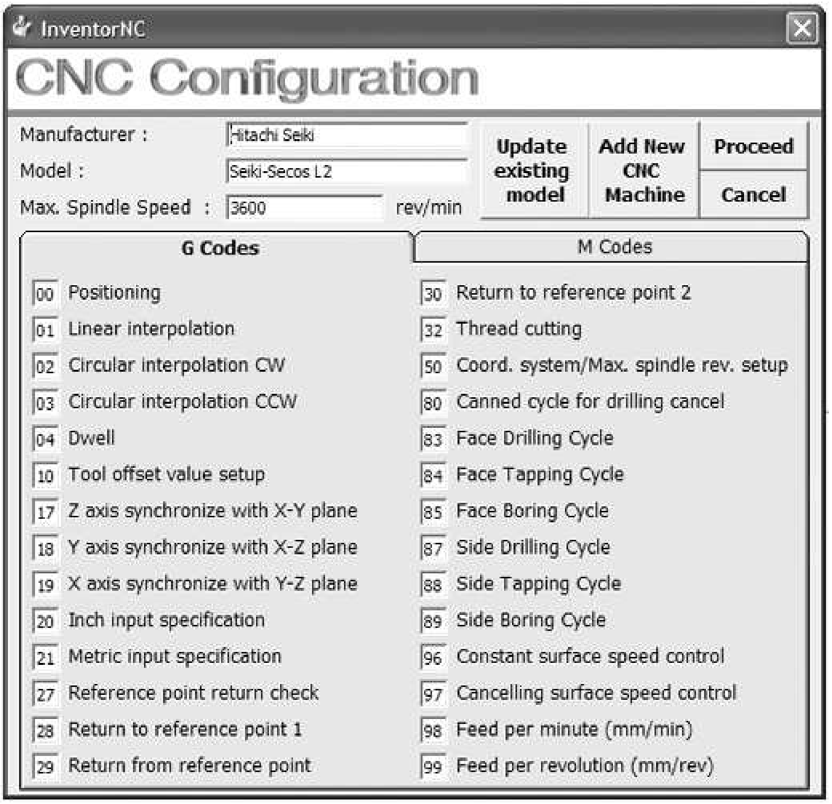


Figure A.6: CNC configuration window.

The selected CNC machine model with appropriate manufacturer is shown together with maximum spindle speed which specifies the upper limit of spindle speed in rpm and necessary G (prepatory) and M (miscellaneous) codes which are used for the machining operations. User can either create or update a machine by entering the properties defined in user manual or select a proper one in the list.

The number of machine tool axis must be specified for the program to disable features that cannot be machined for three or two axis machines.

Maximum length and diameter values of blank or raw part and machined part must be entered according to the figure given in upper right of setup information window. Part program number and coordinates of reference point on X, Y and Z axes must be defined for NC code generation. Workpiece material is selected from the pull-down list and will be used for calculating machining parameters taken from the cutting tool database. The cutting tool database file name specifies the name and the location of the database file created using Microsoft Access. User defines cutting tool information within the program and saves the new record in this database. Again the CNC machine database file name specifies the database file used for storing configurations of CNC machines with different model and manufacturers.

A.4.3 Redraw Menu

Clicking the redraw menu lets the program to clear all the features in the drawing and regenerates raw and machined part according to the feature information in the memory.

A.4.4 View and Display Menu

User can view blank or machined part by selecting different angles from the view pulldown menu. The available chooses are; front, left, top and isometric. Moreover wireframe, hidden line and shaded visualizations can be made from display pulldown menu. (Figure A.7)

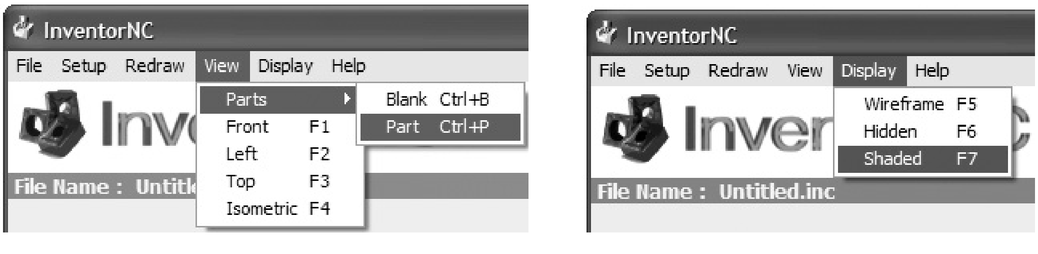


Figure A.7: View and display pulldown menu.

A.4.5 Help Menu

Help menu contains useful information about the main titles of the program.

A.5 Part Design Using Command Buttons

The part design sequence of the program follows the order of the buttons from upper left to the lower right. (Figure A.8) The procedure starts with defining blank geometry. Then, the geometry formed by turning and milling operations must be defined followed by process planning and part program generation. Finally the generated NC code can be transmitted to the machine.

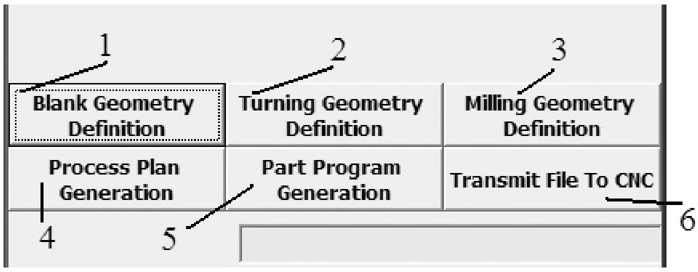


Figure A.8: Design sequence.

A.5.1 Blank & Turning Geometry Definition

Blank and turning geometry is defined in the same manner thus all the features are turning related machining operations. The program refers to "Turning Design" pulldown menu by clicking "Blank Geometry Definition" or "Turning Geometry Definition" command. The "Turning Design" menu (Figure A.9) contains three pulldown menus which are "Feature Operations", "View" and "Display". "Feature Operations" pulldown menu is used for adding, inserting, modifying, moving, copying and deleting features. "View" and "Display" pulldown menus are for visualization and same as in main menu. The "Inner Profile" and "Outer Profile" buttons determines if the added or edited feature is internal or external. The previously designed blank and part features is added in the listbox for future modifications.

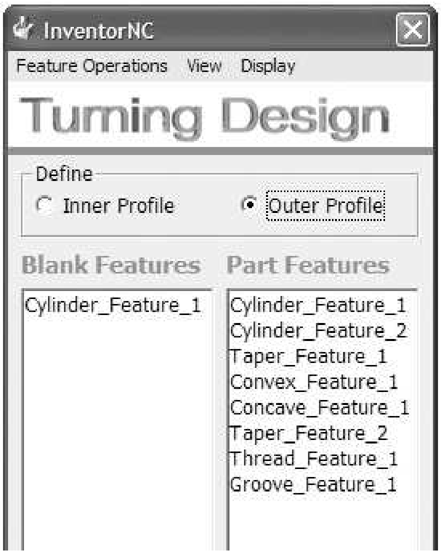


Figure A.9: Turning design menu.

A.5.1.1 Adding Feature

Feature addition can be made by Add command in "Feature Operations" pulldown menu. (Figure A.10)

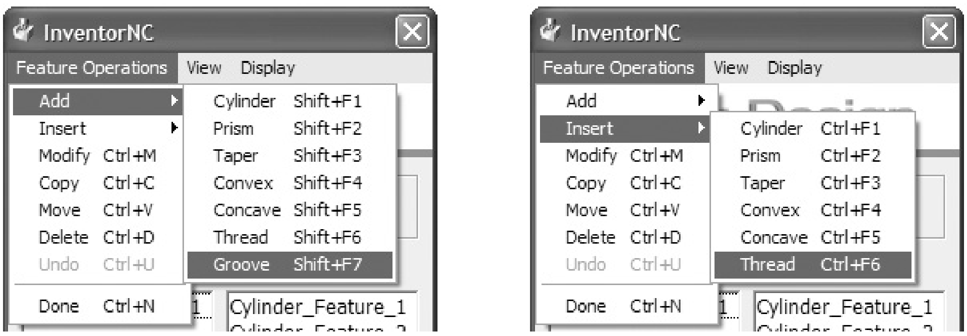


Figure A.10: Feature operations pulldown menu for turning design.

When any of the listed features are selected, a window for information entry appears for each feature. An example of this window for a cylinder feature is shown in Figure A.11. The diameter and length values of the cylinder must be entered and spindle side, turret side conditions have to be defined. The detailed configurations of spindle and turret side edges of cylinder feature can be done on lower section of the menu.

