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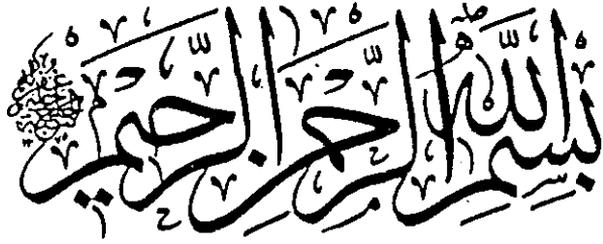
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IN THE NAME OF ALLAH
MOST GRACIOUS AND MOST MERCIFUL.

**A DESIGN WITH FEATURES APPROACH
FOR ROTATIONAL MACHINED COMPONENTS**

by

ARIFFIN ABDUL RAZAK

A Master's Thesis
submitted in partial fulfilment of the requirements
for the award of
Master of Philosophy
of Loughborough University of Technology
November 1991.

Department of Manufacturing Engineering
Loughborough University of Technology

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Loughborough University
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ABSTRACT

A major problem in integrating Computer Aided Design (CAD) and Computer Aided Manufacturing (CAM) arises from the difference in thinking between the design and manufacturing people. Designers think of designing a new product in terms of its intended function whereas manufacturing engineers think in terms of decomposing a product design into a set of manufacturing operations.

Feature Recognition and Designing with Features have been recognised as alternative approaches to the integration of design and manufacturing functions.

In this thesis the second approach has been investigated by developing a feature-based "front-end" to a CAD solid modeller. This produces the geometric representation of the component in terms of manufacturing features and processes, and simultaneously captures this information in a form suitable for an outline process plan.

The objective of the research was to develop software suitable for rotational or turned components and which illustrates the integration of detailed component design and outline process planning.

The software has been developed on an APOLLO-DN3500 workstation, under UNIX and AEGIS operating systems. The system uses the Destructive Solid Geometry (DSG) method of the Designing with Features Approach which is suited to simultaneous design and process planning. Features are represented in a DSG tree structure because this representation can be used directly by feature-based manufacturing systems, where the features correspond to stock removal material volumes.

The software runs interactively and the input data is keyed into the system during a user-friendly interactive session. The material and the shape of the component are first described. The stock bar can be defined geometrically by its length and diameter which are used to generate the cylindrical primitives in the solid model. The workpiece material can be drawn from a list of materials or can be user-defined. The user then interacts with the software by keying an appropriate

values of parameters to describe the machined features of the component.

Description of the stock material and features are written to output files associated with both the model and the outline process planning.

ACKNOWLEDGEMENTS

There were several people who helped me through this enormous project. I would like to express my gratitude to all of them.

Firstly, I would like to convey my thanks to Government of Malaysia and Universiti Teknologi Malaysia (UTM) for giving me an opportunity to carry out this study.

I wish to express my gratitude to my supervisor, Dr. Keith Case, for his guidance and encouragement through the course of this research. His expert guidance, valuable discussions and helpful suggestions have all led to the development of this thesis.

My thanks go to technical staffs, Mr. David Walters and Mr. Robert Doyle for being helpful in discussing some aspects of this work.

For the kind help of "sohabah", I would like to express "JazaKhalLah hu Khairan Kathira" to all of them.

Finally, my sincere gratitude for my wife, Azizah and my children, Afiqah, Afiq and Arifah for their patience and encouragement during the period of my study.

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LIST OF ABBREVIATIONS

1. AI - Artificial Intelligence
2. B-Rep - Boundary Representation
3. CAD - Computer Aided Design
4. CAM - Computer Aided Manufacturing
5. CADOPP - Computer Aided Design and Outline Process Planning
6. CAPP - Computer Aided Process Planning
7. CSG - Constructive Solid Geometry
8. DSG - Destructive/Deforming Solid Geometry
9. FEA - Finite Element Analysis
10. FEV - Faces, Edges, Vertices
11. 3-D - 3-Dimensional
12. 2-D - 2-Dimensional

Chapter 1

CHAPTER 1

INTRODUCTION

1.1 Introduction

Computer Aided Design/Computer Aided Manufacturing (CAD/CAM) is the most promising technique in engineering industries which utilises computers in designing and manufacturing components for increased productivity. The last 20 years has seen the vigorous development of a wide range of computer programs for CAD and CAM to improve the effectiveness and economics of both design and manufacturing functions. A recent survey [SHELLY,1990], [GRANGER,1990] reported an increase in productivity of up to 100% through the use of CAD and CAM systems in companies.

CAD systems were initially used primarily for drafting and storing engineering drawings. Current CAD systems are based on solid modelling, where the solid aspects of an object can be represented instead of 2-Dimensional (2-D) engineering drawing projections. Engineering analysis such as finite element analysis (FEA), stress-strain analysis, moments of inertia etc., can be directly applied using solid modelling.

Currently, these CAD systems represent models in terms of geometric entities such as faces, edges and vertices for Boundary Representation (B-Rep) models or in terms of primitives and Boolean operators for Constructive Solid Geometry (CSG) models. However, these entities do not have any meaning in the manufacturing sense. This is because CAM systems require detailed component information in terms of features such as cylinder, taper, undercut, hole, etc., together with the required manufacturing processes, tolerances and surface finish.

Manufacturing features have been used in process planning, one of the CAM functions [KROUSE,1982], for a long time. Many researchers are trying to interpret the geometric model of an object in terms of these features [BENNATON,1986], [CASE,1986], [LUBY,1986], [DIXON,1987] in order to provide the necessary links between CAD and CAM.

In integrating CAD and CAM systems, the major problem that has been arisen is the difference in thinking between the design and manufacturing people. Designers think of designing a new product in terms of its function whereas manufacturing engineers think in terms of decomposing a product design into a set of manufacturing operations [ECKERSLEY,1988].

As far as the author is concerned, two approaches have been recognised for the integration of these two functions [JOSHI,1988]. The approaches are (i). **The Feature Recognition Approach** and (ii). **The Designing with Features Approach**.

The Feature Recognition approach is necessary when the component description from CAD system is not in terms of features but in terms of the primitives shapes. This method involves recognising patterns of geometry that are features in the model of an object.

The Designing with Features approach provides the designer with a set of manufacturing features as an alternative to the primitive shapes in the solid modellers. These manufacturing features would correspond exactly to those functional features that the designer thinks of when constructing the model of a component. The geometry of the features can be automatically constructed by defining these features parametrically without requiring the user to translate the shapes of the component into geometric primitives.

The consideration of manufacturing features as an alternative to the primitive shapes for integrating the design and manufacturing functions has led the author to use the second approach, that is a designing with features by developing a feature-based "front-end" to a CAD solid modeller. This produces the geometric representation of the component in terms of manufacturing features and processes and simultaneously captures this information in a form suitable for an outline process plan.

1.2 Objectives of the Research

The objective of the research described in this thesis was to develop software which would simultaneously :

- (i). create geometrical specifications of components using a CAD solid modeller, and
- (ii). produce outline process plans for component manufacture.

The research is confined to rotational or turned components; that is the components whose surface to be machined is symmetrical about the axis of rotation.

The major components of the proposed system are illustrated in Figure 1.1.

1.3 Methodology of the Research

The research has been carried out in three main stages :

(i). The first stage of the research was to define a taxonomy or classification of features which is produced by turning operations as illustrated in Figure 1.2, so that these features could be used in the design and process planning system. The feature classification will be discussed in Chapter 4.

(ii). The second stage was to implement an illustration of the approach in software by writing a program in the FORTRAN-77 programming language.

(iii). The final stage involved testing the system by planning the manufacture of a small sample of components. This is described in Chapter 7.

The approach that has been used in this research project is discussed in Chapter 2 and the description of the system is discussed in detail in Chapter 6.

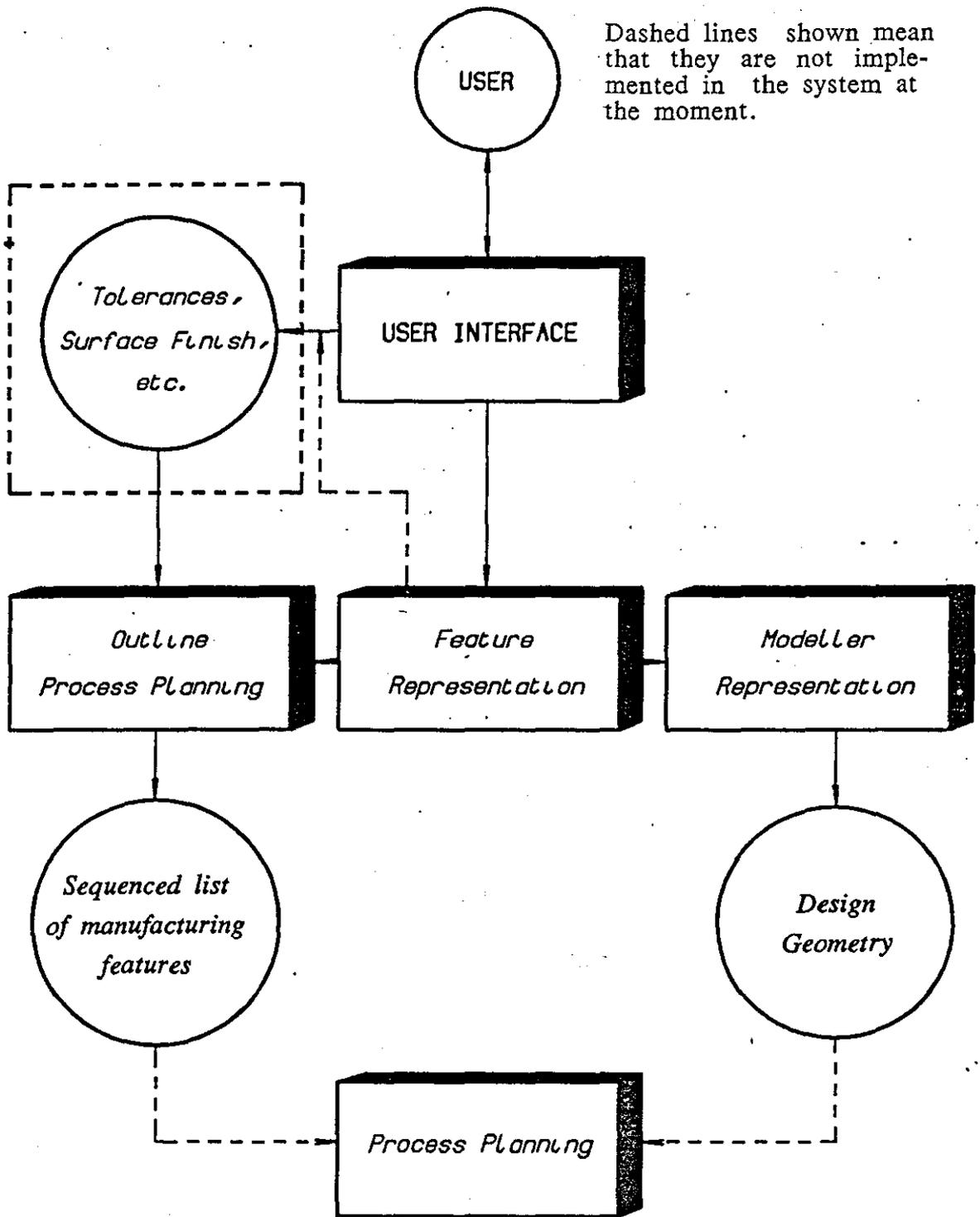


Figure 1.1 : The Major Components of the Prototype System.

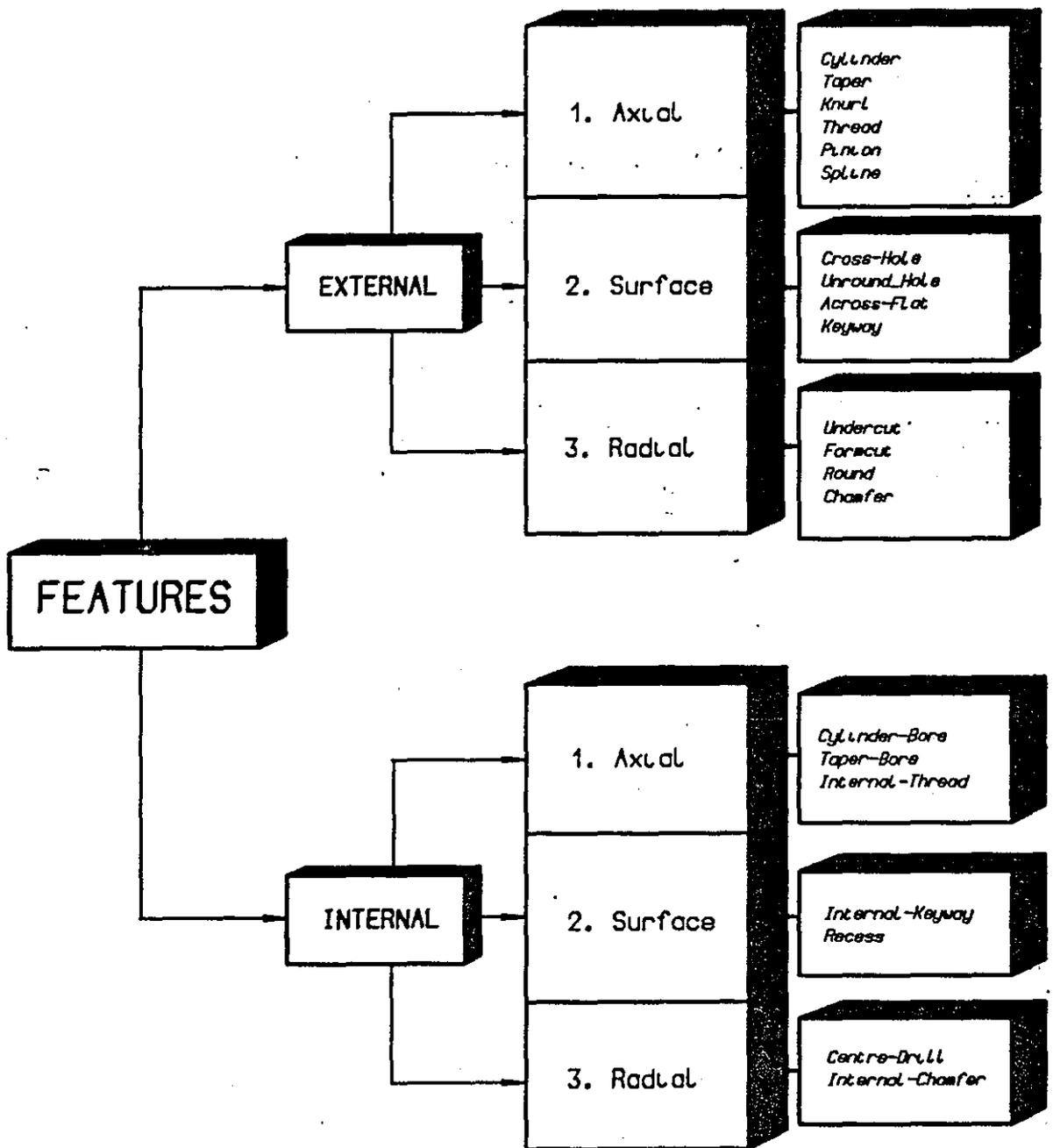


Figure 1.2 : Classification of Features.

1.3.1 The Geometric Specification File

The geometrical specification can be defined as a special program consisting of the parameter values of the features which are used in constructing the geometry in the solid modeller.

In creating the geometrical specification file, the user takes a sketched drawing of a rotational or turned component. These rotational components consist of a number of features, for example, cylindrical surface features, faces, chamfers, tapers, grooves, knurls, threaded features and so on, of different dimensions.

The system asks the user for information about the component material and the shape of the component in terms of length and diameter. Then the user has to decide in which order the features should be machined. The user can re-enter the value of the features in case of misentering. After all the appropriate information has been keyed, the system will ask to save the files and the files will be automatically generated and saved.

The geometric specification file is then used to construct the geometry of the features in the solid modeller. The creation of this file is discussed in Chapter 3.

1.3.2 An Outline Process Plan File

In addition to the creation of the geometric specification file, the system generates another file that contains the outline process plan. Outline process planning can be defined as the determination of a rough draft of processes and tools with the certain feature parameters that are required in order to produce a particular component. This file contains information about the machine tools, cutting tools for machining the feature and the parameter values of the features.

The implementation of the system with examples is described more fully in Chapter 7.

1.4 Organisation of the Thesis

This thesis has been organised into eight chapters. Chapter 2 gives an overview of a literature review covering various types of process planning systems and the relevant research on features in order to solve the problem of gathering the information from CAD systems for manufacturing purposes. The comparison of approaches available and that which has been adopted in this research are also presented. Chapter 3 discusses the different representations of 3-D solid models, the comparison of the modeller storage and the internal representation of the solid modeller.

The feature oriented representation where the features are defined and the role of the features are presented in Chapter 4. The features classification is also described in this chapter. Chapter 5 presents a general consideration for machining operations. This includes the operations which can be performed on lathe machines. The main discussion is in Chapter 6 where the system that have been developed is discussed in detail and this will continue by the implementation of the system in Chapter 7. The test results of the several examples are also presented in this chapter. Finally, the system capabilities, the extensions and future research with the conclusions are presented in Chapter 8.

1.5 Summary

The overall objective of the research is to integrate the design and manufacturing functions through process planning by using a designing with features approach. This approach is one of the approaches available in the literature that has been used by researchers to solve the problem of gathering the information from CAD systems for manufacturing processes. The system that has been developed simultaneously creates the geometrical representation of the component and captures the information related with the component for manufacturing purposes.

Chapter 2

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Process planning generally can be defined as a vital link between design and manufacture functions that involves the selection of operations and processes needed to produce multiple features on a component and transform the raw material into a finished component. Very often process planning also attempts to determine the machine to be used for a particular component.

This function is a difficult and detailed task traditionally carried out manually by a highly skilled process planner who is expected to possess an in-depth knowledge of the wide range of manufacturing processes involved and the capabilities of the machine tools.

Traditionally, process planning involves three separate activities [MARSH,1988] :

(i). The planner looks at the engineering drawing of a component and decides how it is to be made and the type and the size of raw material from which it is to be made,

(ii). To determine machining operations, the planner refers to recommended tools, feeds and speeds for the particular material and machines involved. He/she defines the set-up time and, if required, the time-per-unit quantity for each operation,

(iii). The planner writes all the relevant details on a planning layout (which defines the operations required) and on a method sheet (one sheet for each operation).

The following sections represent the different approaches that have been used in the development of process planning systems.

Because the processes are always associated with the features, a review of relevant works based on features in order to integrate the

design and process planning is essential.

2.2 Process Planning Approaches

Many process planning systems have been reported in the literature [EVERSHEIM,1985]. Three fundamental approaches have been proposed in the development of computer-aided process planning (CAPP) systems [CHANG,1985]. They are (i). The Variant Approach, (ii). The Generative Approach and (iii). The Knowledge-Based Expert System Approach. All of these approaches will be discussed briefly in the following sections.

2.2.1 The Variant Process Planning Approach

The variant approach refers to the method of retrieving a standard plan for similar components. Generally, this approach is based on the concept of Group Technology (GT) to form component families [GROOVER 1984]. For a given component family, the detailed process plan for a new component (a representative member of a family) is generated with the help of an experienced process planner. Plans for several component families are similarly generated and stored on the system's database.

The major attribute of the new component is that it must be in the same family as a component that has previously been planned. The retrieved process plan can be modified based on the characteristics of the new component. Examples of such systems are AUTOCAP [MIDANY,1979], DCLASS [SCWHARTZ,1982] and MIPLAN [LESKO,1985].

The advantages of process planning systems using this approach are that they are useful for producing components within a limited range of component families and also can be implemented in industry fairly easily [GROOVER,1984]. However, they do not truly solve the problem of automating the process planning function since they still rely on human process planners to develop representative process plans and review them for possible new applications.

2.2.2 The Generative Process Planning Approach

The generative approach refers to the methods of generating a process plan automatically for a new component rather than retrieving previous process plans. This approach creates a new process plan for each component from basic machining knowledge based on the characteristics of the component and the machine to be used.

In other words, these generative process planning systems utilize their own built-in logic to select and sequence of machining operations necessary to produce a component. Therefore a new, complete and up-to-date process plan is generated for every new component with little human intervention [CHANG,1985]. ICAPP [ESKICIOGLU,1981], AUTAP [EVERSHEIM,1983] and GCAPPS [PANDE,1988] are examples of these systems. It is estimated that generative process planning systems have the potential to reduce the planning labor costs by 60% [HARVEY,1983].