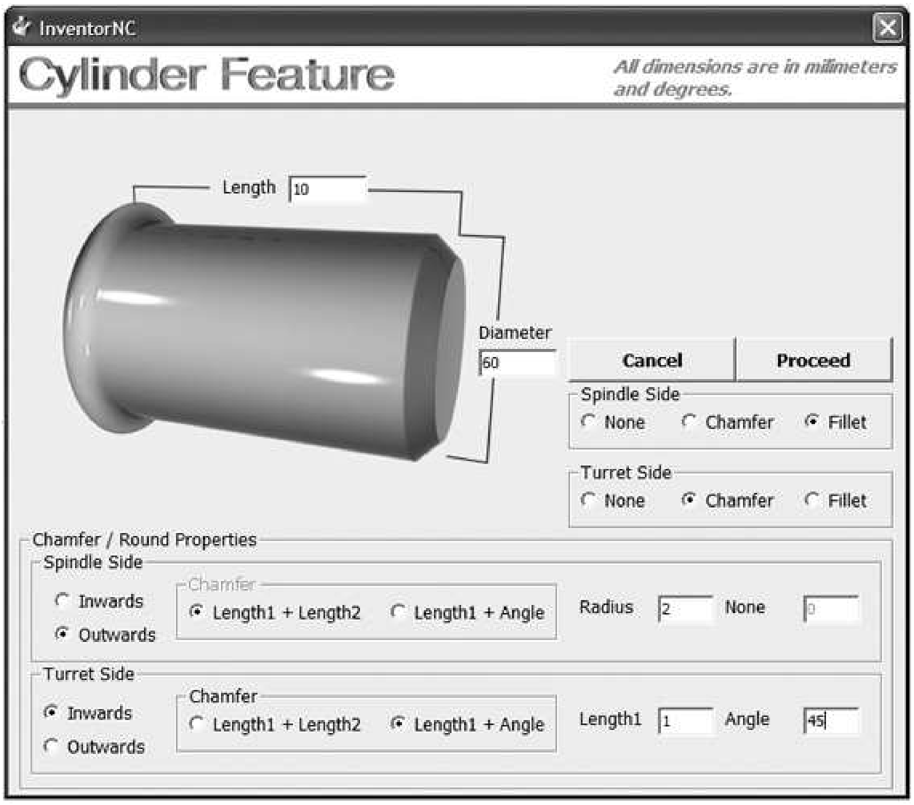


Figure A.11: Cylindrical feature information entry window.

A.5.1.2 Copying Feature

Copy button is used to copy a feature to new position with the same dimensions. First the feature to be copied should be selected from the list then the program asks to select a feature to indicate the insert position. After clicking "OK" on the message text the feature to indicate the insert point of copied feature is selected from the list and again "Copy" button is clicked for confirming the selected feature. Finally the program asks the side of selected feature to copy to complete the process. (Figure A.12)

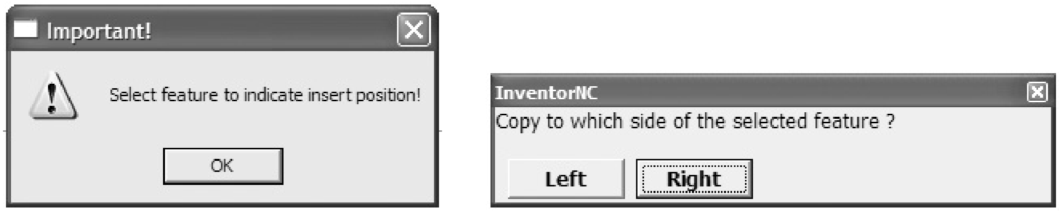


Figure A.12: Copying feature.

A.5.1.3 Modifying Feature

Modify function is used to modify a previously created feature. In order to modify a feature; the user should select the feature in the list and click the modify button. When a feature is selected, size information for that feature shows up as in Figure A.11. Changes to these values are updated in the part design.

A.5.1.4 Deleting Feature

Delete function is used to delete a previously created feature. In order to delete a feature; the user should select the feature in the list and click the delete button. The selected features are erased from the memory and the drawing. The features to the right of the deleted feature are shifted to the left for outer profile, and the features to the left of the deleted feature are shifted to the right for inner profile as the length of the deleted feature.

A.5.1.5 Inserting Feature

Insert function is used if the new feature is required to be placed anywhere else between previously created features. The feature to be inserted is selected from the insert pulldown menu. (Figure A.10) The program asks to select a feature to define the insertion point from the list. After confirming the feature to indicate the insertion point from the list, user select the same feature to insert from the insert pulldown menu. Finally the program asks which side to insert the selected feature and after the selection the information entry window for the inserted feature appears. (Figure A.11)

A.5.1.6 Moving Feature

Move function is used to delete an existing feature and place it to a new location. First the feature to move is to be selected from the list and move button is clicked. The program asks to select a feature to define the point to move from the list. After confirming the pivot feature from the list and pressing move button again, the program asks to select the side of feature to move the previously selected one.

A.5.1.7 Undoing Feature

"Undo" button resets the effects of the last action which can be adding, copying, modifying, deleting inserting or moving a feature.

A.5.2 Milling Geometry Definition

"Milling Geometry Definition" button is used to define the shape of the part that can be manufactured by using rotary tools in X and Z axes directions. After clicking this button, the menu which is shown in Figure A.13 is displayed.

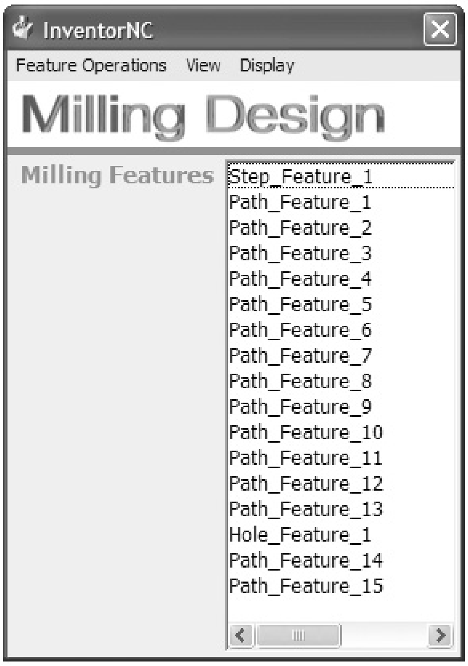


Figure A.13: Milling design menu.

The use of modify, delete and undo buttons are exactly the same as the corresponding buttons in the "Turning Design" menu. However, to add a new feature, first the placement plane of the new feature must be selected. (Figure A.14)

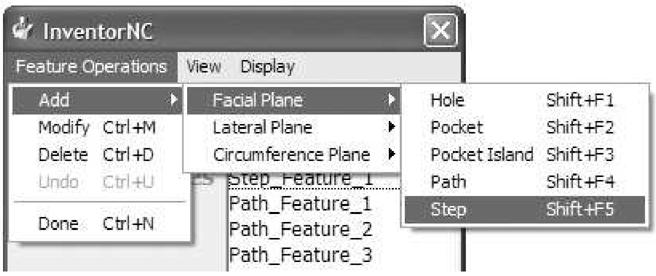


Figure A.14: Feature operations pulldown menu for milling design.

Facial plane features are planar features that can be placed on the face of the part. They require use of Z-axis tools and tool movements along X,Y and C axis. Lateral plane features are planar features that can be placed on the side of the part. These features can be machined by using an X-axis rotating tool and requires X, Y, Z and C axes movements of four-axes lathe. Finally circumferential features are placed on the side of the part to remove material from the circular surface of the main geometry. C, X and Z axes move for machining the circumferential features by using X-axis rotary tools.

A.6 Preparing Process Plan

The machining order of the features in the designed part are determined in ”Process Plan” window by pressing ”Process Plan Generation” button. (Figure A.15) When the window is loaded for the first time, process plan is generated and listed for the user in order to change the machining priority if necessary.

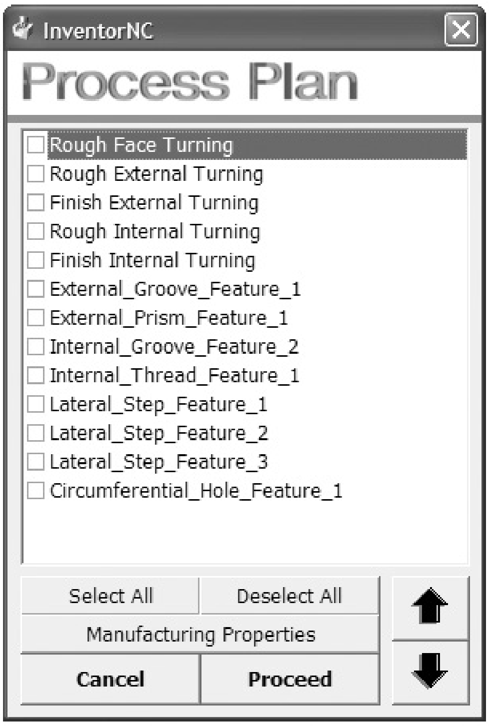


Figure A.15: Process plan window.