2.2.3 The Knowledge-Based Expert System Approach

Recently, many researchers have been attempting to use the Artificial Intelligence (AI) based expert systems technique for automating the reasoning activity in CAPP. This technique is seen to take the place of the traditional generative CAPP approach for the following reasons [WANG,1987] :

(i). The modification to the facts and decisions logic in the traditional generative CAPP systems are extremely difficult because the data and logic are mixed in one program. By using the expert system technique, knowledge can be organised in three separate levels: (a). facts, (b). rules, and (c). control strategy. This allows users to make extensive modification to the program.

(ii). Traditional generative CAPP systems often used decision trees or decision tables for simple decision making processes. However, process planning is far more complicated than these tables allow. Expert systems are able to organize knowledge in a certain level of intelligent reasoning.

(iii). Process planning is a complex task which requires considerable experience and knowledge. Current variant and generative systems can not accumulate knowledge so that they require a significant amount of supervision by an experienced process planner. Expert CAPP systems may be developed to accumulate knowledge, such as manufacturing knowledge, their sequencing and required cutting tools.

GARI [DESCOTTE,1981], SIPP, SIPS [NAU,1985] and MACHINIST [HAYES,1986] are examples of systems based on expert systems ideas.

2.3 Feature Approaches

Features are always related to the manufacturing processes and processes are represented in terms of process planning in order to produce the components. The features will provide the starting point for the process planning activity and allow the operation selections to be based on them [HERBERT,1990]. Recently, the concept of using features for design and manufacturing has gained much attention and research interest [DIXON,1987], [DIXON,1988], [FALCIDIENO,1989], [JOSHI,1990].

As mentioned earlier in Chapter 1, two approaches that are currently being considered by researchers in order to integrate the design and manufacturing functions, will be discussed in the following sections and the techniques that have been adopted in this research project are also presented.

2.3.1 Feature Recognition Approach

The feature recognition approach is an approach where the designer first uses a conventional solid modeller to create a model of a component. This model is then submitted to a post processing operation, which tries to fit a library of predefined features that the computer has been taught, and can effectively "understand" against the given geometry [Figure 2.1]. From this a representation of the component can be constructed that has a higher level of information than the purely geometric representation used in the modeller [WOODWARD 1989].

Generally, three steps should be considered in the feature recognition approach [SAKURAI 1988]. After the model of a component has been created, this solid model is searched for each type of feature. The face sets of recognised features are put into a list of recognised features. The second step is checking whether any of the recognised features are part of more complex recognised features. If so, it is discarded from the list of recognised features. Finally, when a feature is identified, a solid model corresponding to the

feature is combined with the original solid model to result in a new solid model. These three steps are iterated until no additional features are recognised.

The techniques used for feature recognition can be classified into five groups as mentioned in [JOSHI 1988] :

- i. Syntactic Pattern Recognition Technique,
- ii. Decomposition Technique,
- iii. Constructive Solid Geometry (CSG) Technique,
- iv. Expert System/Rules of Logic,
- v. Graph-Based Technique.

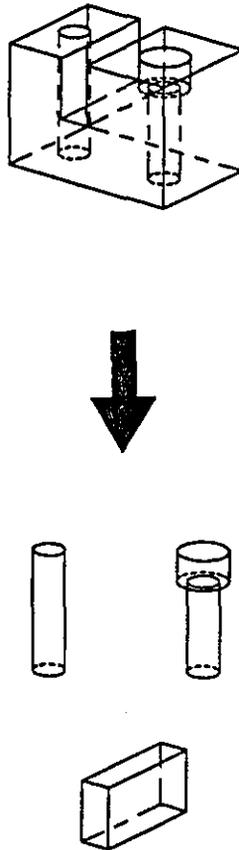


Figure 2.1 : The Feature Recognition Approach.

2.3.1.1 Syntactic Pattern Recognition Technique

This technique consists of three elements that are pattern primitives, the structural rules for combining primitives and the techniques of classification.

This technique recognises the feature in terms of simple subpattern and relations among subpatterns. This decomposition is applied recursively to subpatterns until the simplest subpatterns called pattern primitives are obtained. In this approach the structural information about a pattern, that is the relations among subpatterns, is analogous to the syntax or grammar of a formal language.

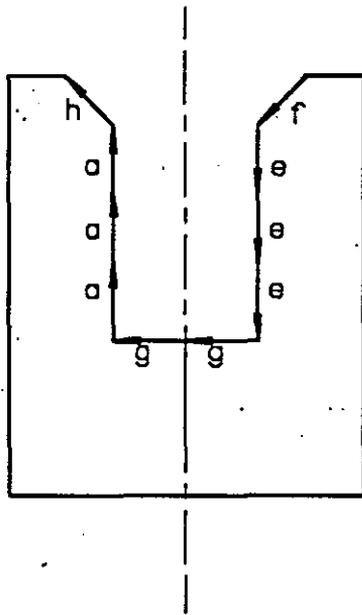
The language that describes a set of patterns in terms of its structural forms and pattern primitives is called a pattern description language. The rules that define valid compositions of primitives into patterns are specified by the grammar of the pattern description language.

Large sets of complex patterns can be described using smaller sets of pattern primitives and grammatical rules.

The required structure rules are the production sets of the pattern grammars that must be defined to discriminate between classes of features.

The classification is done by checking if the grammar can be generated by the pattern grammar using a parser.

This technique is well-described in [JOSHI,1990] and is illustrated in Figure 2.2.



Hole Grammar #3 : Chamfered Hole/Flat,

$G3 = (Vt, Vn, P, S)$

$Vt = \{a, e, f, g, h\}$

$Vn = \{S, A, B, C\}$

$P = \{S \rightarrow fAh,$

$A \rightarrow BifAh,$

$B \rightarrow eCaieBa,$

$C \rightarrow giCg\}$

$L(G) = \{f^n e^m g^p a^m h^n; m, n, p = 1, 2, \dots\}$

Figure 2.2 : A Syntactic Pattern Recognition Technique.

This technique has been used to extract feature information such as machined surfaces from a solid geometric database. Kyprianou [KYPRIANOU,1980] used feature rules to recognize features characterized by presence of material called protrusions and by absence of material called depressions from a solid geometric database and used it to generate group technology codes for classification of features. Jakubowski [JAKUBOWSKI,1982] used syntactic procedures for 2-D pattern recognition to define categories of rotational part families.

Staley et al [STALEY,1983] used linguistic pattern recognition techniques to the bounding contours of longitudinal sections of hole surfaces in order to recognize various types of hole.

Choi [CHOI,1984] described an algorithmic procedure to extract elementary machined surfaces from a 3-D representation of the component for process planning in machining centers. The solid object is represented as a boundary model and manufacturing surfaces identified as holes, pockets, slots and other features. The rules for recognition of elementary machining surfaces are used to classify the feature and generate process planning information based on it.

Srinivasan [SRINIVASAN,1985] presented syntactic pattern recognition for rotational part families. He has suggested the use of tree-grammars for recognition from the 3-D solid model. No procedure or implementation is discussed.

2.3.1.2 Decomposition Technique

This technique has been used to segment a part or the volume to be removed into subvolumes that correspond to form features. A recognition step is needed after the decomposition step to find the semantics of the features.

The decreasing convex hull algorithm [WOO 1982] tries to decompose the volume to be removed into these features. The algorithm is illustrated in Figure 2.3.

The algorithm is to compute the difference between an object and its convex hull recursively, until the null set is obtained (i.e., until the object equals its convex hull). The object can then be represented as a sequence of difference operators, with each being a machining operation. The decomposition is not always useful as it can result in a removal volume that does not correspond to a single machining operation. Also, the stock it assumes can be awkward because it is the convex hull of the initial shape.

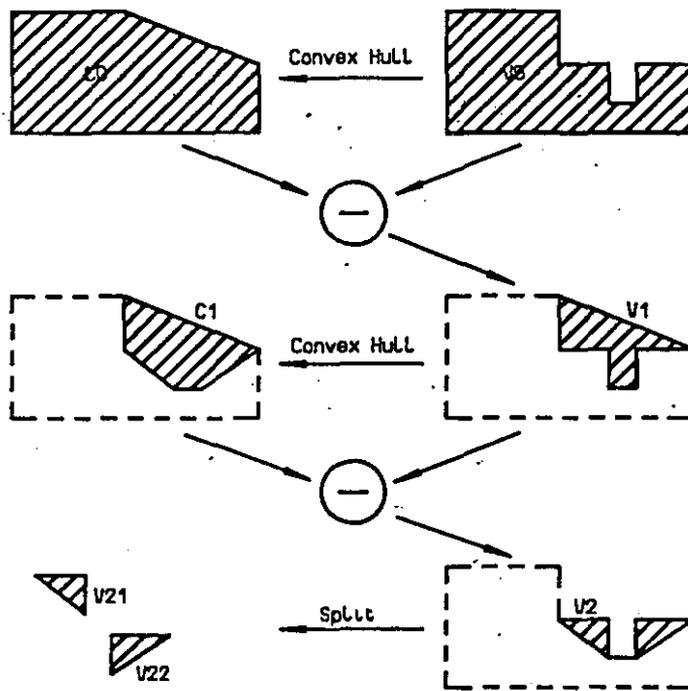


Figure 2.3 : A Decreasing Convex Hull Algorithm.

Another algorithm that has been used in this technique is called delta volume decomposition [FRIDSHAL,1988], and is based on creating the volumes to be removed by machining by using features. The basic idea of this algorithm is to decompose the volume to be machined into a set of disjoint non-overlapping volumes called delta volumes. The delta volumes are created by using the feature volumes from the features, intersecting the feature volumes with the volume to be machined, subtracting the delta volume from the volume to be machined and repeating until the process terminates.

2.3.1.3 Constructive Solid Geometry (CSG) Technique

In order to recognize features from CSG data structures the basic problem that must be overcome is that CSG trees are non-unique as shown in Figure 2.4. In other words there are many ways to define the same form feature with Boolean operations on primitives and these primitives can be scattered around within the tree [CAMI- GM,1988].

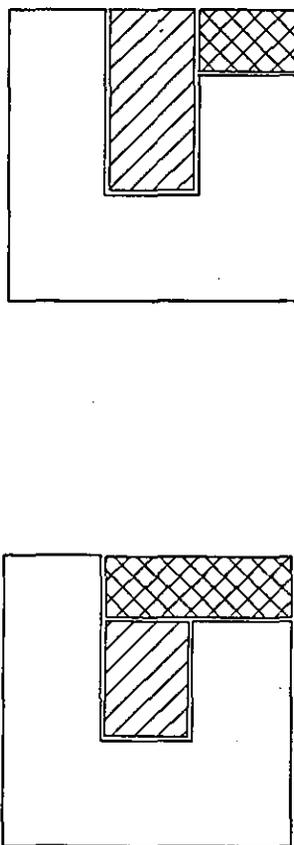


Figure 2.4 : Non-Uniqueness of the CSG Tree.

Two algorithms to recognize form features in CSG trees were found in the literature.

A CSG representation has been used by Lee and Fu [LEE,1987] for the extraction and unification of manufacturing features. The primitives in the model are represented by their principal axes in local coordinates as shown in Figure 2.5.

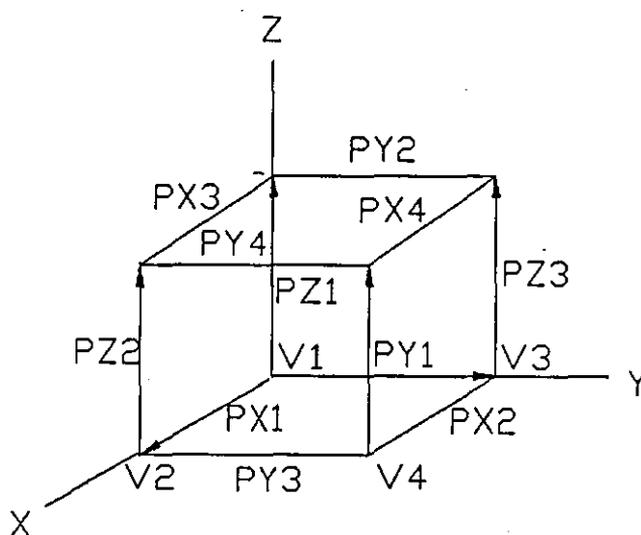


Figure 2.5 : Principal Axes in Local Coordinates.

These principal axes can be collected and clustered according to spatial relationships. In order to solve the problems of redundancy of primitives and operations, the procedure which is based on the concept of unification of a CSG tree are identified and reorganised to reconstruct the tree in a form from which the features can be easily recognised.

Another algorithm proposed by Woodwark [WOODWARK,1988] is matching the CSG tree to shape templates. In this algorithm, features are a pattern represented by a combination of primitives. If the same combination of primitives occurs in a model, then a master version of the feature is instantiated and compared to the volume on the model. If the two models match, the feature has been identified.

2.3.1.4 Expert System/Rules of Logic

This technique is based on using logic rules in recognizing features. In this technique an attempt is made to capture the notion of the feature into some form of rules of logic. The rules are based on searching for certain patterns of elements and relationships until some set of elements can be identified and classified as a feature. The rules are represented in the form of IF-THEN statements. IF $A_1, A_2, A_3, \dots, A_n$ THEN B , represents a rule, where A_1, A_2, \dots, A_n represent conditions and B represents the action when the conditions are satisfied. For example,

IF

Entry surface is Imaginary, and
Exit surface is Imaginary, and
Boundary is Open, and
Exit status is Through, and
EAD's is 4.

THEN

The feature is a Through Slot.

A separate rule to recognize each feature is needed.

This technique has been used by Henderson [HENDERSON,1984a], Henderson and Anderson [HENDERSON,1984b] in order to recognize features from cavity volumes. The features are recognized using rules developed based on geometry and topology of the features. A 3-D boundary representation scheme of solid model is used to create the textual representation of the model in a form to enable application of the feature rules.

This technique was also used to generate Group Technology codes for rotational part families [HENDERSON,1986].

Kung [KUNG,1984] used this technique for extracting 3-D features from a solid geometric database for the purpose of generation of process plans and in the FREXPP system. The domain of the system was limited to components produced by using only the block and cylinder primitives. The system could not recognise any overlapping features.

Hirschtick and Gossard [HIRSCHTICK,1986] also used the expert system technique to develop a feature recogniser in order to extract features in the domain of aluminium extrusions. This system was limited to 2-D components made up of straight lines and circular arcs. The features such as thin walls dividing hollows, knife edges, hollow sections and so forth were selected since the feature recogniser formed a part of a design advisory system.

Many other researchers [BOND,1988],[KENNINGTON,1988] have also used expert system technique in the development of feature recognisers for process planning operations.

2.3.1.5 Graph-Based Techniques

This technique has been pioneered by Joshi [JOSHI,1987] and Joshi and Chang [JOSHI,1988] based on the boundary representation of the component. In this technique, the boundary representation of the component is transformed into an Attributed Adjacency Graph (AAG), that is a graph with attributes assigned to each arc.

An AAG is defined as $G = (N,A,T)$, where N is the set of nodes, A is the set of arcs, T is the set of attributes assigned to the arcs in A . Each face of the part is represented as a node, and each edge or face adjacency is shown as an arc. If the two adjacent faces form a convex angle, the attributes are assigned as "1" and "0" if the angle is concave. Figure 2.6 shows an example of AAG of a component.

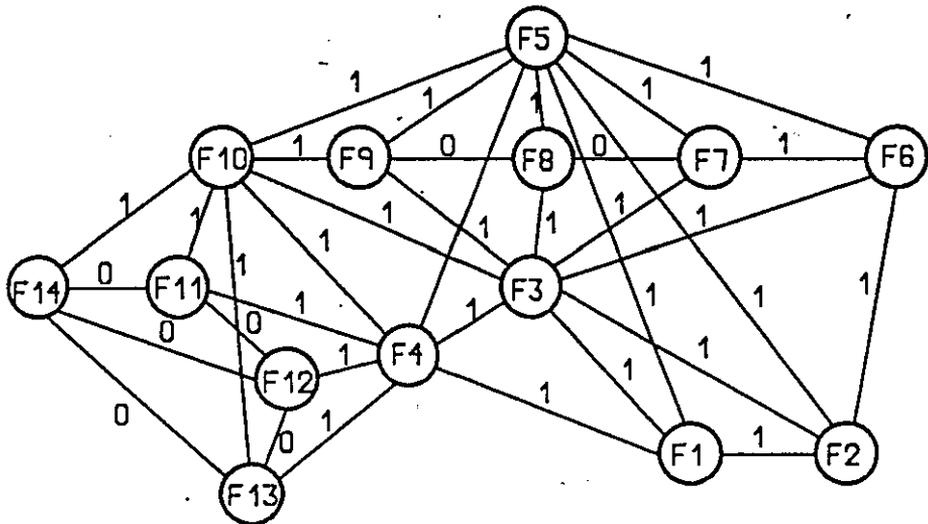
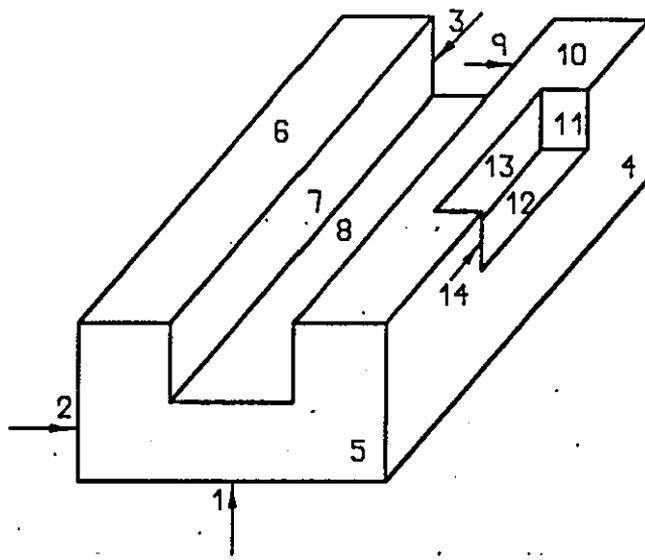


Figure 2.6 : An Example of AAG of a Component.

2.3.2. Designing with Features Approach

Recently, the concept of designing with features, or feature-based design has been proposed by [LUBY,1986], [PATEL,1988], [SHAH,1988b], where this approach is a process in which mechanical parts are specified in terms of their constituent parameterized features [CLARK,1987]. Designing with features for manufacturing involves the use of shapes related to manufacturing processes [LUBY,1986].

Designing with features has allowed designers to create solid models of objects directly from features, selected from choices available to the designer on the screen [Figure 2.7].

In this approach, the feature base is built up in parallel with the CAD solid modeller database, and the CAD solid modeller becomes just a means for displaying the geometry on the CAD workstation screen.

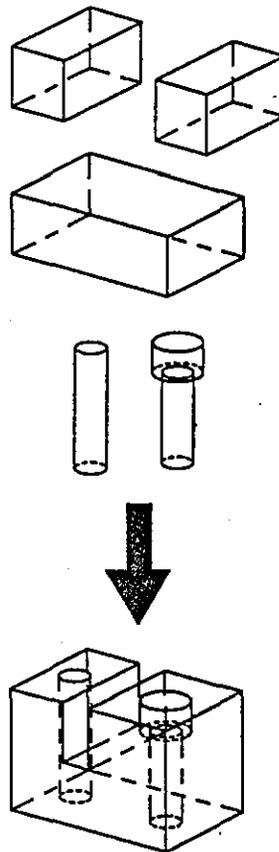


Figure 2.7 : Designing with Features Approach.

Several techniques using this concept have been found but because of major differences between them, they are classified as below :

- i. Destructive Solid Geometry (DSG),
- ii. Compositional Feature Models,
- iii. Features Databases Unassociated with Geometric Models.

2.3.2.1 Destructive Solid Geometry (DSG) Technique

This technique has been explored by Arbab [ARBAB,1982] where he defined a DSG as "*a methodology for describing mechanical parts and assemblies through a sequence of operations resembling those of manufacturing*". This technique of the modelling of stock material removal operations is in principle quite similar to that in CSG which will be discussed in Chapter 3, in which all the operations are of "DIFFERENCE" operation type.

In other words where the initial stock of material is taken as the starting point, the finished component is produced by Boolean subtraction from this stock material in the solid modeller.

A few systems have been developed using this technique. For example, the systems that have been developed by Li et al [LI,1991] and Case et al [CASE,1986]. Li et al proposed a two stage feature-based design concept in 2-Dimensions (2-D) and 3-Dimensions (3-D). This concept is to ensure that the features are machinable, and the construction of features should not only consider by their names but should also consider their shapes and the influence of process knowledge.

Case et al developed a method of removing the volume in a solid modeller in order to produce the component. This was done by creating a feature-based "front-end" to a CAD solid modeller which enabled designers to express their concepts in terms of manufacturing processes and at the same time associating manufacturing methods with the feature definition. This concept is shown in Figure 2.8.

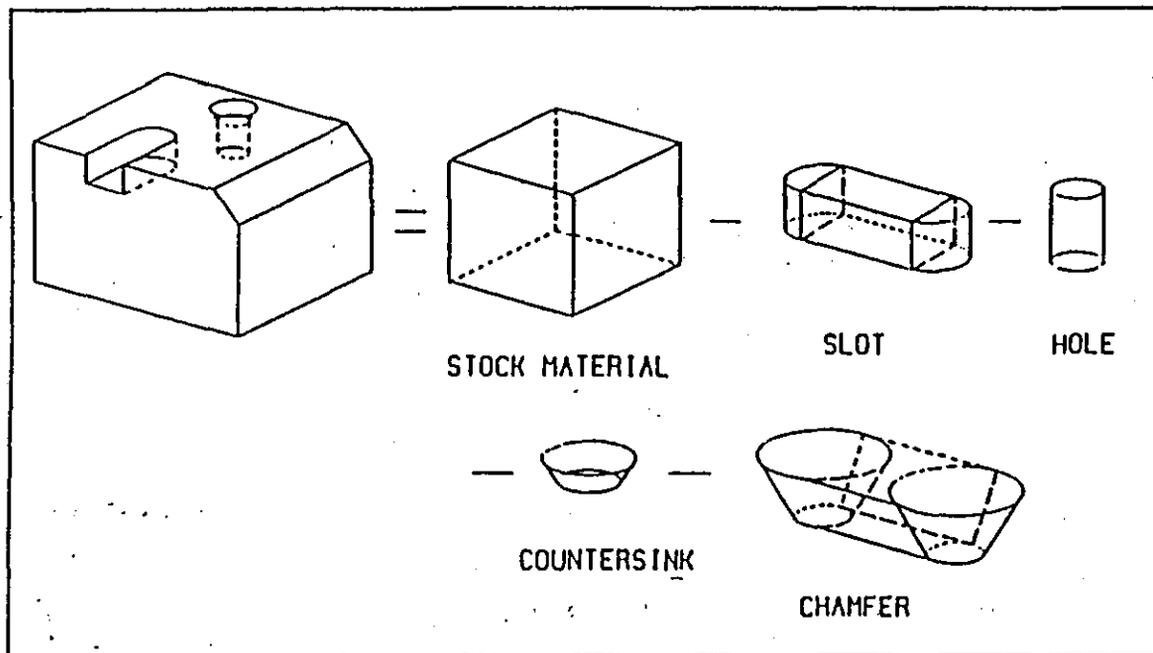


Figure 2.8 : The Destructive Solid Geometry Technique.

2.3.2.2 Compositional Feature Models Technique

In this technique, the features can be constructed and combined together using UNION and DIFFERENCE operations in order to build up the final representation of the component. Figure 2.9 shows the technique of compositional feature models.

This technique does not required the initial stock of material as the starting point in manufacturing processes.

Some of the examples used this technique are the work of Luby et al [LUBY,1986], and Domazet and Manic [DOMAZET,1990].

Luby et al applied this technique for aluminium castings and the resulting program called CASPER employed macro-features such as boxes and L-Brackets together with co-features such as holes, bosses and ribs for use in creating the components.

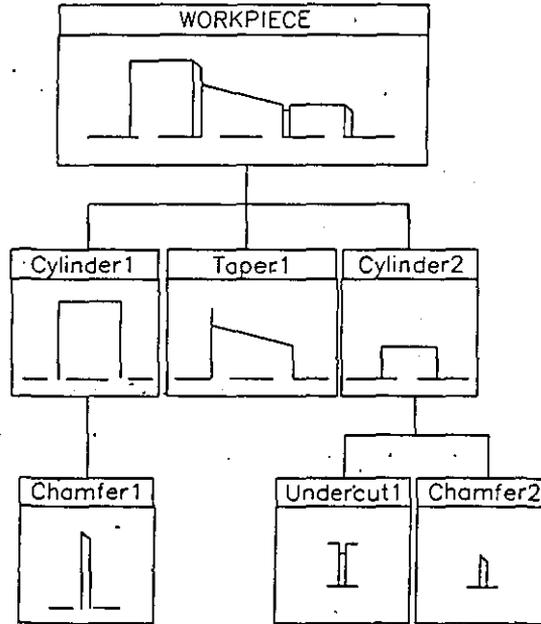


Figure 2.9 : The Compositional Feature Models Technique.

Domazet et al used a collection of form features to create the component. They named their system as CADROT which uses the rotational features' such as cylinders, cones, faces and threads. The details of this are discussed in [DOMAZET,1990].

2.3.2.3 Feature Databases Unassociated with Geometric Models

This technique was employed to input product data for process planning systems. Users input feature information textually using a customized syntax.

Some recent examples of this approach are input systems for GARI [DESCOTTE,1984] and systems discussed in [MILL,1984] and [HUSBANDS,1987].

Since these systems only select processes to create a plan, the reasoning process can be driven by high level parameters, such as feature type, generic parameter values, tolerances and attributes. A geometric model of the component is not required in this technique.

2.4 Other Research on Features

2.4.1 CAM-I Research on Features

Computer-Aided Manufacturing International (CAM-I) is a non-profit organization interested in improving the link between engineering design and manufacturing. The aim of the CAM-I process planning project is to develop a complete product definition or feature information from a process planning perspective. The CAM-I form features are separated into three groups namely sheet features, non-rotational or prismatic features and rotational features. 45 feature classes, 161 individual features, and an additional 50 attributes, notes and miscellaneous terminology were identified. Additional to these are record management data, material data, specification data, engineering notes, relationships, tolerances, constraints and other characteristics. The concept of features is used as the basis for developing automated process planning system in order to provide a part information "language" for implementing Computer Integrated Manufacturing (CIM).

CAM-I project has presented a conceptual approach for incorporating features into manufacturing process planning functions in order to identify rules for creating and defining features and list their process planning requirements.

2.4.2 Features in Standards

The need for capturing a complete product definition information was identified in standards such as PDDI and PDES.

The Product Definition Data Interface (PDDI) was the first attempt at defining a complete product definition information. This study was carried out by the McDonnell Aircraft Company for the United States Airforce Integrated Computer Aided Manufacturing (ICAM) program. The PDDI feature hierarchy is organised into four classes namely machined, turned, composite and sheet metal. The feature entities in these four classes reference needed topological and geometric entities and they are referenced by tolerance entities.

The Product Data Exchange Standard (PDES) is a project begun in 1984 by the Initial Graphics Exchange Specification (IGES) Organisation to develop a mechanism for the exchange of complete product models. This work was performed in conjunction with the establishment of an international Standard for Exchange of Product Data (STEP) under the International Standards Organisation (ISO). PDES is aimed at specifying an approach to communicate a complete product model with sufficient product knowledge in the computer based environment.

Features in PDES have only just been finalized for Version 1.0. The scope is limited to form features as well as tolerances, geometry, topology and assemblies.

PDES groups features into geometric or topological entities (called explicit features) or procedures for augmenting the geometric or topological model (called implicit features).

Implicit features can either be "standard" volumes that are added or subtracted from the pre-existing shape (depressions, protrusions and passages) or operations that modify the geometry (bending, stretching) or area features (usually unevaluated attributes applied to a surface area, e.g. threads, knurls, splines, gears). Implicit feature representation includes specification of the order in which the operations are to be performed.

In this standard, the definition of the feature bound an limit on the extent of a feature or declaration of its immediate neighbours is also included in the conceptual model. Replications of features are represented as a feature and a transformation. Pattern features are represented as a base feature and a layout rule. Compound features are represented as a lists of simpler features of which they are composed.

2.4.3 Feature-Based Commercial Systems

These systems allow designers to work with features in creating a component model. Examples of such systems are Pro-Engineer [PRO-ENGINEER,1987] and Pafec-Imaginer [PAFEC,1990].

Pro-Engineer is a commercial geometric modeller that supports design by features. It combines the DSG technique with the Feature Composition technique. In this system, the user starts with a base solid created by a sweep operation. The system has a small set of pre-defined generic features in a library and the users cannot work outside these generics.

This system classifies features into several groups :

(i). Pick and place are generics such as holes that can be positioned on a face of the base object using two of its edges as references. Sketched features with linear sweeps and revolved features with rotational sweeps are created by sketching the cross sectional profile directly on the part then positioning it.

(ii). Pattern features allow a generic feature to be replicated in a geometric pattern (line or circle). When defining pattern features the user has the option of linking the dimensions of replicated features to the reference feature or to make them independent. In the former option any changes to the reference feature are propagated to replicated features but they cannot be independently modified. The opposite is true for the second option. Any dimension on the part can be related to any other dimension through algebraic or trigonometric equations. Since pre-defined features have built-in parameter inheritance where in the case of a through hole the depth of the hole comes directly from the depth of the block. There does not appear to be any mechanism for validity checking of operations or relationships; the burden is on the user.

As a result nonsense geometry can be created. When pattern features are used and the base object is modified one can get non-generic features and even invalid geometry.

2.5 Comparison of Approaches

The main advantage of the feature recognition approach is its generality where the model could be submitted to several different post processors, each for a different manufacturing process. The designer does not need to commit himself to any one process, and in theory does not even need to know anything about designing for a particular process while he is building the part.

Against this is the problem that if the post processor can not recognise a feature, the information which is passed on to the expert system analysis stage is incomplete, and any decisions or recommendations made will probably not be valid. Given the freedom that the current solid modellers allow the user in methods for constructing a model, it is highly likely that the designer would be able to include at least one feature which would not be recognised or correctly identified.

The major advantage of the designing with features approach is that as the model is built only from features that the system understands, using legal combination operations, it should not be possible for the designer to construct a model whose feature base the computer cannot subsequently evaluate.

The limitations with this approach are that the set of features used in design is not finite and the data management problems are challenging where the tremendous variety and quantity of features data such as generic parameters, relationship; method of positioning, creating, modifying, deleting etc; method for validity checking; method for generating geometric representation when applicable, networking from features and tolerances have to be managed.

The main practical difference between the feature recognition and designing with features approaches is in the amount of knowledge required by the system. For the feature recognition approach, there are an almost infinite number of ways that the user can construct a model, which the system needs to be able to deal with.

For designing with features, however, there is a finite, relatively small number of well defined features and combining operations available for use, which means that the size of the system can be kept within manageable proportions.

It is also would be relatively straight forward to provide the user with a set of manufacturing features as an alternative to the more general primitives of most solid modellers. These features would correspond exactly to those functional features that a designer naturally thinks in terms of when considering a component produced as machining operations. By defining these features parametrically, it would also be possible to construct their geometry automatically, without the user needing to translate the shapes into general system primitives.

Therefore, it was decided to use the designing with features approach.

2.6 Proposed Approach of the Prototype System

Based on the previous work done by [CASE 1986] and the work done in CAM-I research, a system has been developed which implements manufacturing features in design and process planning by using the Destructive Solid Geometry (DSG) technique of designing with features approach.

The details of the system will be discussed in Chapter 6.

2.7 Summary

The DSG technique of designing with features approach has been chosen as the method in trying to integrate design and manufacturing functions. This technique is considered because of close analogy between the representation of the component in a solid modeller with the material removal of many manufacturing processes.

Chapter 3

CHAPTER 3

REPRESENTATION OF 3-D SOLID MODEL

3.1 Introduction

Computer-Aided Design (CAD) began a new era in engineering design. For 20 years, CAD has been interpreted as Computer-Aided Drafting (a computer has been used as a drafting tool) and engineering drawings formed the major portion of representation schemes [MINER,1985].

In recent years, there has been an increasing awareness of the potential of using a computer to model 3-Dimensional (3-D) engineering objects [KOGAN,1986]. This 3-D geometric modelling has been viewed as the solution to the designer's needs. A geometrical model would be used not only for drafting, but also for engineering analysis such as finite element analysis, stress/strain analysis, etc. With a complete component representation, 3-D geometric modelling will become the most important tool in engineering design.

The use of 3-D geometric modelling results in the following benefits : reduced product cost, improved design accuracy giving improved product quality and reliability, and a reduction in lead-times giving increased overall profitability [MARTIN,1985].

3.2 The Design Process

This section presents an overview of the design process which starts with a definition of a need and ends with the production plan.

Design in general terms can be defined as "*a creative operation of products, production processes or more generally of humanity serving system using available materials, products and processes, aiming to achieve a specific function, responding to the demand of the customer and compromising between conflicting constraints such as cost, delivery, delay, feasibility and maintainability*" [AGERMAN,1989], and as "*the means by which solutions are contrived to people's problems and in response to a need.*" [MEGUID,1987]

"The aim of the design function is to translate the functional specification of a product into a product that can perform the desired function(s)." [DELBRESSINE,1990]

The design process can be divided into five phases [KINOGLU,1987] as illustrated in Figure 3.1.

- (i). Feasibility Study Phase,
- (ii). Preliminary Design Phase,
- (iii). Detail Design Phase,
- (iv). Manufacturing Engineering Phase,
- (v). Production Plan Phase.

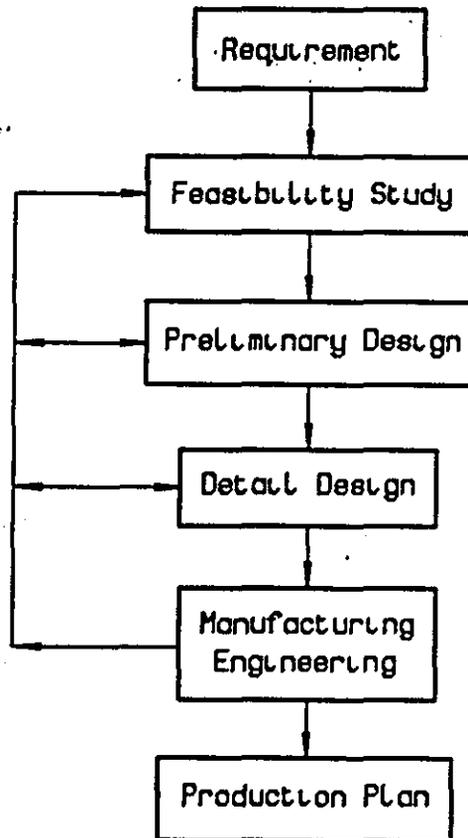


Figure 3.1 Phases of Product Design.

The design process begins with the conceptual work, continues with feasibility study and preliminary design activities, and finalizes with detailed design and manufacturing engineering processes which are used to prepare the production plan. However, each one of the design stages are made up of multiple steps that interact with each other.

3.2.1 Feasibility Study Phase

This is the first phase in the design activity. The purpose of this phase is to achieve a set of useful solutions to the design problem that have arisen from by the design or sale people. This will establish whether the problems can be solved or not.

3.2.2 Preliminary Design Phase

The preliminary design phase starts with the set of useful solutions which were developed in the feasibility study. The purpose of this phase is to establish an overall concept for the project. The resulting preliminary designs are then subjected to the appropriate analysis to determine their suitability for the specified design constraints. If these designs fail to satisfy the constraints, they are then redesigned or modified on the basis of the information gained from the analysis. This will serve as a guide for the detailed design.

3.2.3 Detailed Design Phase

The detailed design phase begins with the design concept evolved in the previous step. In this phase, the components, sub-assemblies or sub-systems are synthesised into the final overall system. The material for the components is defined related with the treatments that are required. The detailed drawings are prepared and reviewed. All the necessary descriptions such as specifications, tolerances on dimensions, clearances, symbols and standard notes are made which are available for manufacturing.

3.2.4 Manufacturing Engineering Phase

The following tasks are performed during this phase :

(i). the manufacturing processes (process planning) are detailed because the problems in the production that cause by design features may be discovered.

(ii). the design of tools and fixtures which are necessary for the manufacture of the component are produced. This is based on the previous step that is process plan.

(iii). the standard times and the quality control system are planned and specified.

3.2.5 Production Plan Phase

In this final phase of the design process, the factory management techniques will be applied to the final design for the manufacture of the product, together with the planning for distribution, consumption and retirement on the product.

3.2.6 Phases of Research Work

Based on the design process as an iterative process [MEGUID, 1987], the main considerations in this research work are in the third and fourth phases that are the detailed design and manufacturing engineering phases. These phases are attempted to give a more opportunity in integrating both functions of design and process planning.

The following sections will describe in general tools that are used in representing the design, that is a geometric modeller.

3.3 Geometric Modelling

The component representation forms a major part of the information needed for process planning. Since the aim of this research is to grasp the information for design and process planning, the component representation should be in a format that understood by a computer.

So, 3-D geometric modelling plays the role in providing the representation of the geometry of a component. A 3-D geometric model is a mathematical representation of the objects in some form suitable for computer-storage and subsequent processing.

Basically, there are three levels of 3-D geometric modelling based on the dimensionality of the representation domain. These levels are wire-frame or line modelling, surface modelling and solid modelling [INGHAM,1989]. Figure 3.2, Figure 3.3, Figure 3.5 and Figure 3.7 display the essential differences in the representational domain of each of the three levels of modelling.

3.3.1 Wire-Frame Modelling

Wire-frame modelling is sometimes called "line" modelling. The object is described by the lines which make up its edges as shown in Figure 3.2. This representation is suitable for visualisation an objects of which all the surfaces are flat. Although this is a simple way to represent a model, it can, as may easily be demonstrated, give rise to ambiguous definitions. Wire-frame modelling is useful to display an uncomplicated objects in minimum processing time but is not really suitable for complex objects. Some draughting systems have wire-frame modelling facilities - PAFEC DOGS-3D is an example [PAFEC, 1986].

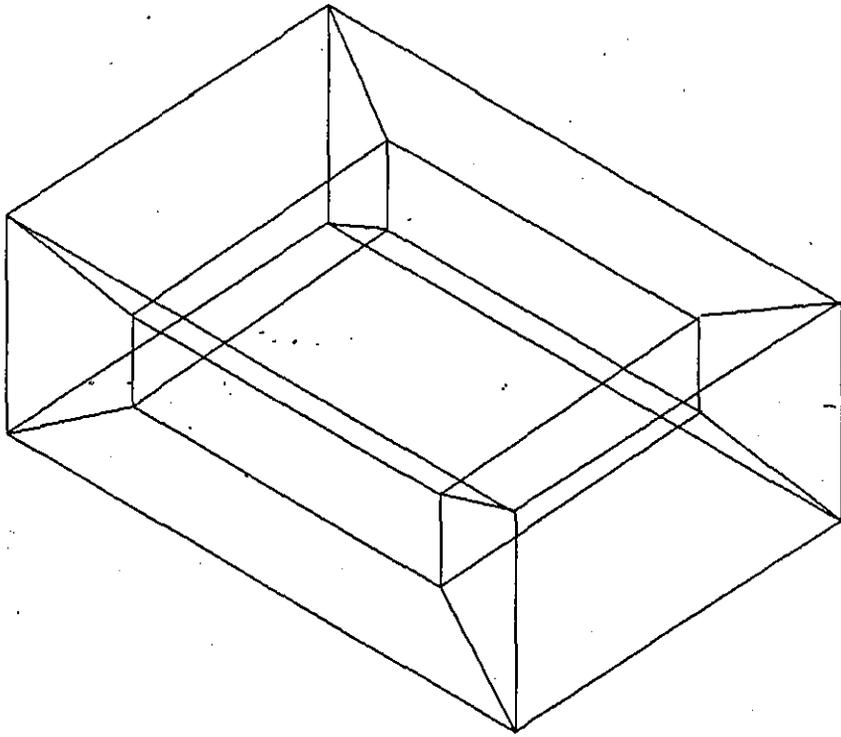


Figure 3.2 : Wire-Frame Modelling.

3.3.2 Surface Modelling

Surface modelling is a considerable advance on wire-frame modelling and defines objects by the surfaces, possibly curved, which bound them (Figure 3.3). These surfaces are built up from mathematically defined patches so that the co-ordinates of all the points on the surface of the body can be determined. An example of a surface modeller is DUCT [DELTACAM,1987].

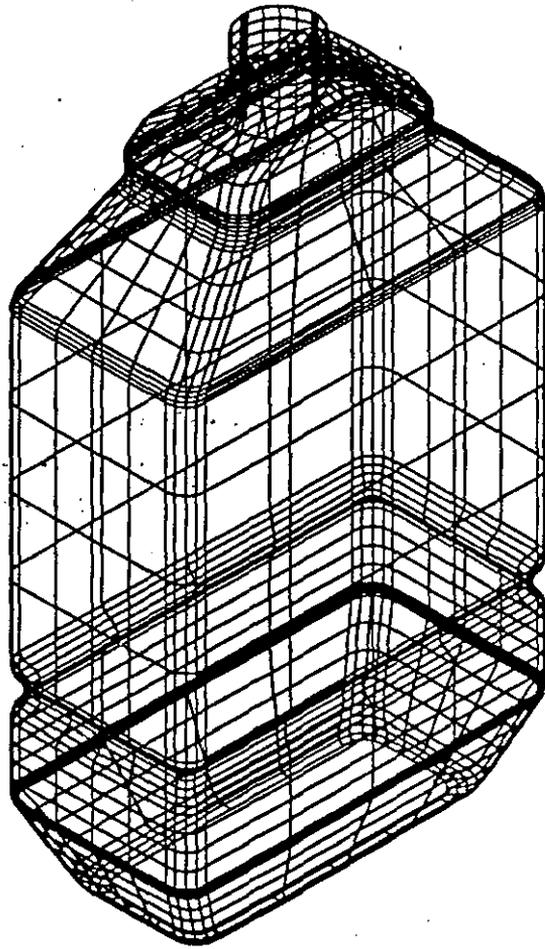


Figure 3.3 : Surface Modelling.