The checkboxes to the left of each item in the process plan shows which features are going to be machined. A feature with an empty checkbox will be skipped for NC code generation. "Select All" button checks all the boxes and "Deselect All" button clears all the check marks in the boxes. By clicking Proceed button the program saves the process plan. When the window is loaded next time, the program asks permission to recalculate the process plan.

If the blank is longer than the defined turning geometry, automatically rough face turning operation is added prior to the rough external turning operation. Finish external turning involves the finish operation of the face and if the user clears the checkbox for facing operation then the extra amount of the tip of the part is machined during rough external turning operation.

Up and down arrow buttons on the right of the process plan are used to change the order of the selected items in the list. When the up arrow is pressed, the selected item in the list shifts one step higher and when the down arrow is pressed it moves down a step.

A.6.1 Manufacturing Properties

"Manufacturing Properties" button calls up an appropriate window for entering manufacturing related information for the selected operation. (Figure A.16) This window is developed to make the cutting tool selection process more easy and flexible. User can select the operation on frame ***1.Select Operation***. First two options declares roughing and finishing operations and selecting one of these disables ***3.Select Feature*** frame. Other options in the first frame are enabled if the designed workpiece involves the related feature. Selecting an enabled option in the first frame rather than rough or finish turning shows the related features on ***3.Select Feature*** frame.

The workplane of machining operation must be selected from ***2.Select Plane*** frame. For turning related operations; user can choose between external and internal profiles whereas for milling related operations facial, lateral and circumferential planes are available. User also have to enter the necessary clearance values stated in ***Finish Clearance*** frame for the generation of NC code.

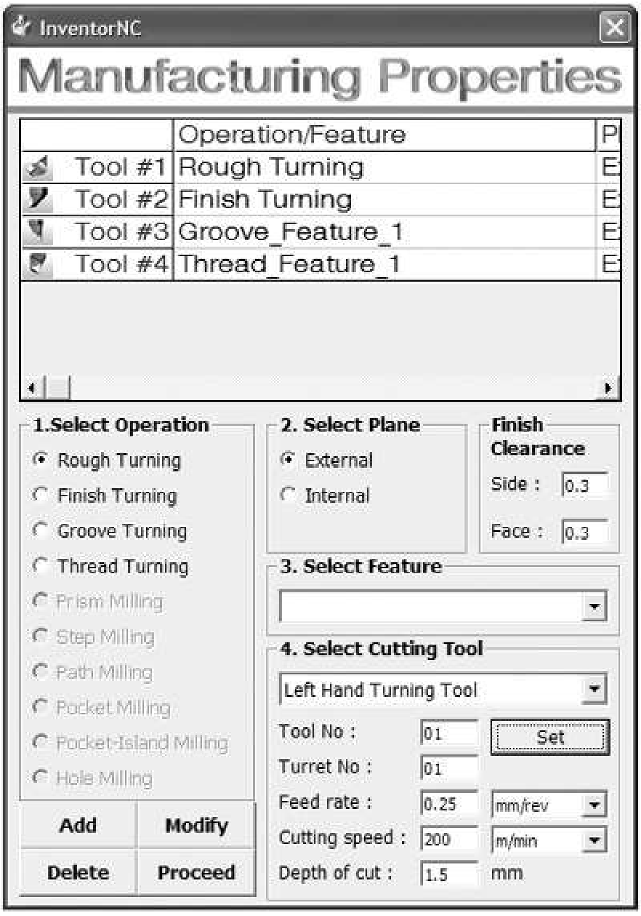


Figure A.16: Process plan window.

A.6.2 Cutting Tool Selection

Cutting tool can be selected in frame ***4.Select Cutting Tool*** shown in Figure A.16 and other parameters can either be entered by the user or automatically calculated by the program. By clicking the "Set" button the cutting tool selection window appears (Figure A.17) Cutting tool selection window helps user to select the proper model of tool holder with insert from the catalog of manufacturers. If a turning tool is selected from frame ***4.Select Cutting Tool*** shown in Figure A.16, program refers to external cutting section of the cutting tool database to look for recorded grip systems for selected manufacturer.

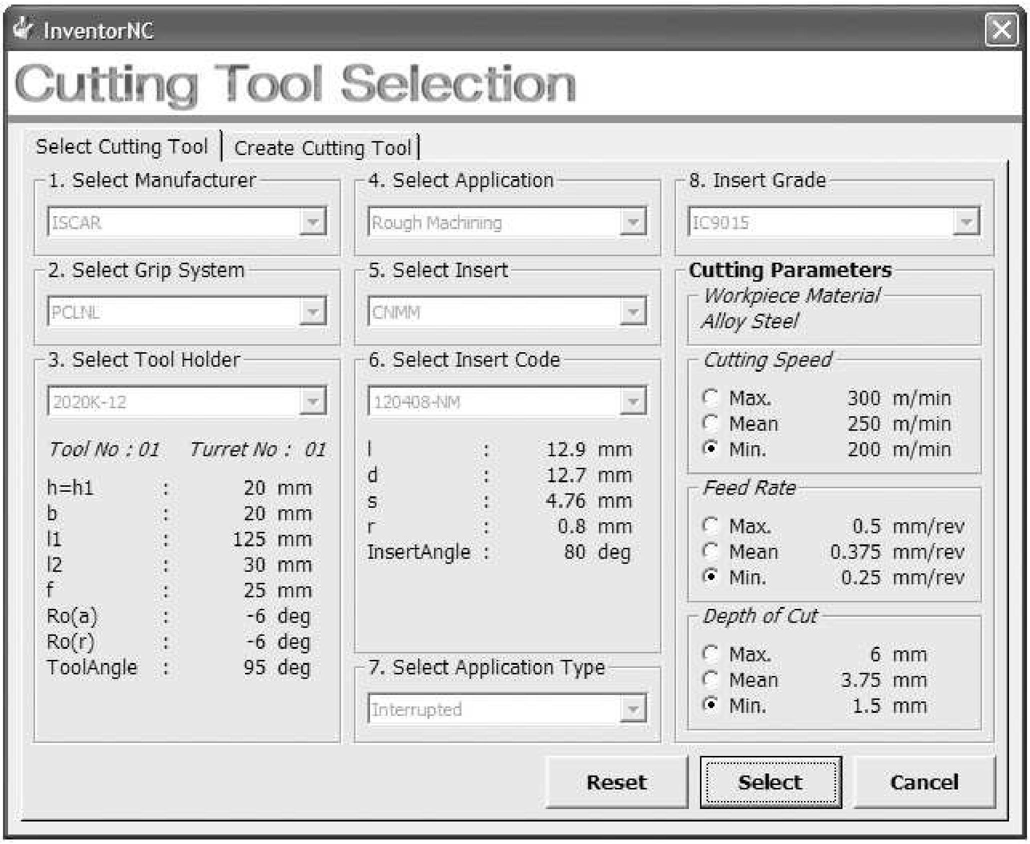


Figure A.17: Cutting tool selection window.

Selected cutting tool parameters are saved to memory by pressing the "Select" button in "Cutting Tool Selection" window and machining parameters are written in related boxes shown in frame ***4.Select Cutting Tool*** tab in Figure A.16. If all the configurations are stated for the selected cutting tool, user can press "Add" button to save to memory in "Manufacturing Properties" window.(Figure A.18) "Delete" button is used for deleting a previously defined cutting tool by entering the tool number stated left corner of the list. (Figure A.19)

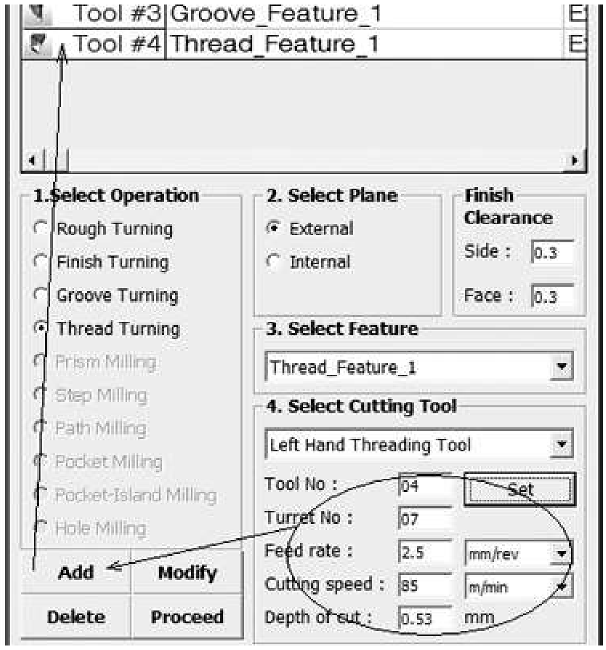


Figure A.18: Add cutting tool.

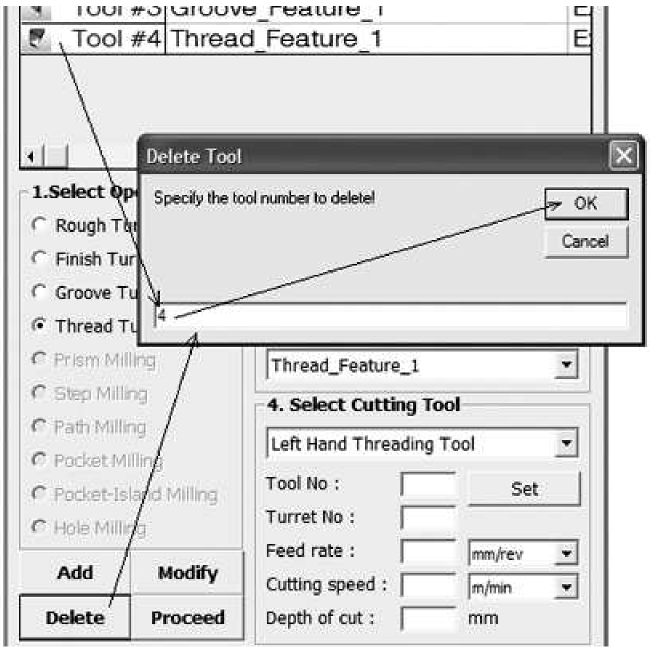


Figure A.19: Delete cutting tool.

"Modify" button is used for modifying a previously defined cutting tool. After stating the new configurations "Modify" button is pressed and by entering the tool number stated left corner of the list, all the properties are written over previously selected one. (Figure A.20)

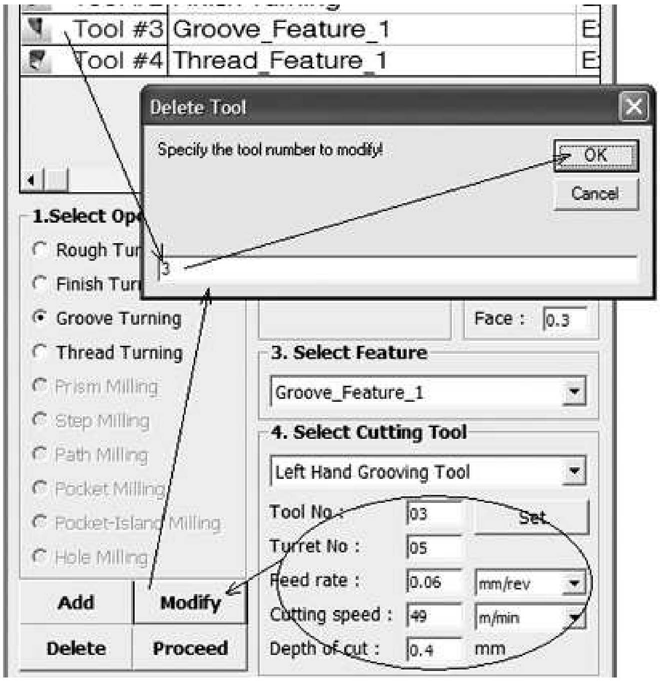


Figure A.20: Modify cutting tool.

Users must create cutting tools in order to select from the database. The cutting tool creation window is designed for selecting cutting tools like in manufacturer catalogs. When the cutting tool creation window first shown user have to fill all necessary information of the tool to be defined. ISCAR's turning and milling catalogs are taken as a reference for developing the database.

A tutorial of creating a cutting tool is shown below for a reference to users. The following tutorial is the illustration of selecting a left hand turning tool for rough external machining of an alloy steel workpiece material.

A.6.2.1 Tool Selection Tutorial

The following pages and information of cutting tools are taken from ISCAR's catalogs [17], [15], [18], [16] and website. [3] First the "PCLNL" grip system is selected for external turning application from index A. (Figure A.21) The related tool holders for this system is in page I 13 and L 11.

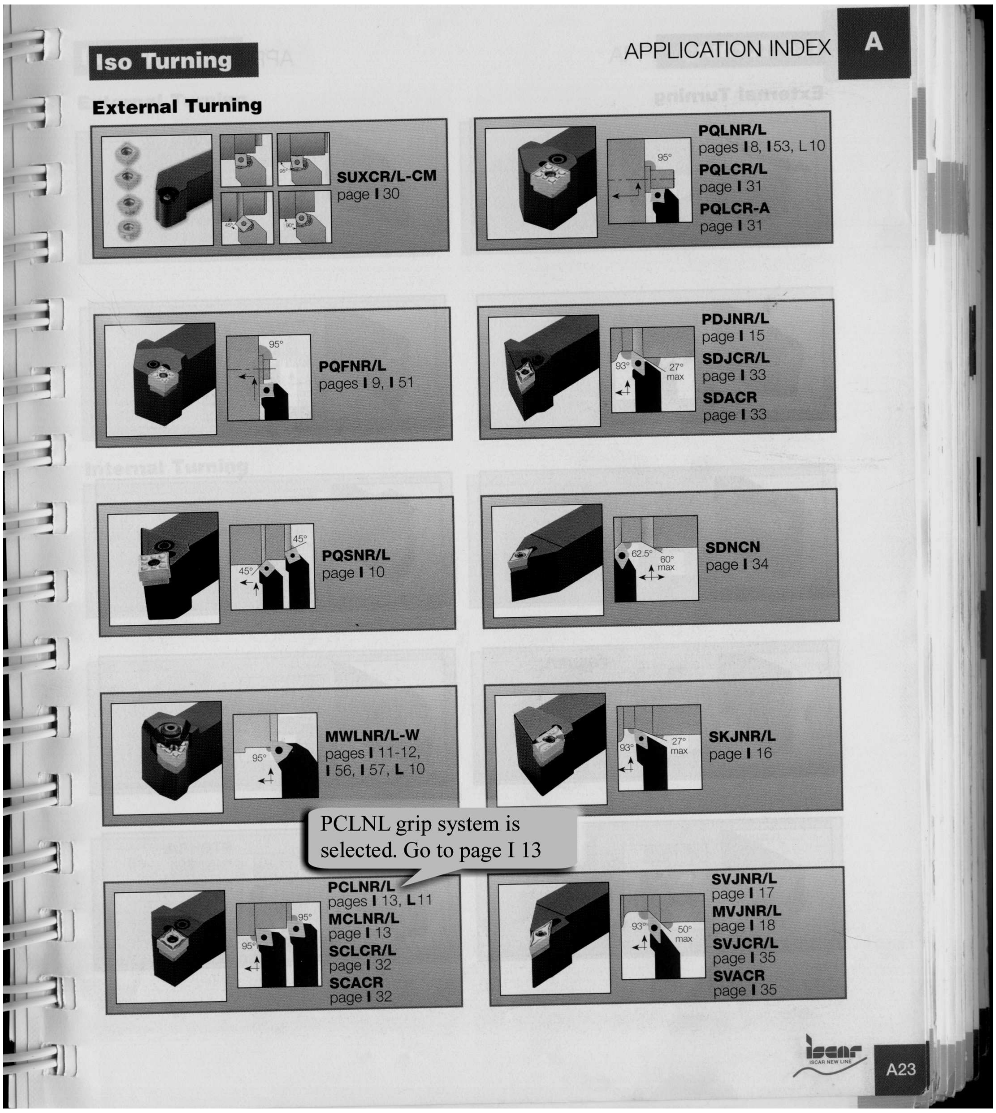
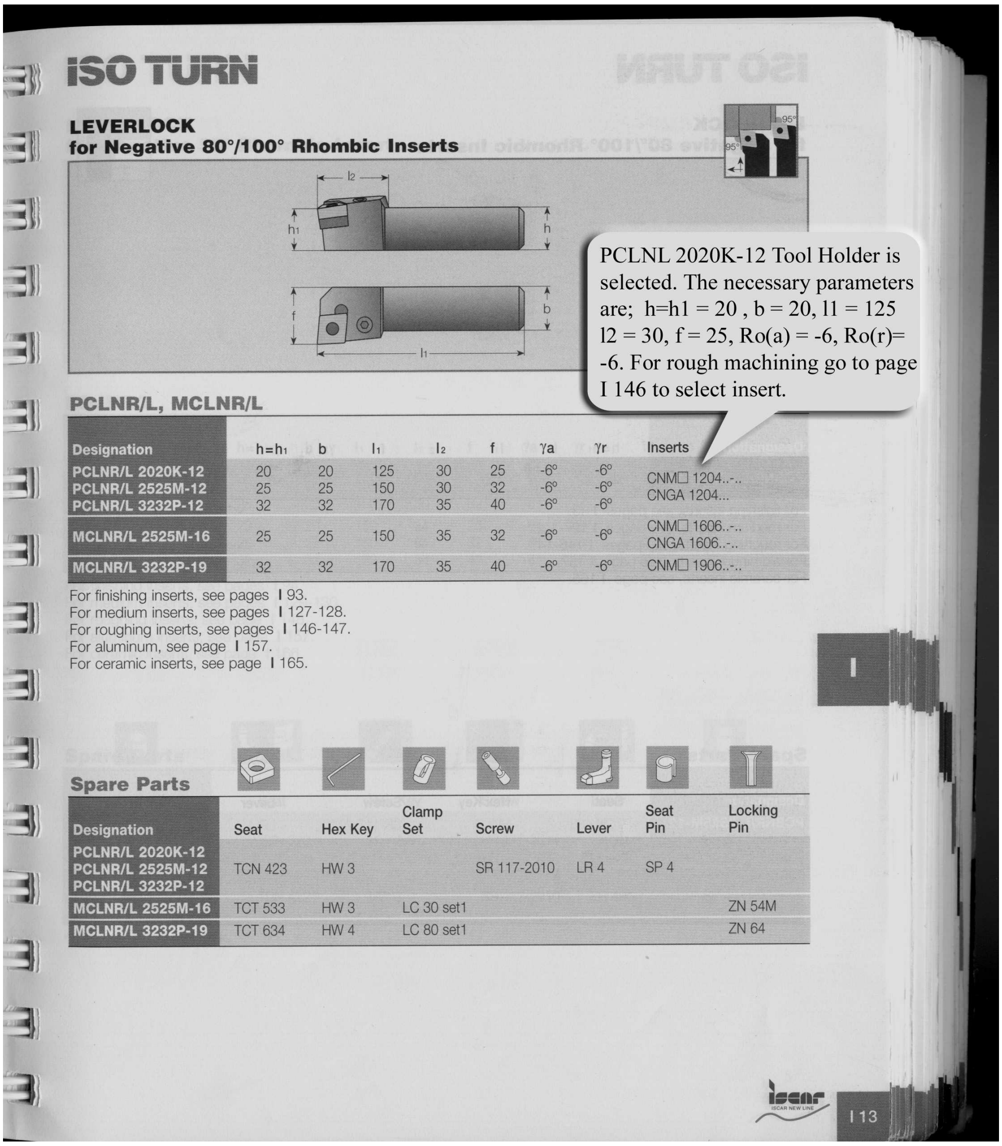