3.3.3 Solid Modelling

Solid modelling is considered by many to be the most advanced form of geometric modelling as it provides a full 3-D solid definition of objects. Unlike the previous two types, this type of modelling takes account of the solid properties of the object.

At present, several techniques exist for representing solid objects in the computer. The two most common techniques are known as Constructive Solid Geometry (CSG), where the objects are built up from standard solid building blocks called "primitives" and Boundary Representation (B-Rep), where the objects are represented in terms of faces, edges and vertices (FEV). These techniques are well described in the literature [REQUICHA,1983]. Both of these representation schemes in solid modelling will be described in detail in the following sections.

3.3.3.1 Constructive Solid Geometry (CSG)

Constructive Solid Geometry is a direct and effective way of representing a solid object model. CSG modellers describe objects in terms of solid primitive shapes which may be combined in a variety of ways. Primitive shapes such as BLOCKS, CYLINDERS, SPHERES, CONES and TORI are illustrated in Figure 3.4.

A CSG modeller represents an object as a binary tree where primitive objects are the "leaves" of the tree and boolean set operations are the "nodes" of the tree. The tree structure of the CSG model is shown in Figure 3.5. The boolean operators used to combined these primitives are union, difference and intersection. These boolean operators are discussed in section 3.4.

A Destructive Solid Geometry (DSG) is a special type of a CSG tree. By using this technique in which to model the material removal operations, only the "DIFFERENCE" operators is used. This type of technique has been used as a basis in this research work. Figure 3.6 shows the tree structure of the DSG model.

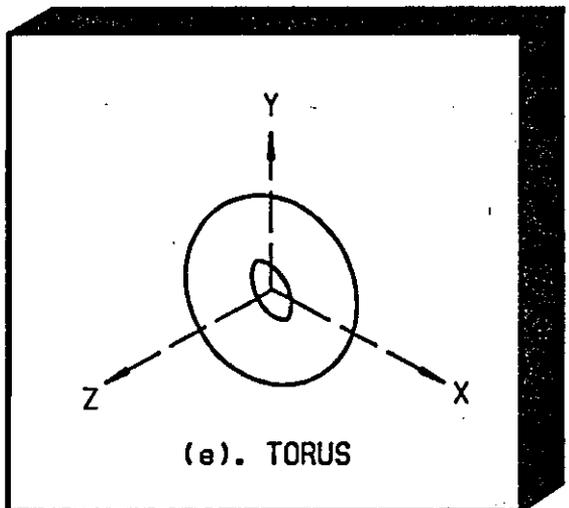
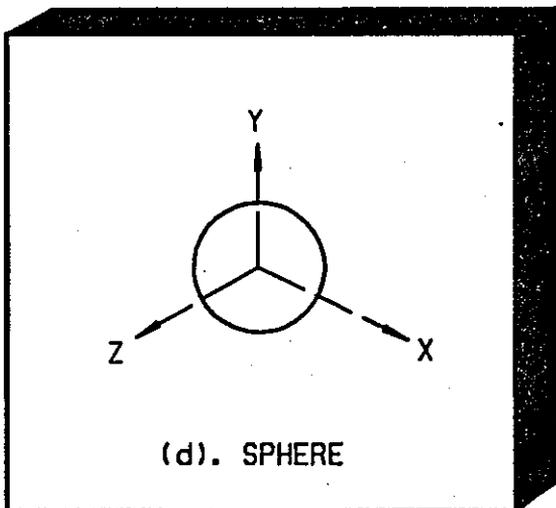
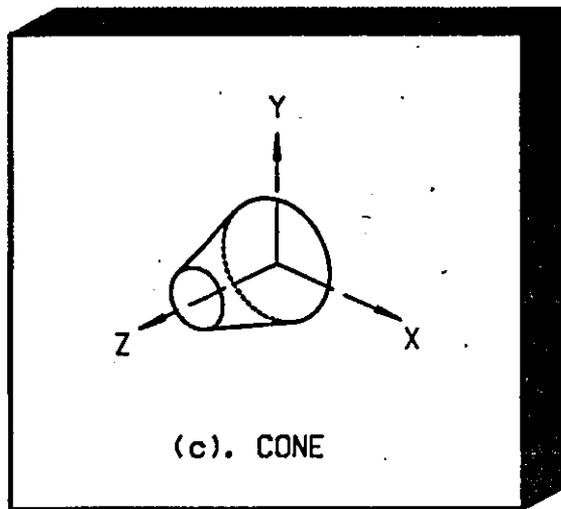
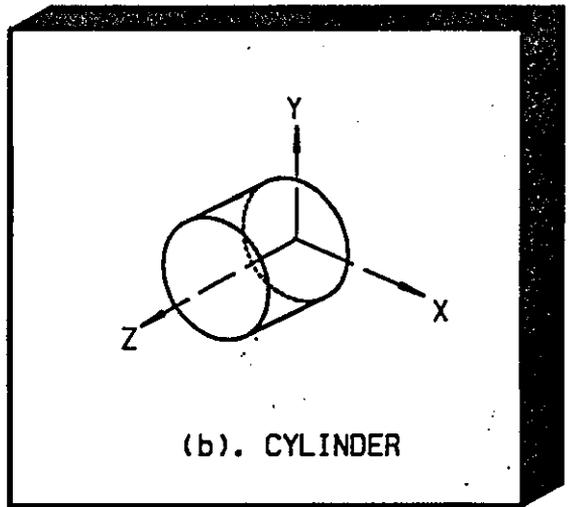
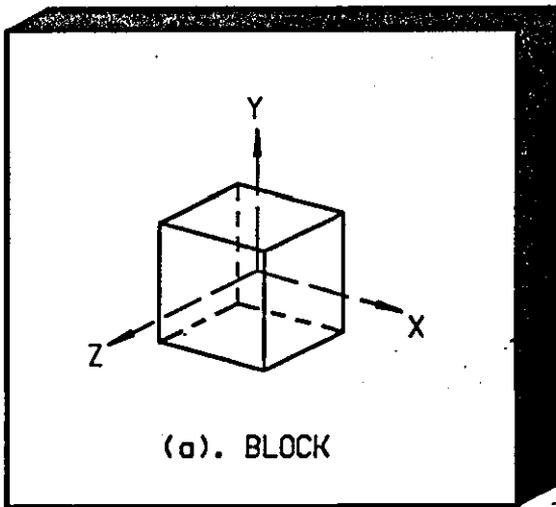


Figure 3.4 : CSG Primitives.

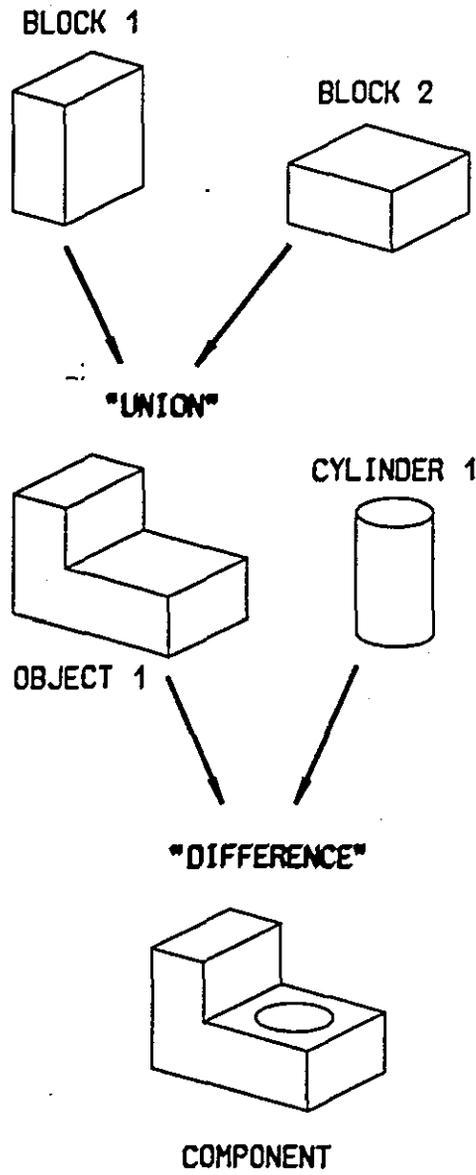


Figure 3.5 : The Tree Structure of the CSG Model.

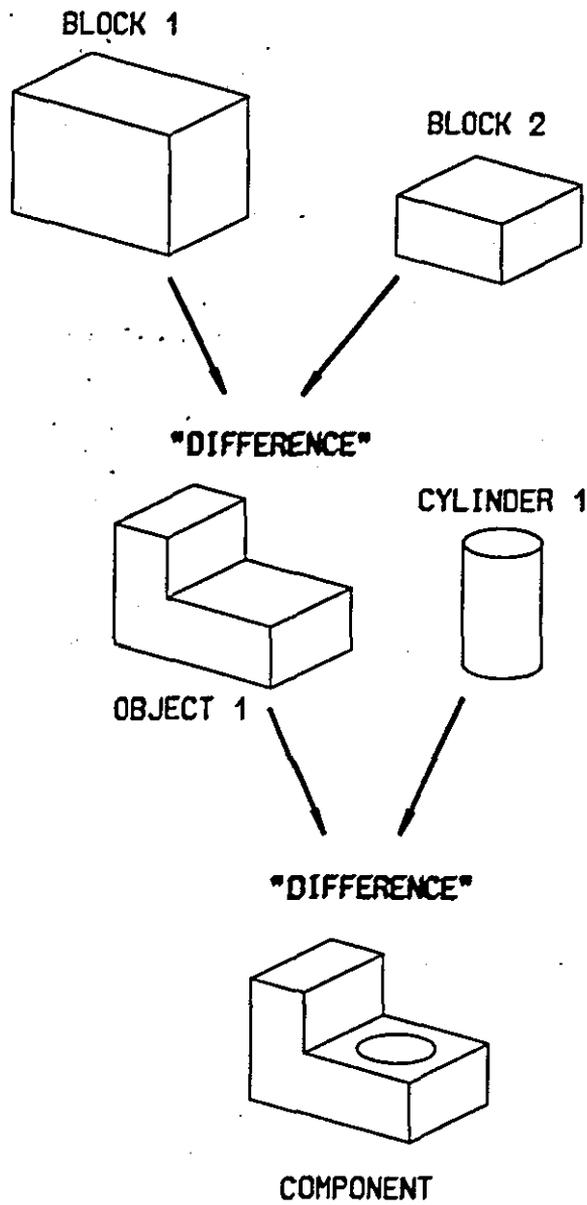


Figure 3.6 : The Tree Structure of the DSG Model.

3.3.3.2 Boundary Representation (B-Rep)

B-Rep represents an object by its boundary faces. In any B-Rep technique, there are 2 distinct aspects of the representational scheme : the topology and the geometry of the boundary faces. The topology is manifested in the form of faces, edges and vertices (FEV) specified with the relevant adjacency relationships between them in making the boundary faces. This representation is shown in Figure 3.7.

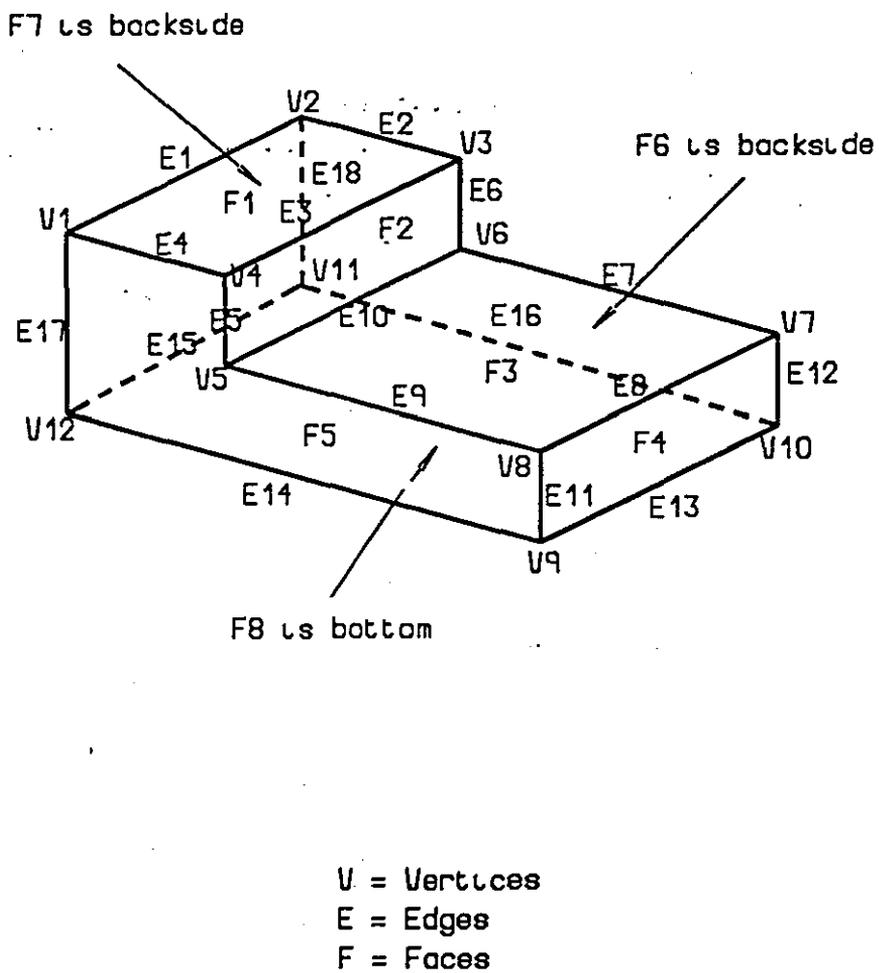


Figure 3.7 : Boundary Representation (B-Rep) Scheme.

3.3.4 Comparison of the Modellers Storage

A comparison between CSG and B-Rep modelling techniques and their representation schemes [HENDERSON,1984] is shown in Table 1.

Modelling Technique	CSG	B-Rep
Model Representation	Implicit	Explicit
Feature Representation	Explicit	Implicit

Table 1 : Comparison of Modellers Storage.

The above table shows that the CSG tree stores a model in an implicit or "unevaluated" form; the edges and vertices of the final component must be calculated from the set operations on the primitives.

Features, however, such as holes, slots and so forth may actually be presented in the CSG tree. In other words, they may be presented explicitly. An example of explicit representation of a feature is the subtraction of a cylinder to create a hole. The definition of the hole can be derived directly from the definition of the cylinder. It is possible that the feature may not be presented in the tree. The features may arise from the evaluation of the CSG tree. For example a slot may be formed by adding two slightly separated blocks to a large block or by subtracting a block from a large block as shown in Figure 3.8.

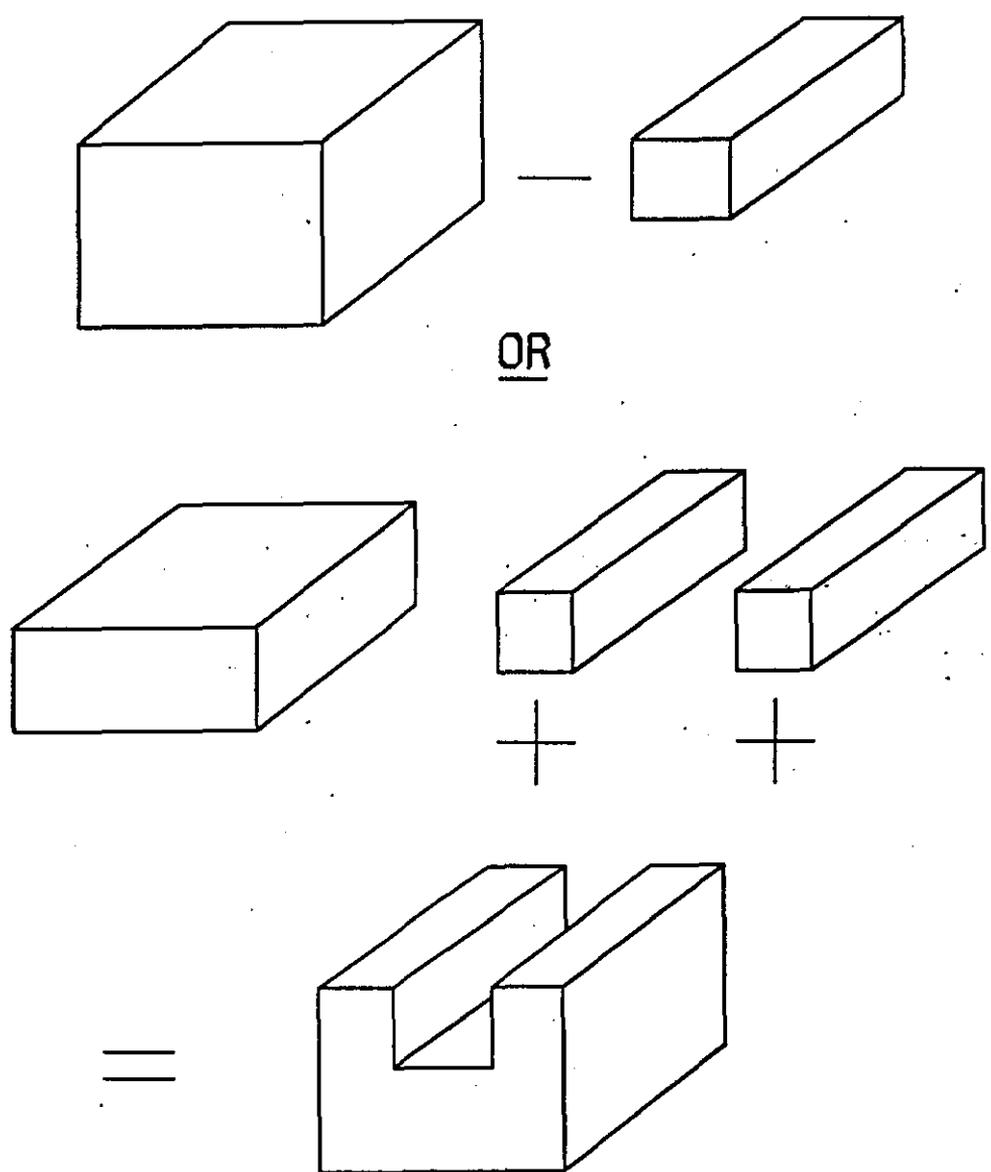


Figure 3.8 : Alternative Way in Creating a Slot.

In the B-Rep scheme, the model is represented explicitly. The faces, edges and vertices are represented explicitly exactly as they are in the final component. In this representation the features are represented implicitly, requiring interpretation of the geometry and topology. For example, a hole may be presented as a collection of faces (Figure 3.9).

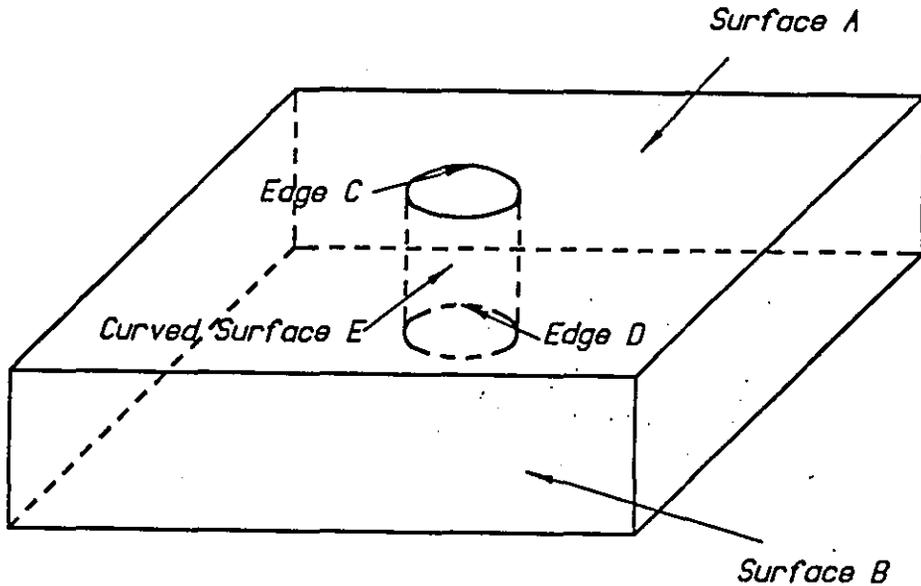


Figure 3.9 : Collection of Faces for a Hole.