Figure A.21: Grip system selection from catalog.

Holders on page L 11 are for exchangeable heads. The standard tool holders are in page I 13. (Figure A.22) After filling the "Manufacturer" and "Grip System" in Figure A.24, The tool holder model is selected "2020K-12" and filled with the "Tool No" and "Turret No" parameters. "Tool Holder Parameters" in Figure A.24 is filled with respect to the parameters in page I 13 which are; h=h1=20mm, b=20mm, l1=125mm, l2=30mm, f=25mm, Ro(a)=-6deg, Ro(r)=-6deg



ISO TURN

LEVERLOCK
for Negative 80°/100° Rhombic Inserts

PCLNL 2020K-12 Tool Holder is selected. The necessary parameters are; h=h1 = 20 , b = 20, l1 = 125 l2 = 30, f = 25, Ro(a) = -6, Ro(r)= -6. For rough machining go to page I 146 to select insert.

PCLNR/L, MCLNR/L

Designation	h=h1	b	l1	l2	f	γa	γr	Inserts
PCLNR/L 2020K-12	20	20	125	30	25	-6°	-6°	CNM□ 1204..-..
PCLNR/L 2525M-12	25	25	150	30	32	-6°	-6°	CNGA 1204...
PCLNR/L 3232P-12	32	32	170	35	40	-6°	-6°	
MCLNR/L 2525M-16	25	25	150	35	32	-6°	-6°	CNM□ 1606..-.. CNGA 1606..-..
MCLNR/L 3232P-19	32	32	170	35	40	-6°	-6°	CNM□ 1906..-..

For finishing inserts, see pages I 93.
For medium inserts, see pages I 127-128.
For roughing inserts, see pages I 146-147.
For aluminum, see page I 157.
For ceramic inserts, see page I 165.

Spare Parts

Designation	Seat	Hex Key	Clamp Set	Screw	Lever	Seat Pin	Locking Pin
PCLNR/L 2020K-12 PCLNR/L 2525M-12 PCLNR/L 3232P-12	TCN 423	HW 3		SR 117-2010	LR 4	SP 4	
MCLNR/L 2525M-16	TCT 533	HW 3	LC 30 set1				ZN 54M
MCLNR/L 3232P-19	TCT 634	HW 4	LC 80 set1				ZN 64

iscar

I 13

Figure A.22: Tool holder selection from catalog.

Page I 146 can be referred to select insert for rough machining conditions. (Figure A.23) If "CNMM 120408-NM" insert is selected, user has to fill "Rough Machining" in application text and "CNMM" in insert text shown in Figure A.24. The code of the insert is filled in "120408-NM" and workpiece material is automatically entered as the material of designed part. Although user is free to change or assign a new workpiece material for the future tool selections.

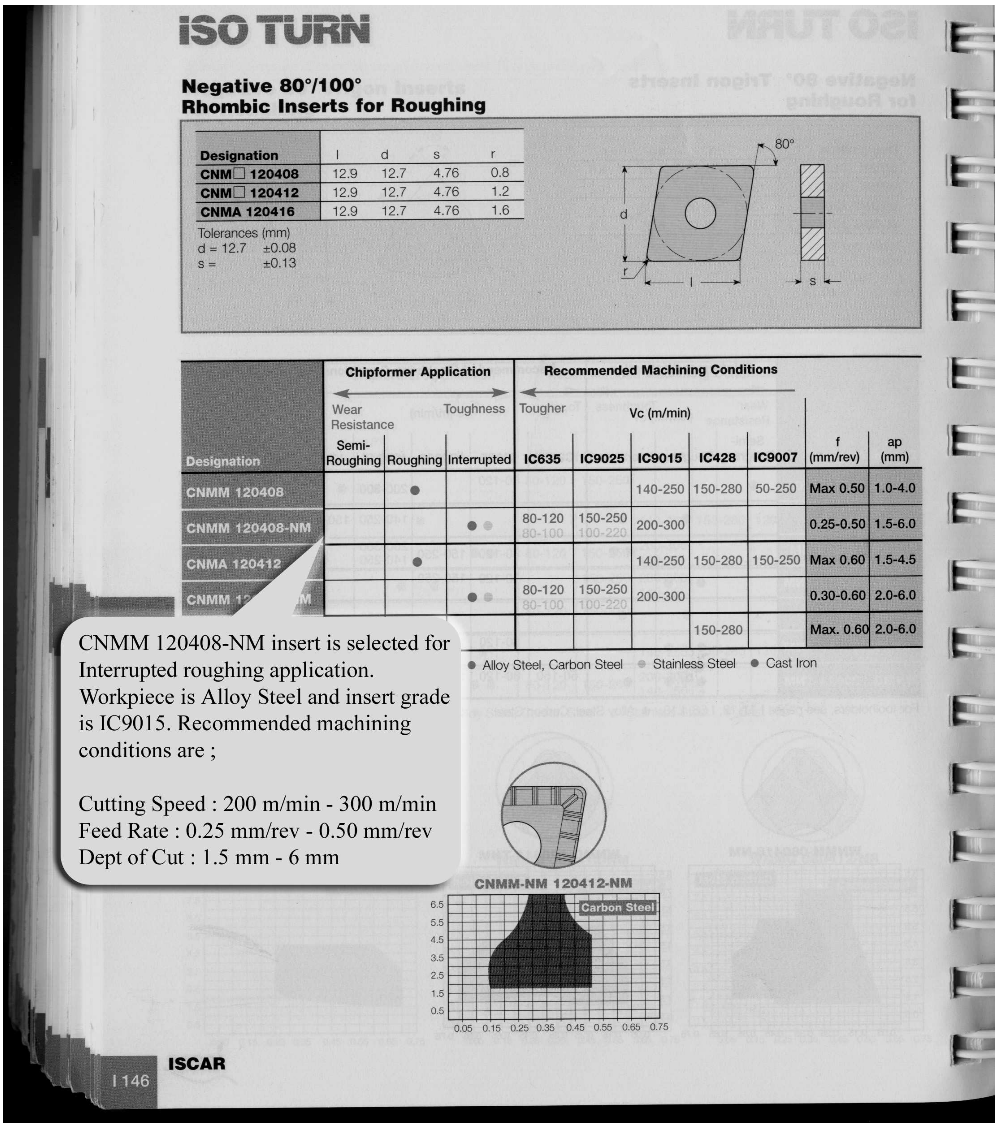


Figure A.23: Insert selection& machining parameters definition.