This comparison seems to suggest that manufacturing features may be more easily extracted from the CSG tree because it can describe them explicitly. For example, if the designer used only machining primitives, the object could be created by beginning with a stock bar of material and simulating the manufacturing processes during the design process by subtracting primitive volumes [BENNATON 1986], [CASE 1986] until the object of the finished component is obtained.

This direct technique is intended to be used in the proposed system which has been discussed in the previous chapter and will be further discussed in Chapter 6.

3.4 Boolean Operators

The relationships between primitives are defined using the BOOLEAN operators UNION, DIFFERENCE and INTERSECTION. The UNION operator combines two bodies into a single new body as shown in Figure 3.10. This combination can now be treated as a single homogeneous body. Any point in either of the original bodies becomes a point in the new body.

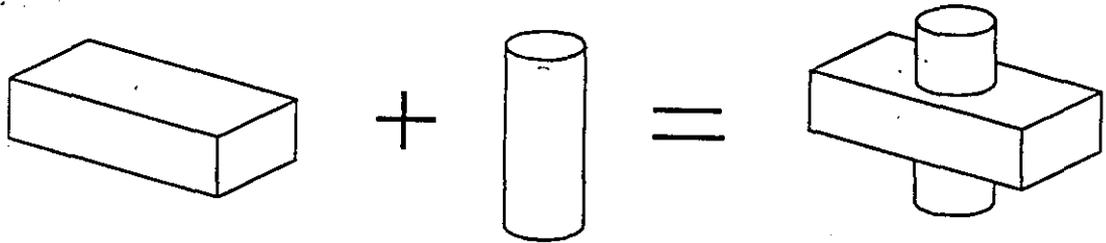


Figure 3.10 : UNION Operation.

The DIFFERENCE operators may be used if the removal of material from a body required as illustrated in Figure 3.11. A DIFFERENCE operation applied to two bodies removes from the first those components common with the second.

This operator is analogous with material removal manufacturing processes and is used as the basis of this research.

The INTERSECTION operator has the effect of calculating the common volume shared by two bodies. The result of an INTERSECT operation contains only those points that lay within both original bodies, as shown in Figure 3.12.

There is one special operator, ASSEMBLY, in which objects are combined as in UNION but instead retain their own identity. Figure 3.13 illustrates the distinction between UNION and ASSEMBLY.

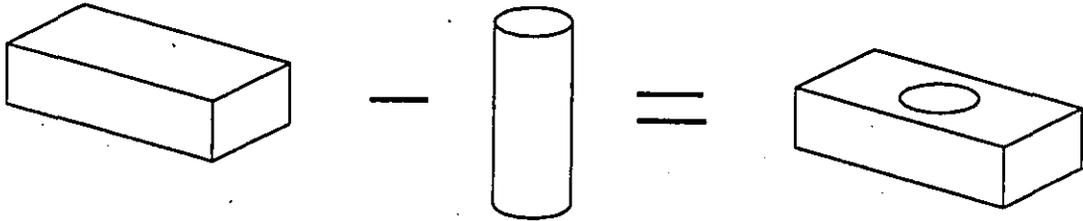


Figure 3.11 : DIFFERENCE Operation.

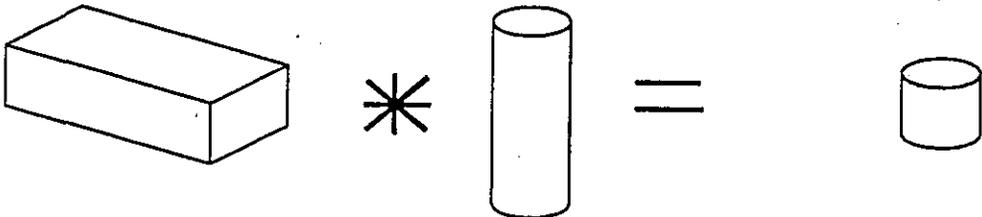


Figure 3.12 : INTERSECT Operation.

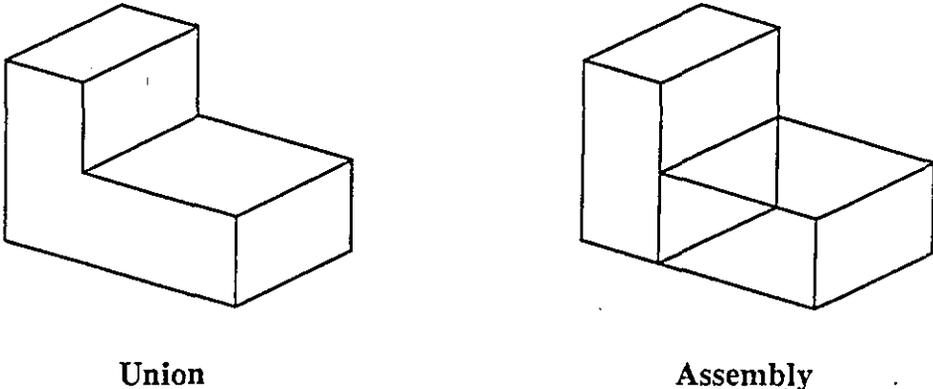


Figure 3.13 : ASSEMBLY Operation.

3.5 Data Structure

A tree structure with the primitives as its leaves and the BOOLEAN operations as its nodes is the data structure used to build the CSG model. The tree is known as a BOOLEAN tree and is illustrated in Figure 3.14, where U* stands for UNION and -* stands for DIFFERENCE.

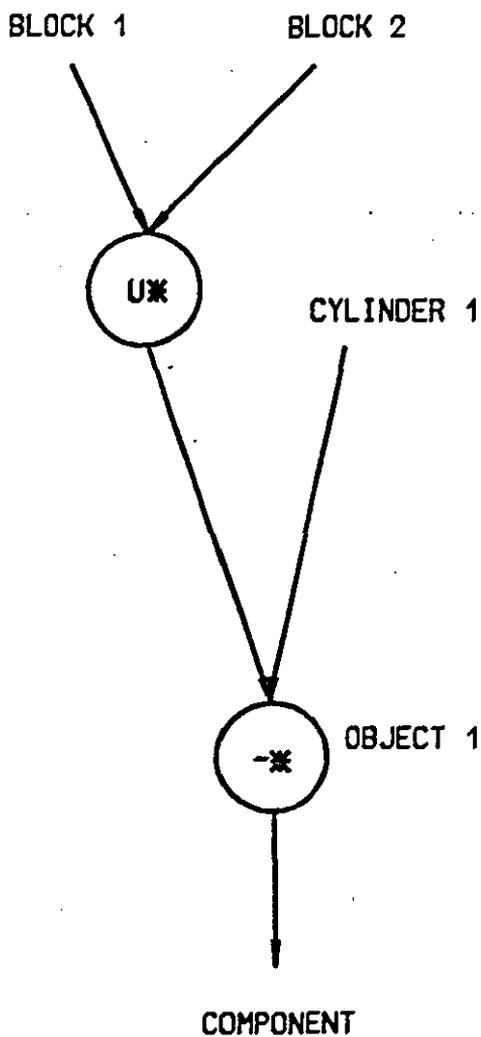


Figure 3.14 : Data Structure of CSG Model.

3.6 Summary

This chapter has described the geometric modelling that can be used in representing a solid model of an object. The CSG solid modelling method has been chosen as a tool in representing the object because this system uses a "DIFFERENCE" Boolean operator to subtract primitives shapes which is similar to manufacturing processes. This is one of possible ways of integrating the design and manufacturing functions [BENNATON,1986].

Chapter 4

CHAPTER 4

FEATURE ORIENTED REPRESENTATION

4.1 Introduction

The sufficient and proper representation of "higher-level" information such as features, tolerances and surface finish are the key to realize the integration of CAD and CAM systems [MINER,1985]. Most CAD solid modeller systems create a solid model of an object such as Constructive Solid Geometry (CSG) or Boundary Representation (B-Rep), but they themselves do not have the capability to represent the complete specification of the object for manufacturing.

This is because CAD systems represent models in terms of geometric entities such as faces, edges, and vertices for Boundary model and primitives and Boolean operators for a CSG model. These entities do not have any meaning in the manufacturing sense where CAM systems require detailed part information in terms of features such as "cylinder", "taper", "undercut", "hole", etc.

With respect to features, a designer should be able to create features in a solid model using the feature-based commands and then a feature's dimensions and location should be stored in the solid modeller and be available to manufacturing applications.

4.2 The role of features

The internal representation of the solid modeller provides a representation of components which is not useful for the process planning function. This representation is expressed in terms of "lower-level" information such as geometry (point coordinates, surface equation etc.) for wire-frame, geometry and topology (connections between points, edges, faces) for B-Rep and primitives and operators for CSG, that is unsuitable for direct application to process planning because this CAM function needs information in terms of "higher-level" features. These "higher-level" features will provide the information for the process planning activity and this will allow the operations

in order to produce the component to be based upon the features.

Research to provide CAD integration for manufacturing applications has been categorized in two main approaches for obtaining the relevant information. These are (i). Feature recognition approach, and (ii). Designing with Features (feature-based design) approach. Both of these approaches have been discussed in detail in Chapter 2.

4.3 Definition of Feature

At the moment, the term "feature" does not have a definition which is agreed upon by everyone. The Collins Dictionary and Thesaurus [MCLEOD 1986] defines feature as : "1. *any one of the parts of the face*, 2. *a prominent or distinctive part.*"

The definition of feature is dependent on context. Features have been defined in various drafting standards [ISO-129 1985], [BS-308 1985], [ANSI-Y14.5 1982] as shown in Table 2. There are also many different definitions of feature that have been given by researchers looking from a design or manufacturing perspective. Some of them are stated below:

1. [CHAN,1986]

A feature can be defined as "*a chunk of surface geometry on the finished part with some associated surface geometry that is to be produced - by a specified machining process.*"

2. [CHANG,1990]

A feature refers to "*a subset of geometry on an engineering part which has a special design or manufacturing characteristic.*"

3. [CLARK,1987]

A feature is "*a geometric shape defined by a parameter set, which has special meaning to a design or manufacturing engineer.*"

Standard	Feature definition
ISO 129 ^(*)	<i>A feature is an individual characteristic such as a flat surface, a cylindrical surface, two parallel surfaces, a shoulder, a screw thread, a slot, a profile, etc.</i>
BS 308 (Part 2) ^(**)	<i>A feature is defined as the element part of an object, such as a plane, a cylindrical surface, an axis, or a profile.</i>
ANSI Y14.5 ^(***)	<i>A feature refers to a physical portion of a part, such as surface, a hole or a slot.</i>

Table 2 : Definitions of Feature from Various Drafting Standards.

(*). International Organization for Standardization : "Technical drawing-dimension-general principles, definitions, methods of execution and special indications", ISO 129-1985 (E).

(**). British Standard Institution BS 308 Part 2 : "Recommendations for dimensioning and tolerancing of size 1985", BSI (1985).

(***). The American Society of Mechanical Engineers : "Dimensioning and tolerancing", ANSI Y14.5M - 1982.

4. [CUNNINGHAM,1988]

A feature is *"any geometric form or entity that is used in reasoning in one or more design or manufacturing activities; features can have geometry, topology, and attributes."*

5. [DAVIES,1986]

Features are *"Patterns."*

6. [DONG,1988]

A feature is *"a set of geometrical and topological entities on a part that possesses certain design or manufacturing significance."*

7. [HENDERSON,1986]

A feature is defined as *"a geometric entity or set of entities which determine a code related digit."*

8. [HERBERT,1990]

A feature is *"a group of geometric entities with some meaning for the particular activity to be performed with them i.e. process planning."*

9. [HUMMEL,1986], [JOSHI,1988]

Features are *"regions of the part that have some degree of manufacturing significance."*

10. [LONGNECKER,1988]

Features are defined as *"recognizable in sentential forms of sentences generated from shape grammars and generable as elements of production rule of the grammar itself."*

11. [LUBY,1986]

A feature is *"a geometric form or entity whose presence or dimensions are required to perform at least one engineering task."*

12. [NAU,1987]

Features are defined from a machining viewpoint. Every machining process is said to create such as a hole, a slot, a pocket, etc.

13. [OSTROWSKI,1986]

A feature is *"a region of interest on the surface of a part."*

14. [PETERS,1990]

A feature is *"a distinctive or characteristic part of a workpiece defining geometric shape which is specific for the operation intended."*

15. [REQUICHA,1985]

A feature is *"portions of the objects' surface i.e. topological boundary."*

16. [ROGERS,1987]

A feature is *"recurring patterns of information related to a parts description."*

17. [SHAH,1986]

For manufacturing engineering features are *"forms that can be produced by certain manufacturing operations."*

18. [SHAH,1988a]

Features are defined as *"information sets related to product engineering."*

19. [SHAH,1988b]

Features are defined as *"information sets ; recurring patterns of information related to a parts description."*

20. [TURNER,1988]

A feature is *"a representation of shape with geometry and attributes that are recognizable and meaningful to humans and programs that must interact with them."*

21. [VAGHUL, 1985]

A feature is *"a geometry entity : 1. whose presence location, or dimensions are germane to the functionality or manufacturability of the part, or 2. whose availability as a primitives facilities design in the domain."*

22. [WANG,1987b]

Every solid line in a 2-D drawing of a turned part, which does not cut the axis of the part, is considered a machining feature.

23. [WANG,1987c]

All lines and arcs which represent machined surfaces are termed features.

24. [WOODWARK,1988]

Features are *"pattern represented by a combination of primitives."*

In this research work, a feature corresponds to the removal of a solid volume of material from the stock material. So, a feature in this proposed system can be defined as *"a volumetric shape which is defined by a set of parameters such as length, diameter, etc., which can be produced by certain manufacturing processes based upon their names"*.

4.4 Feature Taxonomy

Features can be classified in different ways [CHANG,1990] :

- i. Based on the geometry,
- ii. Based on the applications.

4.4.1 Features Based on the Geometry

Based on the geometry, features can be classified into the following :

- i. Face features
- ii. Volumetric features

Face features may be used for 2-D or 3-D modelling CAD system. Some typical features can be hole pattern, gear, fillet, etc.

Volumetric features are best used in conjunction with a CSG modeller. The features are primitives which carry special meanings. Set operators can be used to construct a solid model using features and regular solid primitives. Some typical volumetric features are hole, boss, simple slot, T-slot, V-slot, keyway, groove, etc.

4.4.2 Features Based on the Application

Based on the application, features can be defined into the following :

- i. Design features,
- ii. Manufacturing features.

Design and manufacturing features can both be either face features or volumetric features. Design features are those features which are meaningful to the design. Some design features are hole, chamfer, groove, counter-bore, counter-sink, etc.

Manufacturing features are features which are meaningful to manufacturing. Some manufacturing features are hole, chamfer, groove, taper, screw thread, counter-bore, counter-sink, pocket, fillet, etc.

From the above classification, design features and manufacturing features actually overlap, and it can also be seen that many manufacturing and design features are similar.

4.4.3 Features in the Prototype System

In this research project, a number of volumetric features of manufacturing features for rotational components proposed by [PANDE, 1988] were considered. These features are classified into two groups, namely external and internal features, and further subclassified into axial, surface and radial types. Manufacturing features that are used in the prototype system are shown in Figure 4.1. An example of the standard features and their definitions are stated below :

- (i). **Cylinder** : any cylindrical surface (see Figure 4.2)
- (ii). **Cylinder-Bore** : any internal cylindrical surface (see Figure 4.3)
- (iii). **Face** : any surface that is perpendicular to the axis of rotation (see Figure 4.4)
- (iv). **External-Taper** : any external tapering surface (see Figure 4.5)
- (v). **Internal-Taper** : any internal tapering surface (see Figure 4.6)
- (vi). **Knurl** : any portion of the surface that is knurled (see Figure 4.7)
- (vii). **External-Thread** : any portion of the surface that is threaded externally (see Figure 4.8)
- (viii). **Internal-Thread** : any portion of the surface that is threaded internally (see Figure 4.9)
- (ix). **Undercut** : any portion of the component that is undercut (see Figure 4.10)

EXTERNAL FEATURES

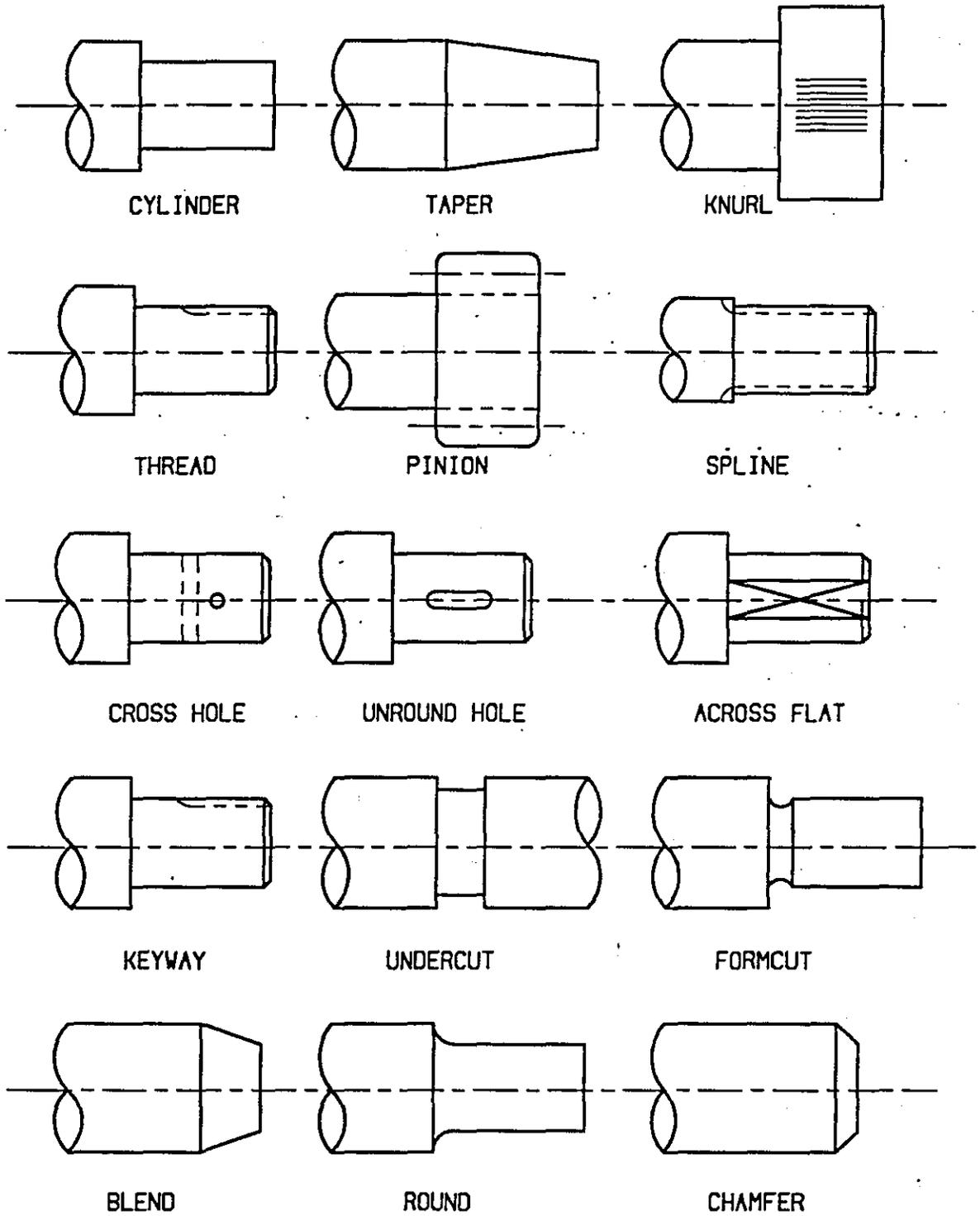


Figure 4.1 : Manufacturing Features of Rotational Component.