Finally the grade of cutting insert must be entered for selecting the machining conditions from Figure A.23. The cutting conditions of the interrupted roughing operation can be read from Figure A.23 to fill the necessary spaces for the values of maximum and minimum cutting speed, feed rate and depth of cut.

As it is stated from figure cutting speed values vary from 200 to 300 m/min, feed rate values vary from 0.25 to 0.5 mm/rev and depth of cut values vary from 1.5 to 6 mm for Alloy steel.

After all the necessary parameters are filled. (Figure A.24). "Create" button allows user to save the tool in cutting tool database for future selections.

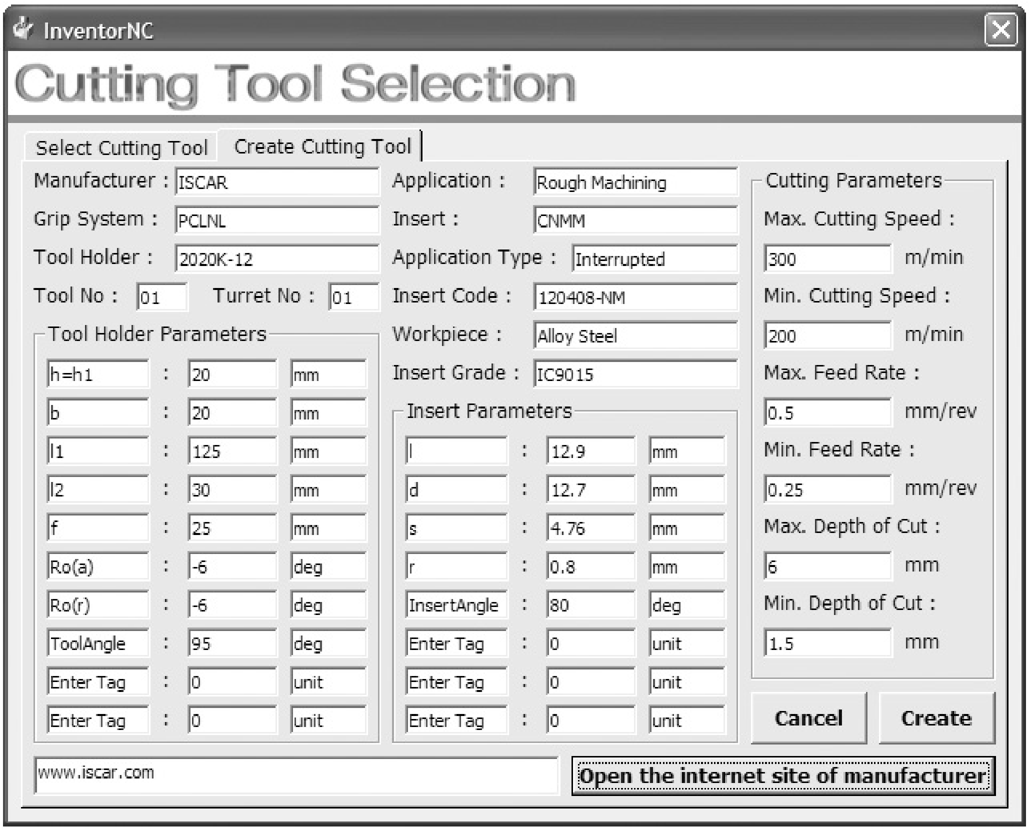


Figure A.24: Cutting tool creation window (Filled).

A.6.3 Part Program Generation

The "NC program generation window" appears by clicking the "Part Program Generation" button in the main menu. Part program generation process can be done by pressing the "Generate NC Code" button in "Operations" menu. (Figure A.26) When the code generation process starts the program asks for the proper cutting tool for each cutting operation. The program displays the generated part program in the "NC Program Generation" window. (Figure A.25)

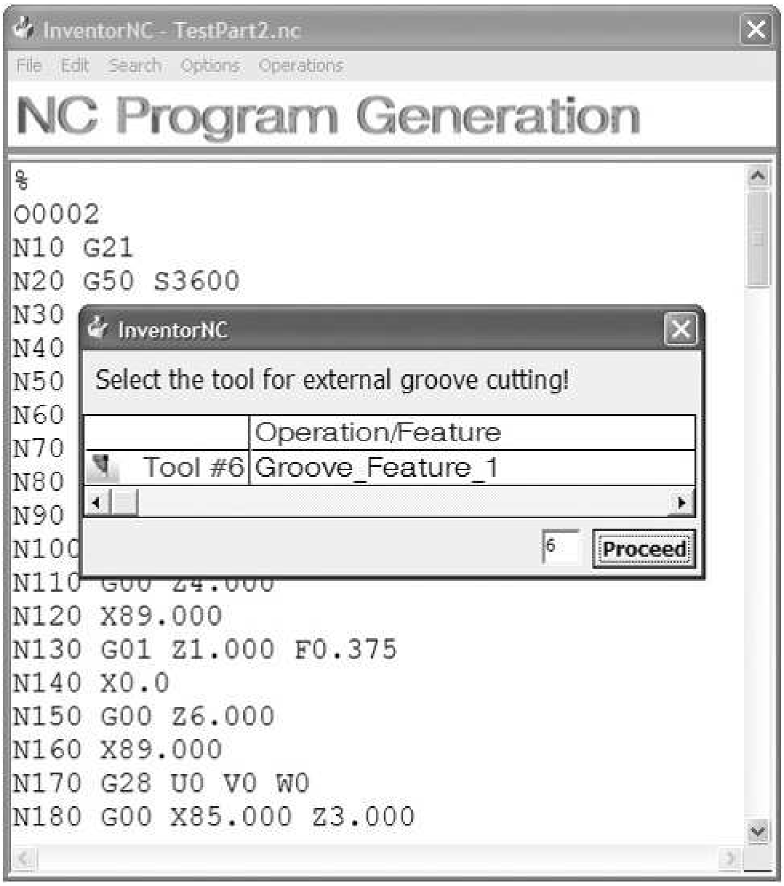


Figure A.25: Part program generation window.

User can easily cut, copy and paste the code using the "Edit" menu commands. Search menu speeds up the search process with the use of "Find" and "Find Next" commands. Besides user can change fonts from "Options" menu and open,save NC programs using "File" menu commands.

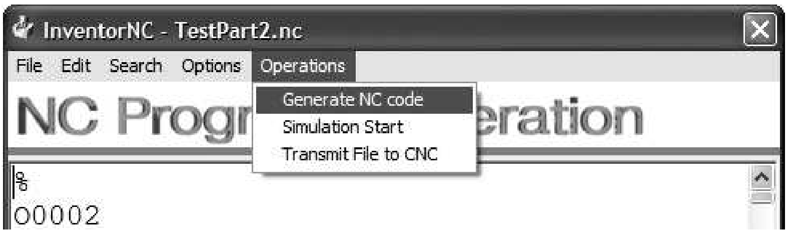


Figure A.26: Part program generation menu.

Part program simulation is done by pressing the "Simulation Start" command in "Operations" menu shown in "NC Program Generation" window. "Tool Path Simulation" window appears (Figure A.27) and simulation process starts in INVENTOR screen. (Figure A.28) The process can be paused, slowed, fastened or cancelled using the command buttons shown in Figure A.27 and the program warns user when the simulation process is finished.

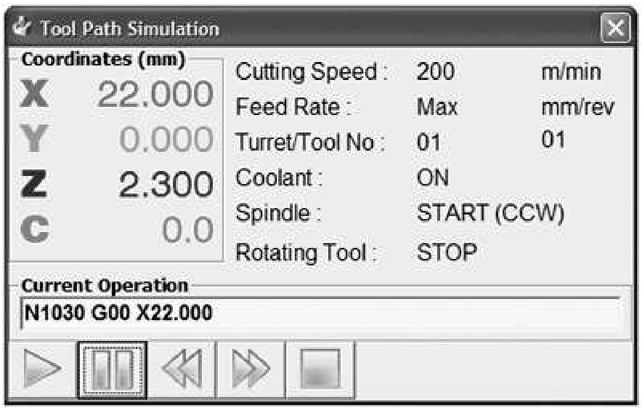


Figure A.27: Part program simulation status window.

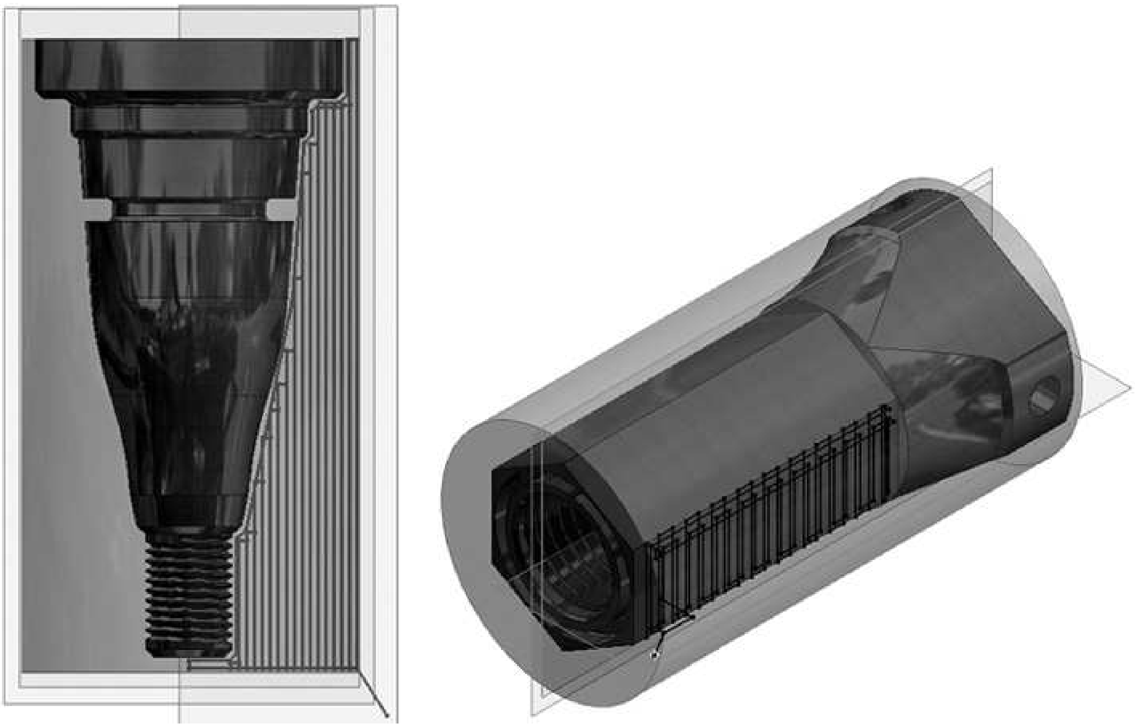


Figure A.28: Part program simulation.

After the generation of part program is completed; the code can be transferred to machine via serial port by using a terminal program. (Figure A.29) User can either press the "Transmit File To CNC" button in operations menu or in main menu to start transfer process.

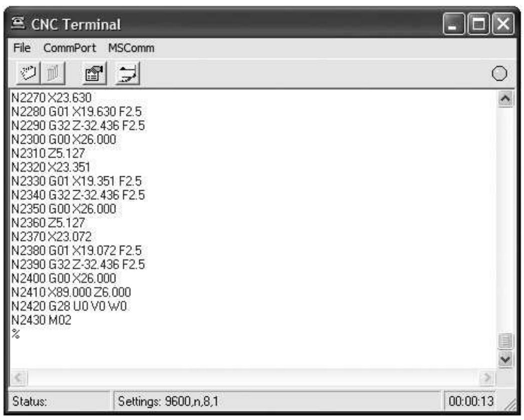


Figure A.29: CNC Terminal.

APPENDIX B

THE PROGRAM STRUCTURE

The developed program is written in Microsoft Visual Basic 6.0 with service pack 6 installed and uses ActiveX automation to run on INVENTOR release 7. ActiveX automation, which is previously known as OLE automation, is a programming interface for INVENTOR and provides developing scripts, macros and third party applications using automation programming environments. With the advantage of automation INVENTOR exposes programmable objects, which can be manipulated by Visual Basic and it is possible to combine the capabilities required by the developed program with INVENTOR's inherent capabilities.

The program contains approximately 30100 lines of codes which are the summation of 24 visual forms and 6 modules. The source code is given in setup cd for future references.

B.1 Forms

The developed forms and their applications are explained below for future reference.

coordFrm : Used to visualize the information about machining simulation.

cuttoolFrm : Used to select and create cutting tool information.

frmInfPath : Used to create path feature.

frmInfStep : Used to create step feature.

frmPathProfile : Used to select the profile of path feature.

frmPP : Used to assign process plan.

frmSDI : Used to generate nc part program.

InfHoleFrm : Used to configure hole feature.

InfPatternFrm : Used to select hole pattern.

InfPocketFrm : Used to create pocket feature.

InfPocketIslandFrm : Used to create pocket-island feature.

mainFrm : Used to assign the main operations.

millFrm : Used to create milling features.

ncconfigFrm : Used to select and configure CNC machine.

setupFrm : Used to assign setup configurations.

turndesign : Used to create blank and part geometry.

turnFrm : Used to create turning features.

turnmanFrm : Used to assign cutting tool and machining data.

B.2 Modules

The developed modules contains subroutines and used for different application areas. Each one explained briefly below for future reference.

Module1 : Contains subroutines that creates all features using INVENTOR application programming interface (API) library.

Module2 : Contains subroutines of opening, saving and updating operations of part data files.

Module3 : Contains subroutines that saves information and parameters of each feature to program memory.

Module4 : Contains subroutines that generates part program for each operation.

moduleNote1 and moduleNote2 : Contains subroutines of opening, saving operations of nc part program.

B.3 Databases

cnc_database.mdb : Contains database information of cnc machines.

threaddata.mdb : Contains database information of thread standards used for milling operations.

threadlathe.mdb : Contains database information of thread standards used for turning operations.

tool_database.mdb : Contains database information of cutting tool parameters with respect to manufacturers.

B.4 Flow Chart

A brief flowchart of the developed program is shown in Figure B.1 to visualize the workflow for the code structure of the program.