INTERNAL FEATURES

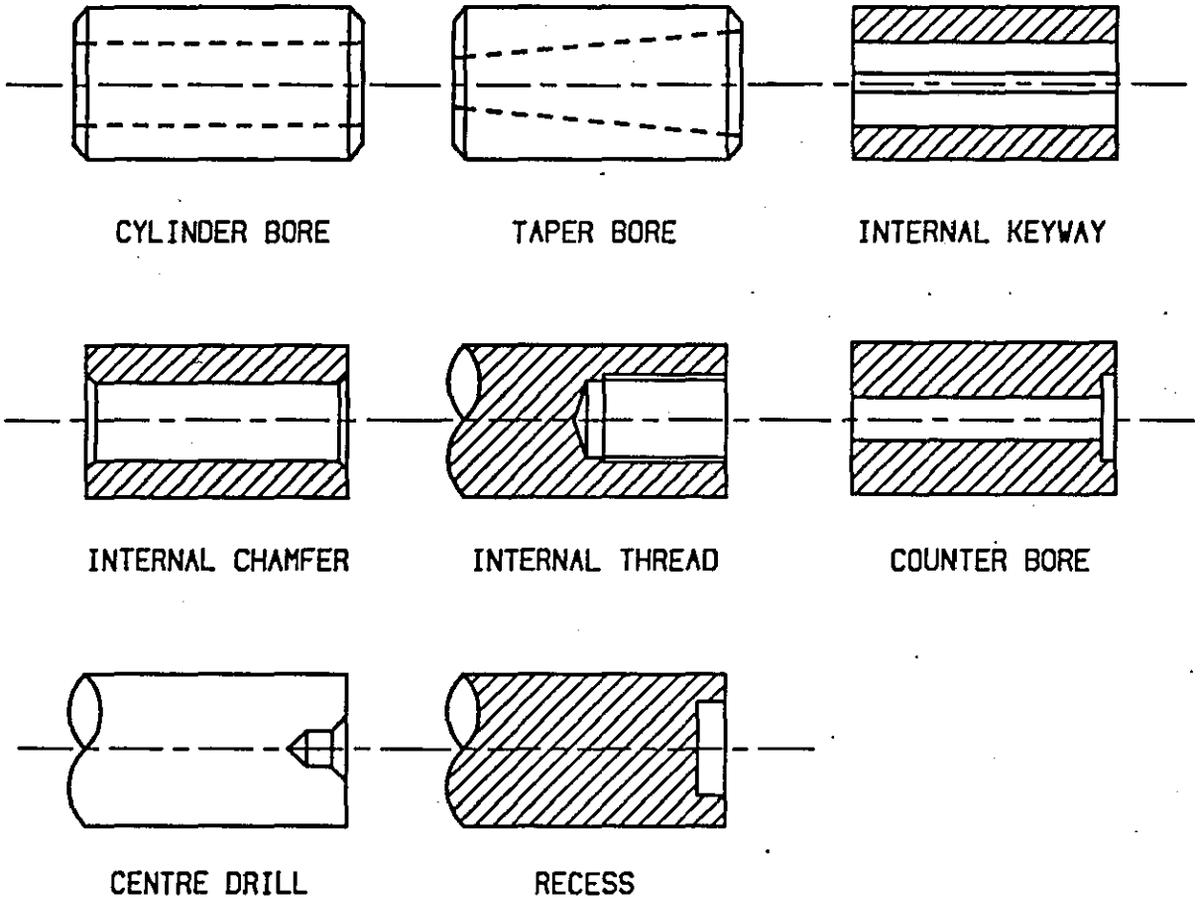
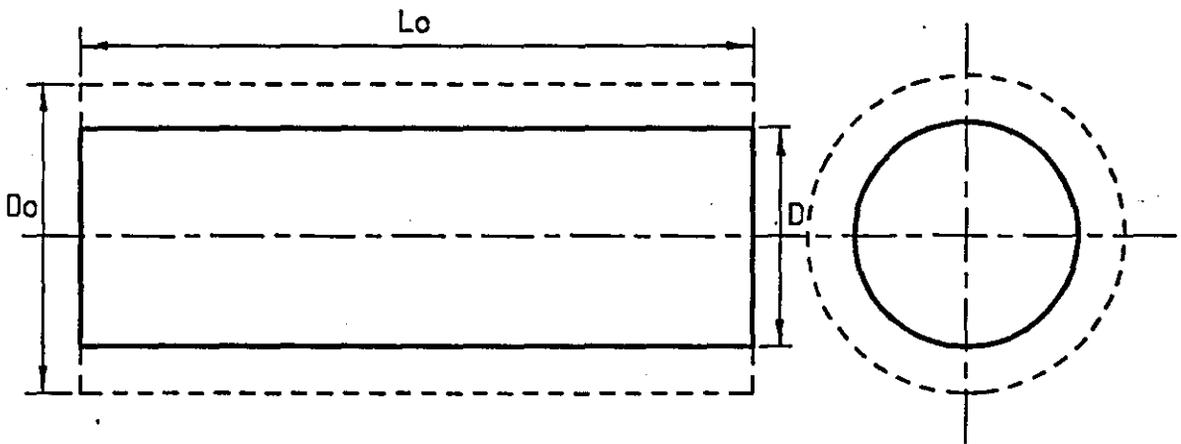
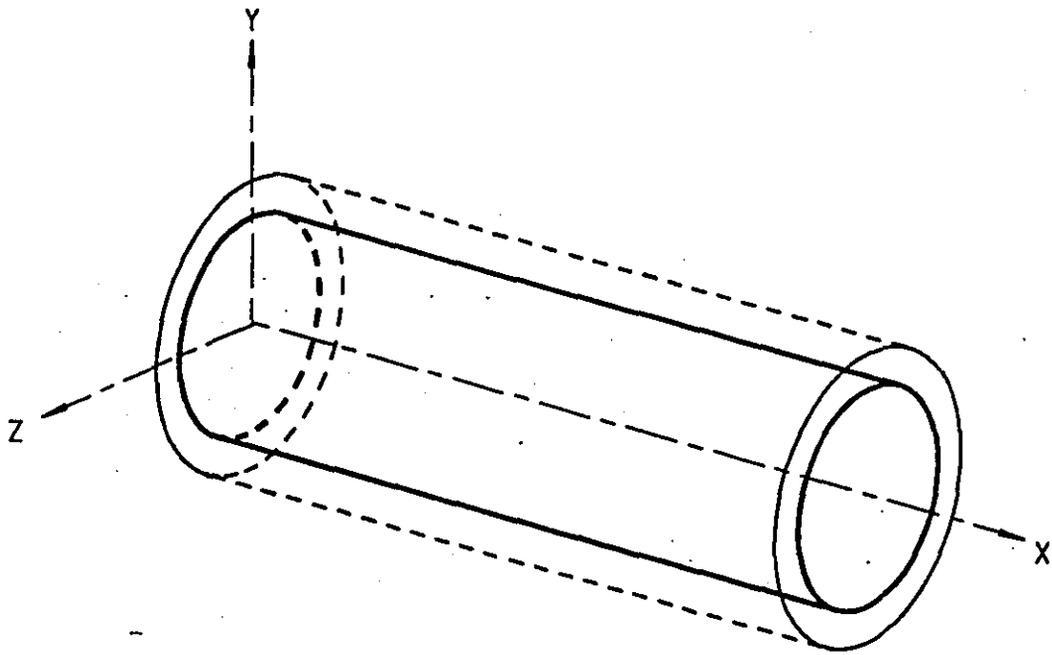


Figure 4.1 : Continued

- (x). **Chamfer** : any portion of the component that is chamfered (see Figure 4.11)
- (xi). **Form** : any portion that is curved in nature (see Figure 4.12)

Only a few features such as cylinder, face, taper, chamfer, undercut, form and recess have been implemented in the prototype system. However, features like knurl, thread and others which could not be implemented in the geometrical specification system are represented as a cylinder feature with their respective names. All of these features can be used in producing an outline process planning sheet.



Dashed lines represent the original dimension of stock material.

Figure 4.2 : Cylinder Feature.

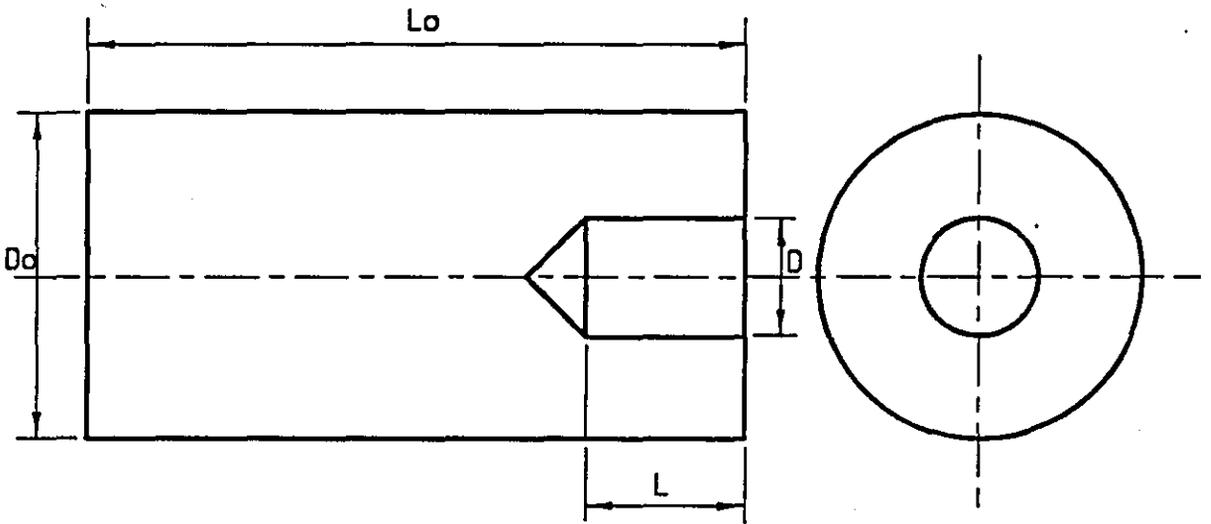
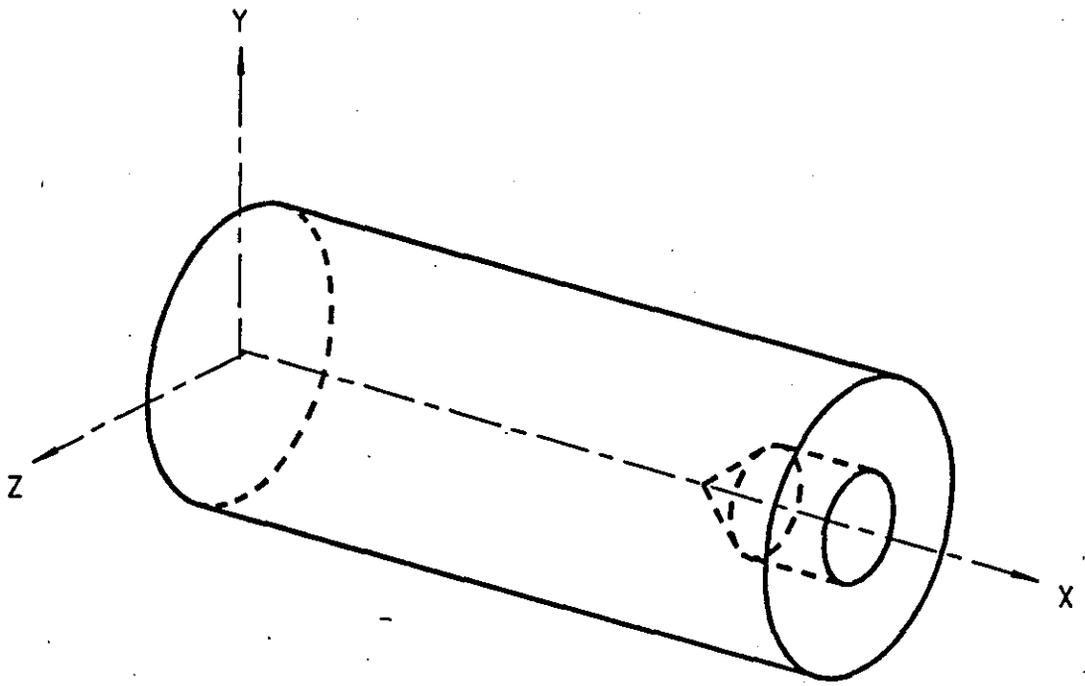


Figure 4.3 : Cylinder-Bore Feature.

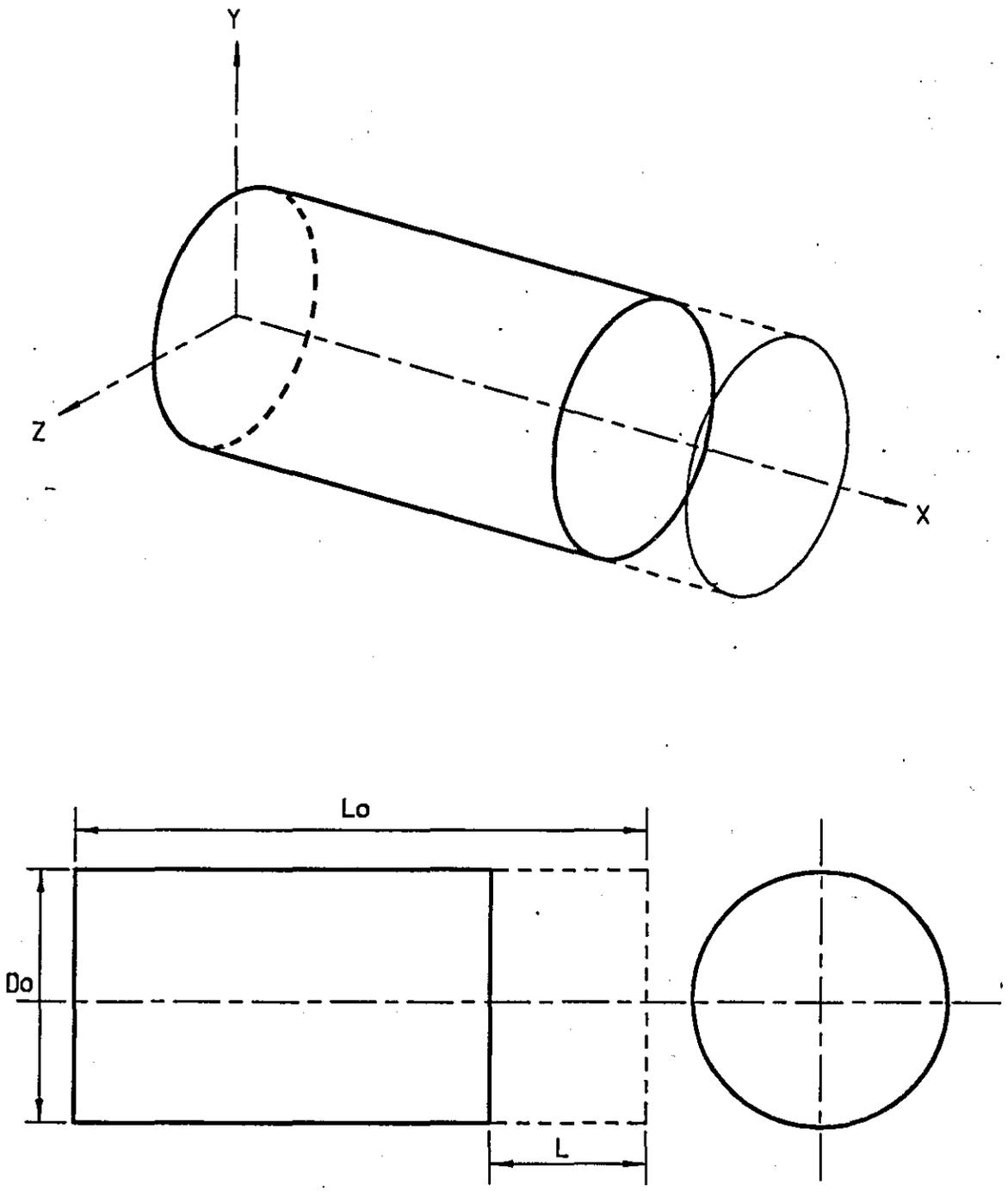


Figure 4.4 : Face Feature.

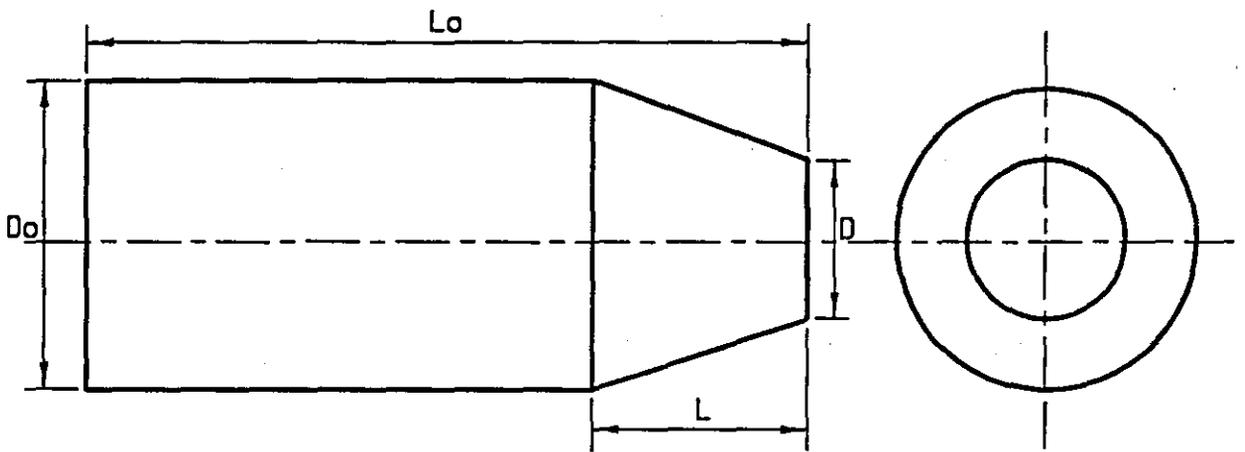
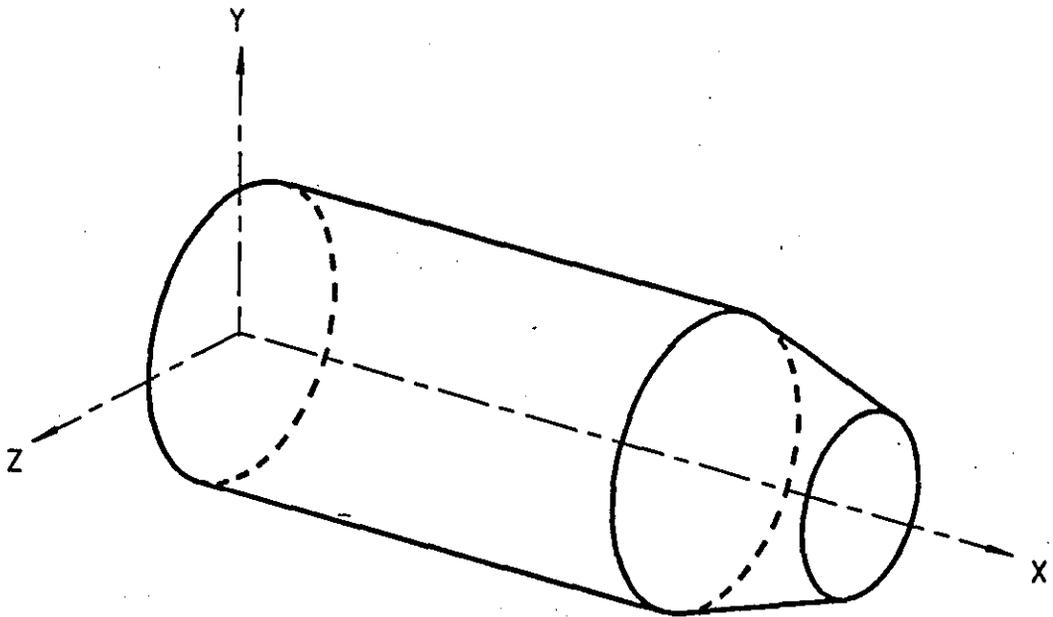


Figure 4.5 : External-Taper Feature.

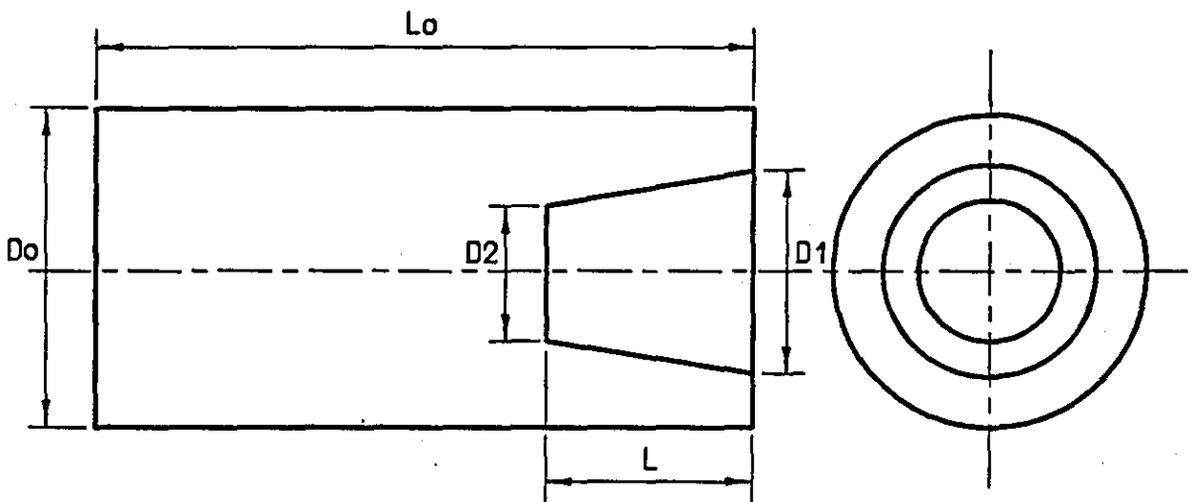
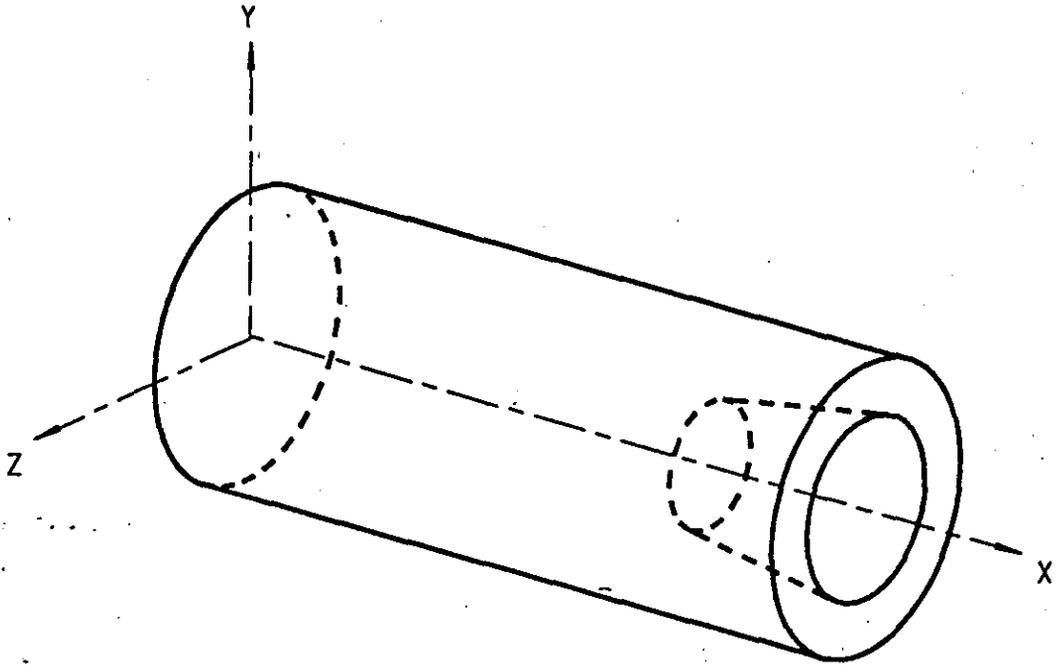


Figure 4.6 : Internal-Taper/Taper-Bore Feature.

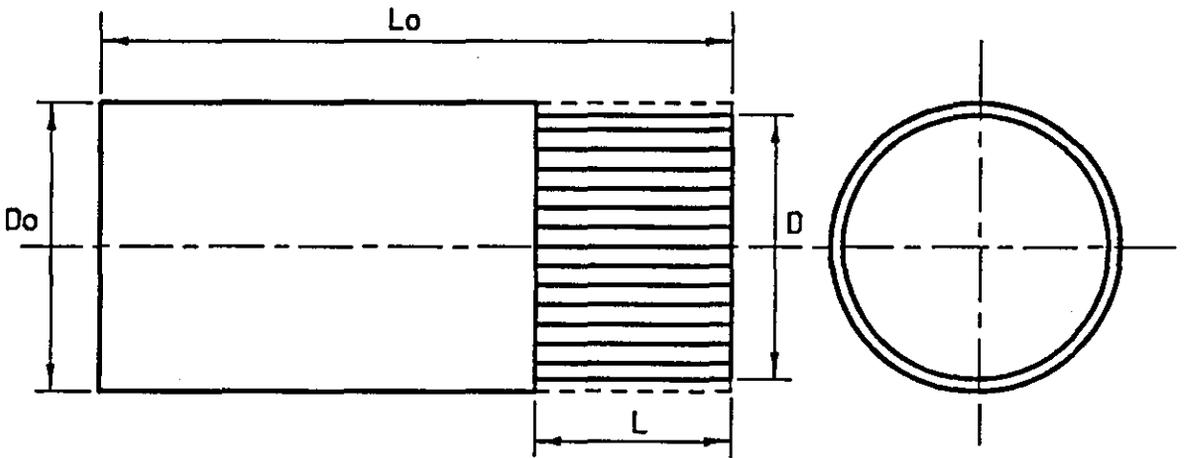
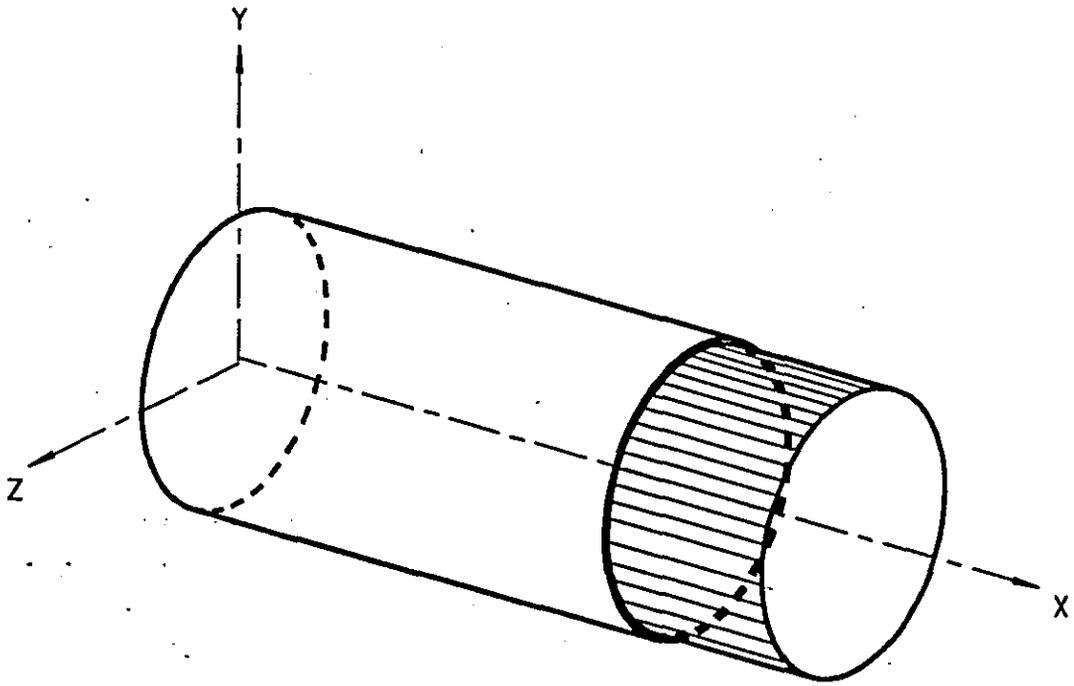


Figure 4.7 : Knurl Feature.

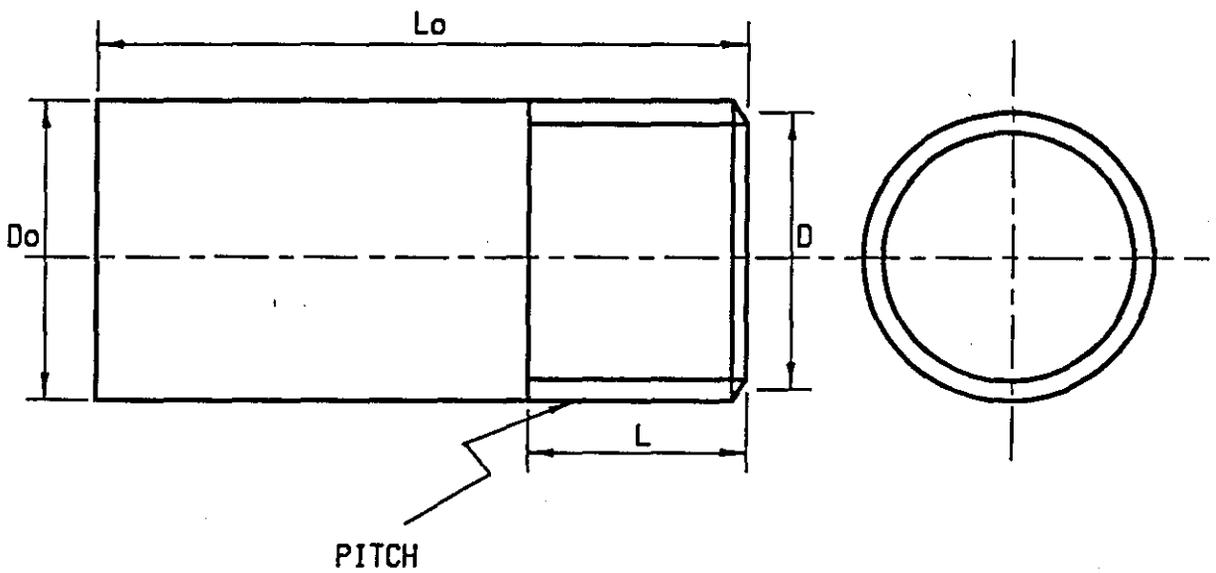
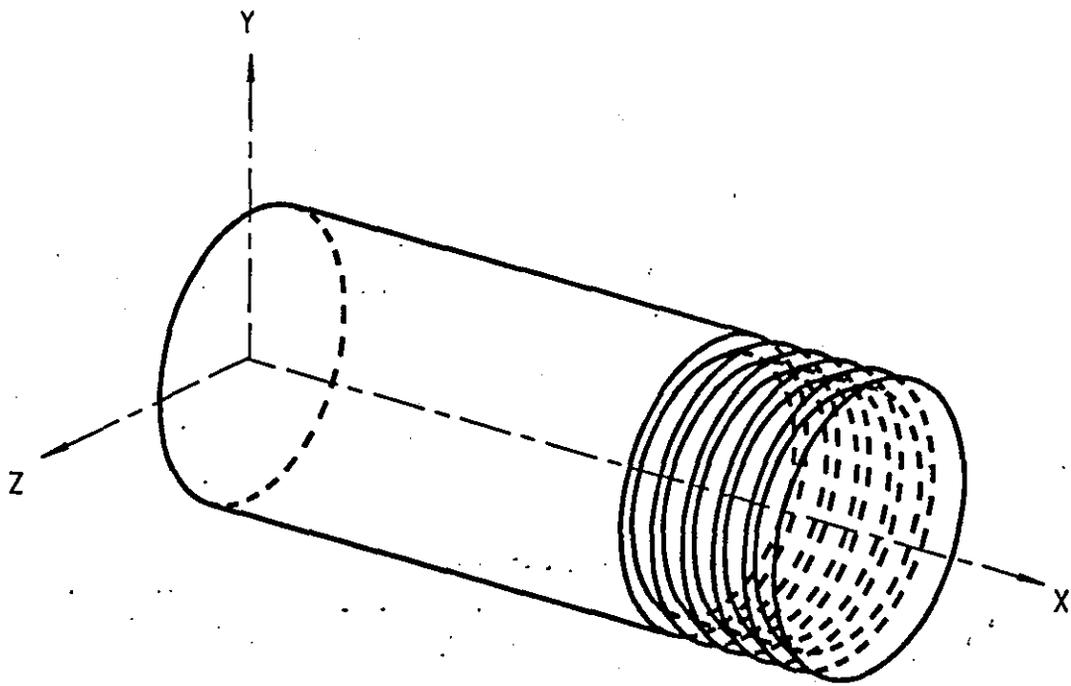


Figure 4.8 : External-Thread Feature.

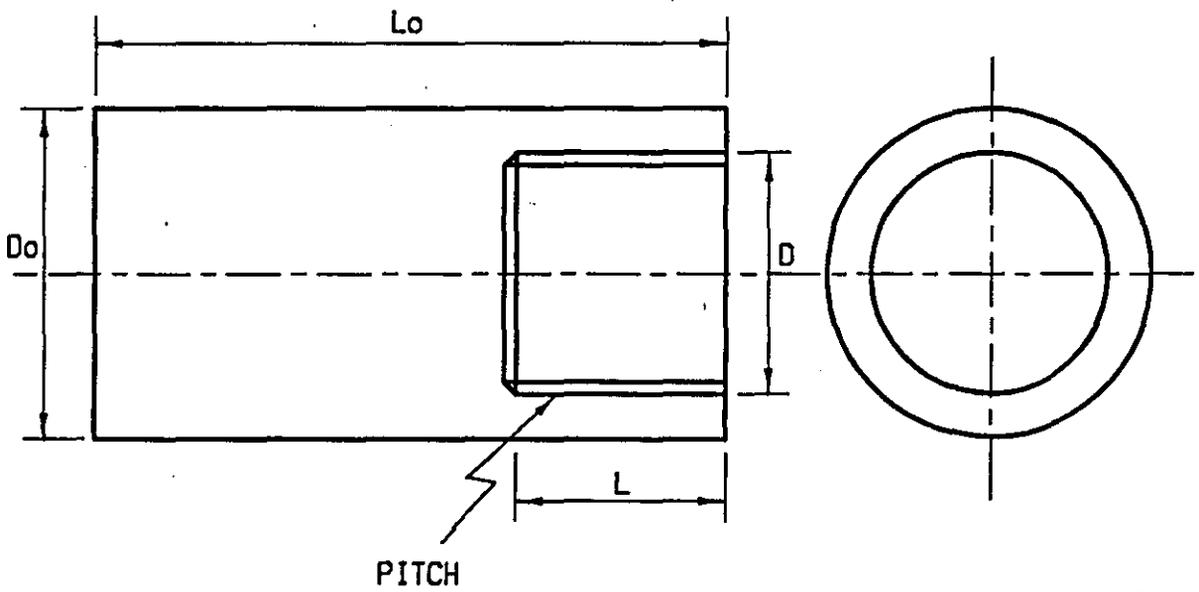
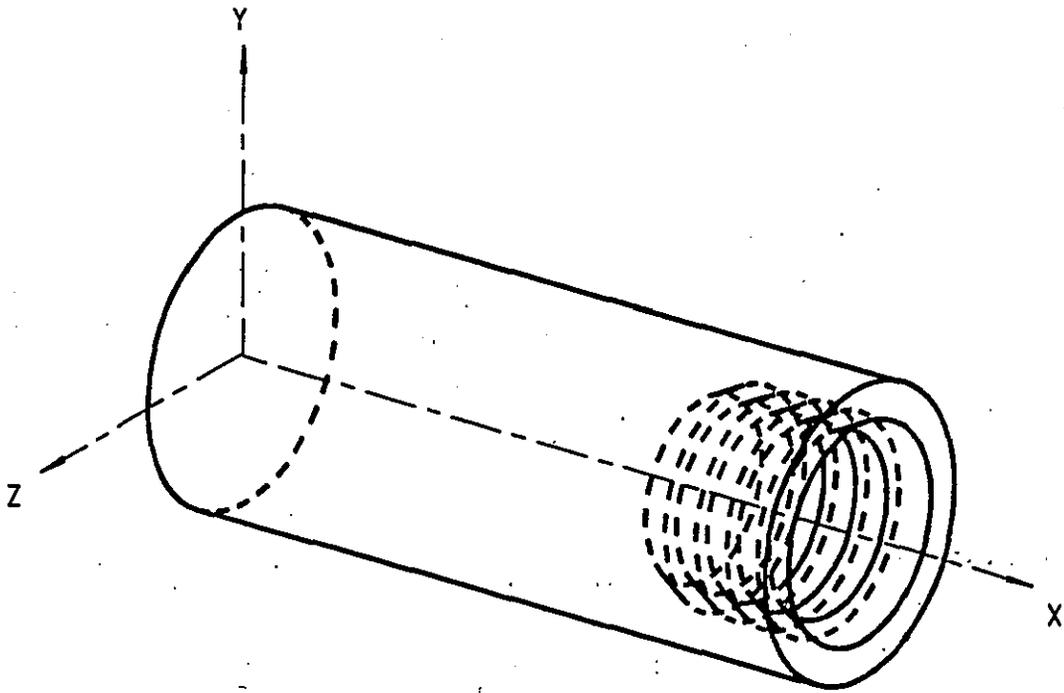


Figure 4.9 : Internal-Thread Feature.

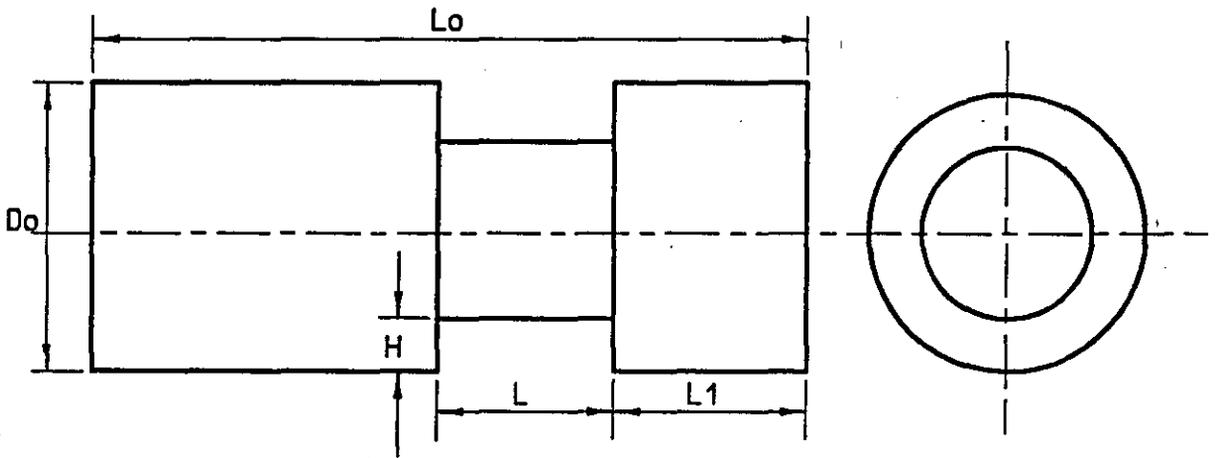
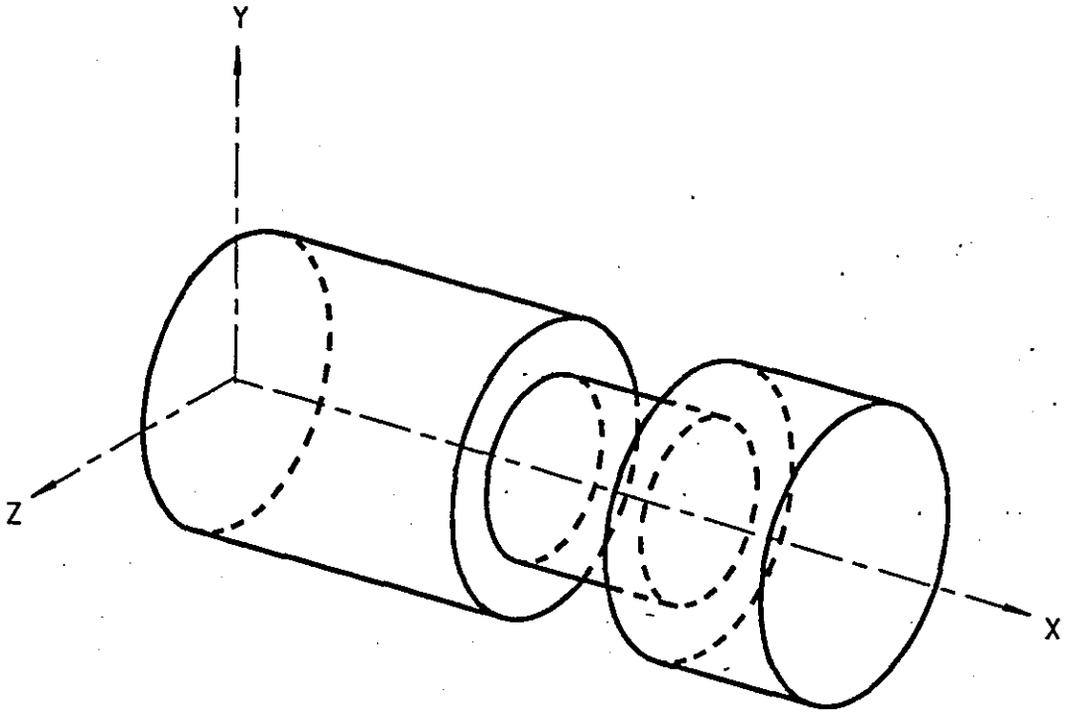


Figure 4.10 : Undercut Feature.