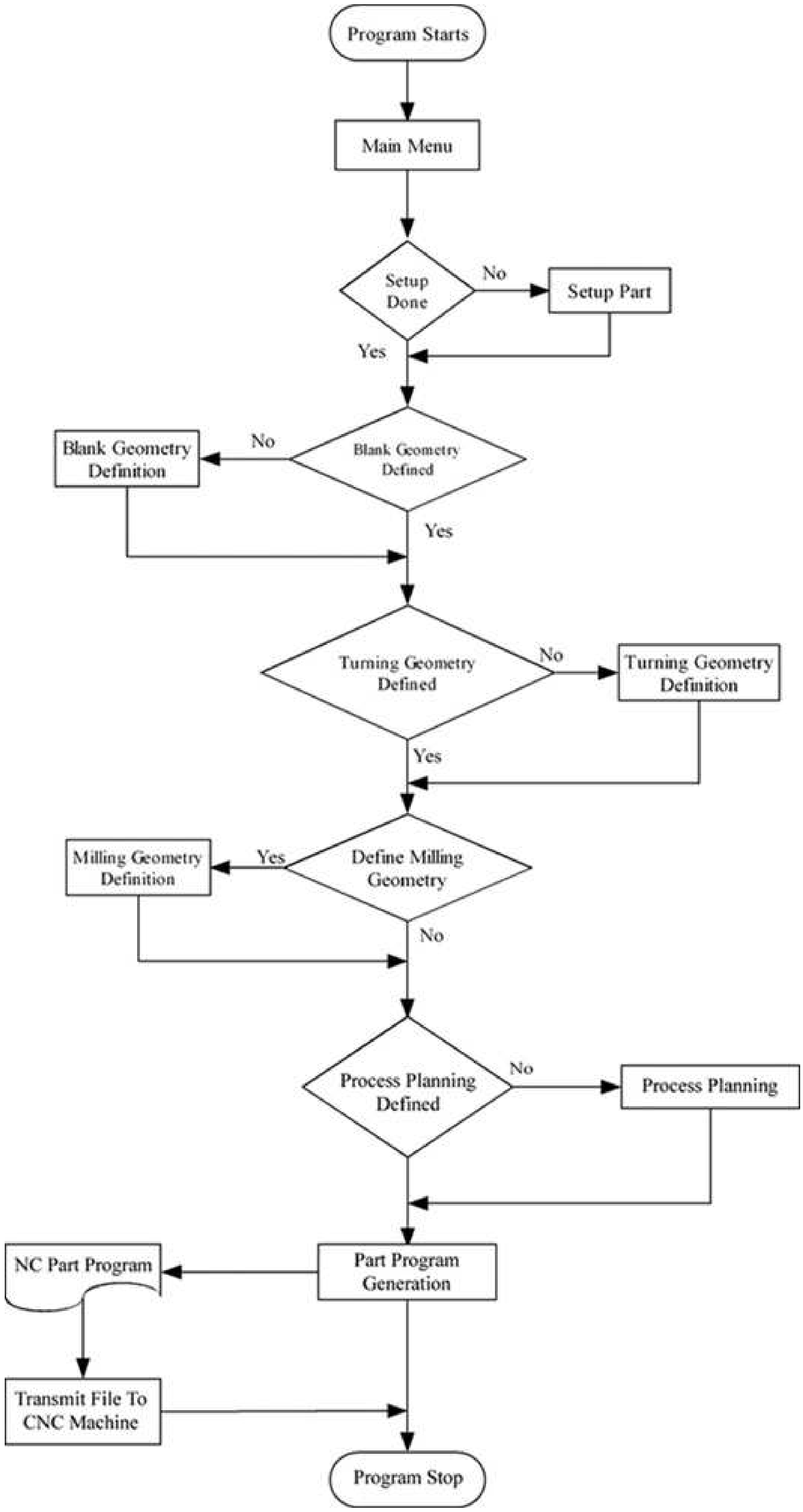


Figure B.1: Flowchart of program.

APPENDIX C

NC PROGRAMMING BASICS

C.1 Details of G-Function

C.1.1 G00 Positioning (Rapid Traverse)

This code is specified when feeding a tool by rapid traverse where X axis at 20 m/min and Z axis at 24 m/min. Therefore is used for approaching the tool to the workpiece or when retreating it after cutting is completed. X, Y, Z and C indicates absolute coordinates where U, V, W and H indicates incremental coordinates.

G00 X(U)___.___ **Y(V)**___.___ **Z(W)**___.___ ⟶ where "___.___" indicates destination point coordinates.

C.1.2 G01 Linear Interpolation

This code is specified when performing linear cutting which involves chamfering and taper cutting. An F code which specifies the feed rate must be used.

G01 X(U)___.___ **Y(V)**___.___ **Z(W)**___.___ **F**___.__ ⟶ G99 mode (mm/rev) is used for feed.

Note: If G98 mode (mm/min) is used then a decimal point cannot be used for declaring feed function.

C.1.3 G02/G03 Circular Interpolation

These codes are specified when performing circular cutting. G02 is used for clockwise and G03 is used for counter-clockwise directions.

Plane designation

- G17 ⟶ X-Y plane designation for circular cutting by Z-axis rotating tool.
- G18 ⟶ X-Z plane designation for circular cutting by turning operation.
- G19 ⟶ Y-Z plane designation for circular cutting by X-axis rotating tool.

A circular command consists of the following 3 factors:

1. Circular arc direction G02 or G03.
2. Circular arc end points of two coordinate values depending on prior declaration of G17,G18,G19. Where X and Z is used for G18.
3. Circular arc radius value.

G02 X(U)___.___ Z(W)___.___ R___.___ F___.__

Note: When specifying a circular arc exceeding 180 degrees, "-" sign have to be specified prior to R value.

C.1.4 G04 Dwell

This command interrupts feed for the length of time designated by address U or P.

G04 U(P)__._ ⟶ where "__._" value designates second.

C.1.5 G28 Automatic Reference Point Return

The tool automatically returns to the machine reference point after moving to the position (intermediate point) specified with X(U), Y(V) and Z(W). G28 moves as the same rapid traverse rate as G00.

G28 X(U)___.___ Y(V)___.___ Z(W)___.___

C.1.6 G32 Threading/Tapping

This command can enable straight/taper/face thread cutting and tapping operations. The work plane of threading operation is always XZ plane because of the turning principle of lathe.

G32 X(U)___.___ Z(W)___.___ F(E)___.__ B___.___ $\longrightarrow$ B is the angle command where; $0 \leq B \leq 360.000$ and F indicates pitch value of thread.

Note: A value of 0.01 mm/rev for F command equals a value of 0.0001 for E command.

C.1.7 G80 to G89 Canned Cycle for Drilling

A canned cycle for drilling is handled by specifying the proper code and the drilling mode continues until G80 (drilling cancel) or G00, G01, G02/03 commands executed.

Generally, the drilling cycle consists of the following 6-step operational sequence.

1. Positioning of the X(Z) and C axes.
2. Rapid traverse to the R point.
3. Drilling.
4. Operation at the hole bottom position.

5. Release to the R point.

6. Rapid traverse to the initial point

Command data for canned cycle for Drilling:

G__ X(Z)___.___ C(H)___._ Z(X)___.___ R___.___ Q___.___ P__._ F___.__ L_ M__

- **G** ⟶ Drilling mode (G80, G83, G84, G85, G86, G87, G88, G89)
- **X(Z) and C(H)** ⟶ Hole position data (incremental or absolute value))
- **Z(X)** ⟶ Absolute point of the hole bottom)
- **R** ⟶ Distance from the initial point to the R point)
- **Q)** ⟶ Each depth of cut in peck drilling)
- **P** ⟶ Dwell time at the hole bottom)
- **F** ⟶ Cutting feed rate)
- **L** ⟶ Repeat frequency of a series of operations)
- **M** ⟶ When a C-axis clamp M command is given)

C.1.8 Drilling Canned Cycles and Operations

C.1.8.1 G80

Canned cycle for drilling cancel.

C.1.8.2 G83 (Canned cycle for Z-axis direction drilling)

Drilling axis : Z axis

Drilling operation : Cutting feed/intermittent feed

Operation at the hole bottom : Dwell

Release operation : Rapid traverse

C.1.8.3 G84 (Canned cycle for Z-axis direction tapping)

Drilling axis : Z axis

Drilling operation : Cutting feed

Operation at the hole bottom : Spindle reverse rotation

Release operation : Cutting feed

C.1.8.4 G85 (Canned cycle for Z-axis direction boring)

Drilling axis : Z axis

Drilling operation : Cutting feed

Operation at the hole bottom : Dwell

Release operation : Cutting feed (Double feed)

C.1.8.5 G87 (Canned cycle for X-axis direction drilling)

Drilling axis : X axis

Drilling operation : Cutting feed/intermittent feed

Operation at the hole bottom : Dwell

Release operation : Rapid traverse

C.1.8.6 G88 (Canned cycle for X-axis direction tapping)

Drilling axis : X axis

Drilling operation : Cutting feed

Operation at the hole bottom : Spindle reverse rotation

Release operation : Cutting feed

C.1.8.7 G89 (Canned cycle for X-axis direction boring)

Drilling axis : X axis

Drilling operation : Cutting feed

Operation at the hole bottom : Dwell

Release operation : Cutting feed (Double feed)

C.2 The Lists of G and M Functions

Finally the list of all the G and M codes with their explanations are given in Table C.1 and Table C.2.

Note: All of these codes listed in tables and the previous explanations are taken from the user's manual of Hitachi-Seiki Seicos LII CNC turning center [13].

Table C.1: List of G Functions.

G-code	**Function**
G00	Rapid traverse, positioning
G01	Linear interpolation
G02	Circular interpolation (CW)
G03	Circular interpolation (CCW)
G04	Dwell
G10	Tool offset value setup
G17	Z-axis synchronize with circular interpolation of X-Y hemihedry
G18	Y-axis synchronize with circular interpolation of X-Z hemihedry
G19	X-axis synchronize with circular interpolation of Y-Z hemihedry
G20	Inch input specification
G21	Metric input specification
G28	Return to reference point
G29	Return from reference point
G32	Thread cutting
G50	Coordinate system/Maximum spindle revolution setup
G80	Canned cycle for drilling cancel
G83	Face drilling cycle
G84	Face tapping cycle
G85	Face boring cycle
G87	Side drilling cycle
G88	Side tapping cycle
G89	Side boring cycle
G96	Constant surface speed control
G97	Cancelling the constant surface speed control
G98	Feed per minute (mm/min)
G99	Feed per revolution (mm/rev)

Table C.2: List of M Functions.

M-code	Function
M00	Program stop
M01	Optional stop
M02	Program end
M03	Spindle forward start
M04	Spindle reverse start
M05	Spindle stop
M08	Coolant start
M09	Coolant stop
M13	Rotating tool spindle forward
M14	Rotating tool spindle reverse
M15	Rotating tool spindle stop + positioning
M18	Turning spindle positioning off
M19	Turning spindle positioning
M23	Chamfering on
M24	Chamfering off
M25	Tailstock forward
M26	Tailstock retract
M43	C-axis coupling
M44	Rotating tool coupling
M45	Rotating tool uncoupling
M68	Chuck close
M69	Chuck open