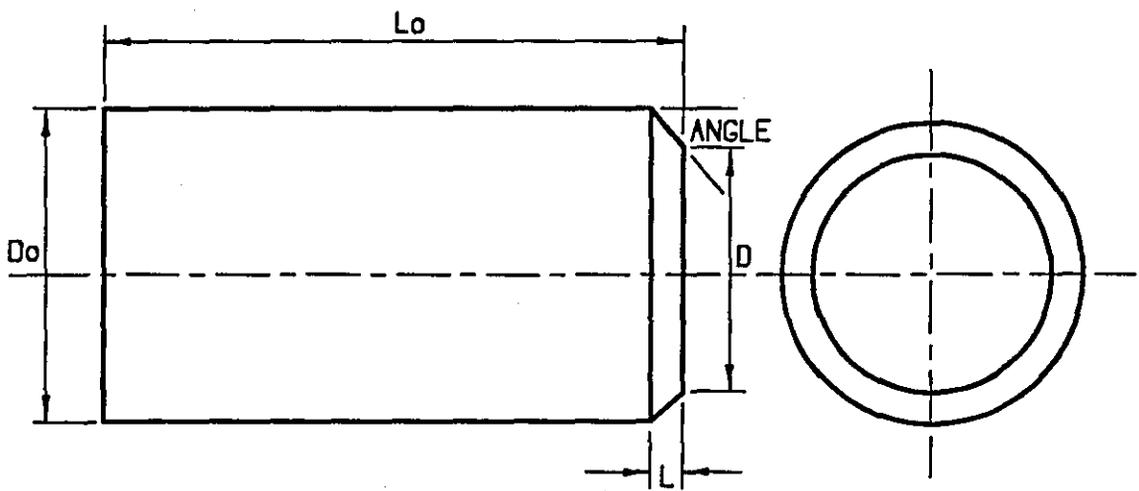
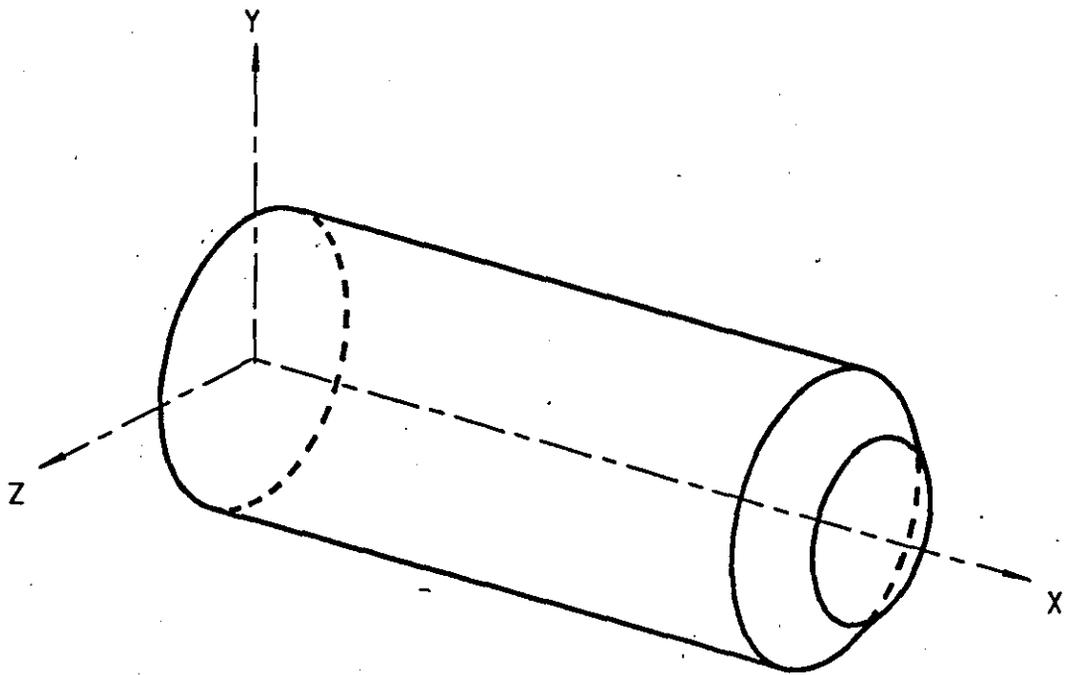


Figure 4.11 : Chamfer Feature.

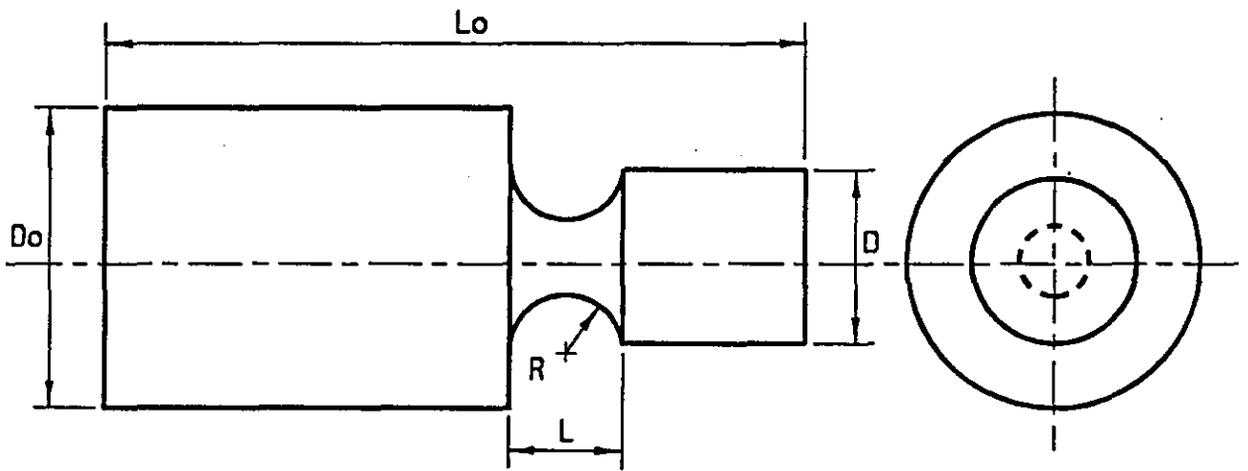
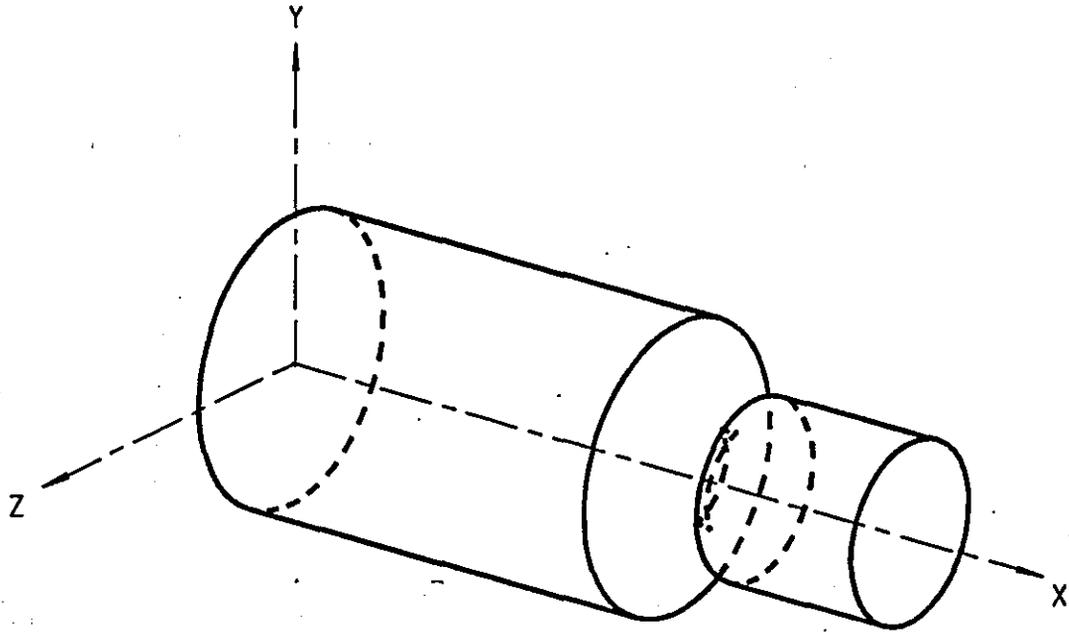


Figure 4.12 : Form Feature.

4.5 Summary

This chapter gave the definitions of features that have been defined from many design and manufacturing views. A new definition of features has been explored for use in the prototype system. By using the designing with features approach of DSG technique, the system represents features as a volume from the removal of the stock of material and the excess material.

Chapter 5

CHAPTER 5

GENERAL CONSIDERATIONS FOR TURNING OPERATIONS

5.1 Introduction

Turning operations can be defined as machining processes by which cylindrical, conical or irregularly shaped external or internal surfaces are produced on a rotating workpiece [YANKEE,1979]. The basic machine tools that are used for turning operations are called lathes. Some of the common operations that can be performed on a lathe are : parallel or straight turning, taper turning, knurling, grooving, threading, facing, drilling, boring, tapping, taper boring, counterboring and reaming. This chapter discusses the general information related to the operations which can be performed in producing the turned components in the prototype system.

5.2 Non-Machining Operations

There are two non-machining operations in the operation of a lathe [JURI 1990]. These are workpiece clamping and workpiece reversing.

Two methods have been considered for clamping workpieces in lathes. The workpieces can be either held between centres or in a chuck. The workpieces are held between centres if they are relatively long and slender with respect to their diameters. Workpieces that are difficult or impossible to hold by the first method are usually held in a chuck.

Reversing of the workpieces is done when the external and internal turning features have been formed but there are still turning features incomplete.

5.3 Machining Operations

Machining operations on lathes can be classified into two groups according to workpiece clamping method [DOYLE,1969] :

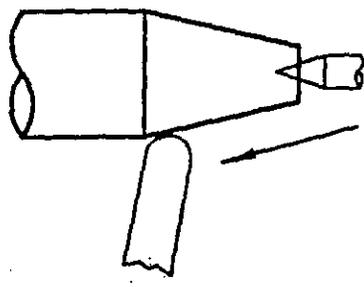
- (i). machining between centres,

(ii). machining in a chuck.

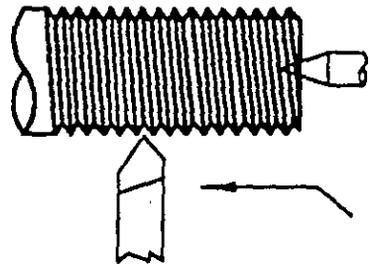
Machining between centres involve longitudinal turning and facing operations. These operations can be either roughing or finishing. The most common operations done by machining between centres are depicted in Figure 5.1.

Machining in a chuck is most commonly associated with internal features. The main operations are drilling, boring, counterboring, internal threading (tapping), taper boring, internal forming and reaming as illustrated in Figure 5.2.

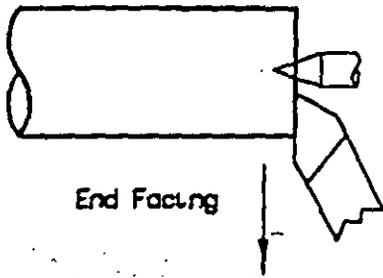
The following sections describe a few examples of machining operations on lathes for the future use in the prototype system.



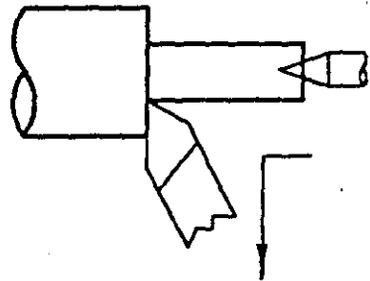
Taper turning



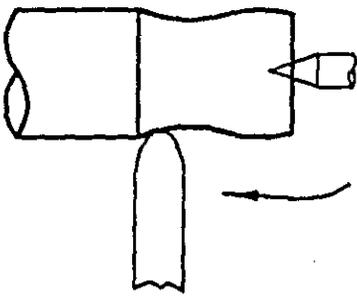
External threading



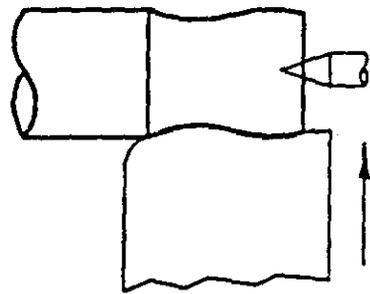
End Facing



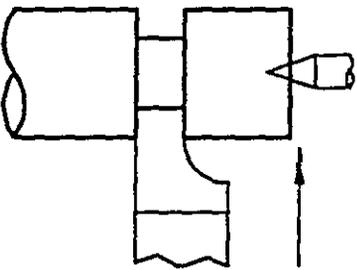
Shoulder facing



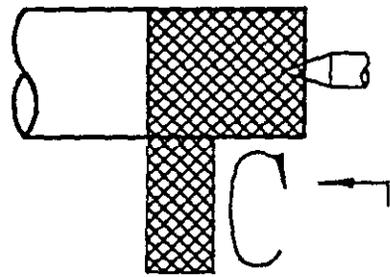
Contour turning



Forming

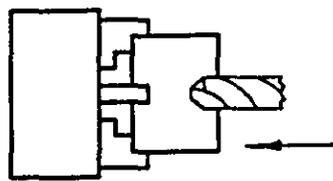


Necking

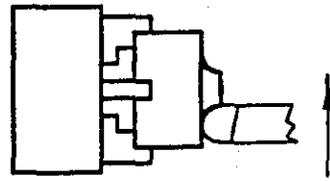


Knurling

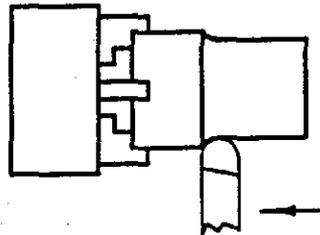
Figure 5.1 : Turning Operations Between Centres.



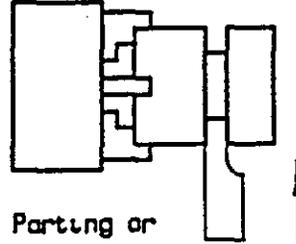
Drilling



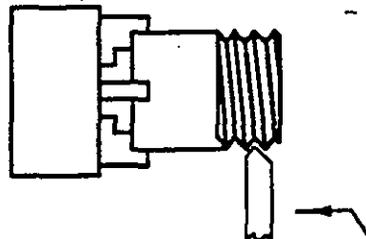
Facing



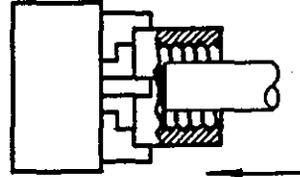
Straight turning



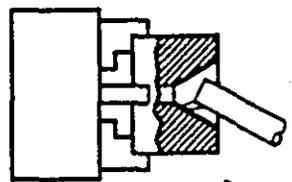
Parting or cut-off



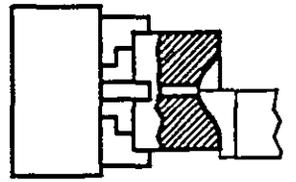
External threading



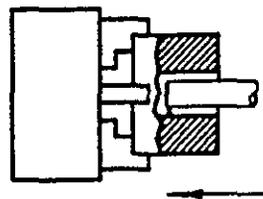
Internal threading



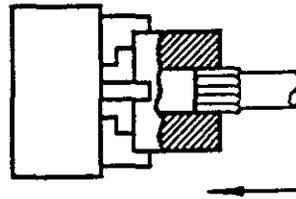
Taper boring



Internal forming



Boring



Reaming

Figure 5.2 : Chucking Operations in Turning.

5.4 Facing

Facing is the process of machining the end face to produce a smooth, flat surface which is square with the axis of the workpiece as illustrated in Figure 5.3. This operation can be done between centres or in the chuck and is performed for the following reasons:

- a. To square the end surface,
- b. To have an accurate surface from which to measure,
- c. To cut work to length.

This operation is known to the prototype system as a FACE feature. The cutting tool that has been assigned to produce this feature is left or right-hand cut-side facing tool.

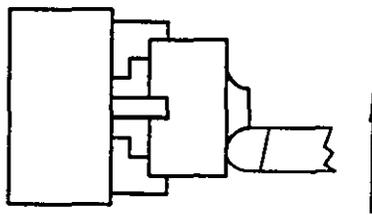


Figure 5.3 : Facing Operation.

5.5 Parallel or Straight Turning

Parallel or straight turning involves the majority of lathe work. For this operation, the workpiece can be held between centres or in a chuck. This operation can be divided into two types [DE GARMO,1984] :

- i. Rough Turning,
- ii. Finish Turning.

5.5.1 Rough Turning

Rough turning is used to remove most of the excess material as quickly as possible and to obtain the workpiece diameter.

The roughing cut should be taken to within 0.79 mm of the finished size of the workpiece. Generally one roughing cut should be taken if up to 1.27 mm is to be removed from the diameter.

5.5.2 Finish Turning

The purpose of finish turning is to bring the workpiece to the required size and to produce a good surface finish. Generally only one finish cut is required since no more than 0.79 to 1.27 mm should be left on the diameter for the finish cut.

Parallel or straight turning is appointed to produce the CYLINDER feature. The cutting tool that has assigned is left or right-hand cutting tool. Figure 5.4 shows the parallel or straight turning operation.

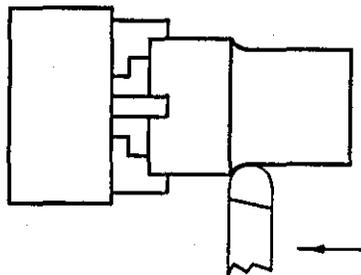


Figure 5.4 : Parallel or Straight Turning.

5.6 Taper Turning or Tapering

Taper turning or tapering is the process of cutting a diameter form on the workpiece. Taper turning can be done by five different methods on a lathe [KIBBE,1987].

- (i). The taper attachment method,
- (ii). The offset tailstock method,
- (iii). The compound slide method,
- (iv). The use of the form tool,
- (v). The use of a tracer or CNC lathe.

This operation has been appointed by the prototype system in order to produce the TAPER feature. The cutting tool that has been assigned is a left or right-hand cutting tool. This operation is illustrated in Figure 5.5.

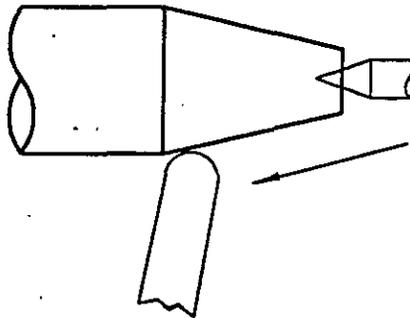


Figure 5.5 : Taper Turning or Tapering.

5.7 Knurling

Knurling is a process of creating a rough geometrical surface around the workpiece. This may be either straight or diamond pattern and may be a fine, medium or coarse pattern as illustrated in Figure 5.6. The purposes of knurling are to improve the appearance of the workpiece and to provide a better grip. This is done by forcing a knurling tool containing a set of hardened cylindrical patterned rolls against the surface of revolving workpiece.

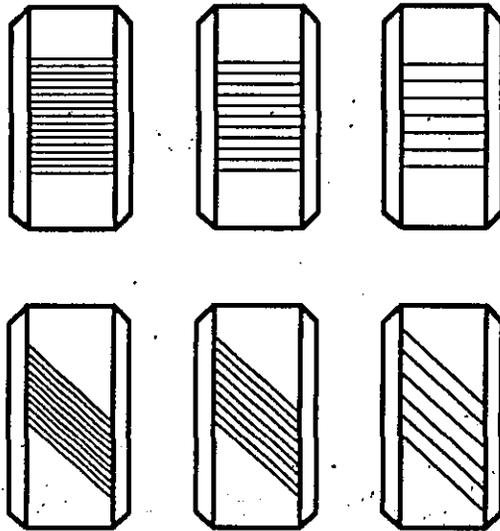


Figure 5.6 : Pattern of Knurl.

Knurling operation has been appointed by the proposed system for the KNURL feature. The cutting tool that has been assigned is a knurling tool. Figure 5.7 illustrates the operation of knurling process.

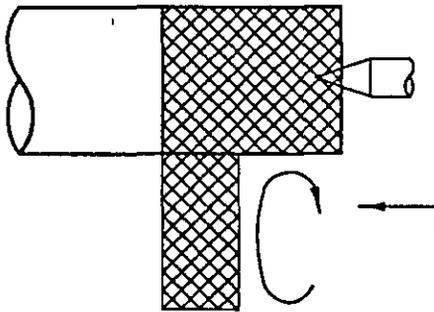


Figure 5.7 : Knurling Operation.

5.8 Undercutting

Undercutting, sometimes called necking, is the process of cutting a groove form on the workpiece. The shape of the cutting tool and the depth to which it is fed determines the shape of the undercut. This operation is shown in Figure 5.8.

This operation is appointed by the proposed system for the UNDERCUT feature. The tool that has been assigned is a square-nose or end cutting tool.

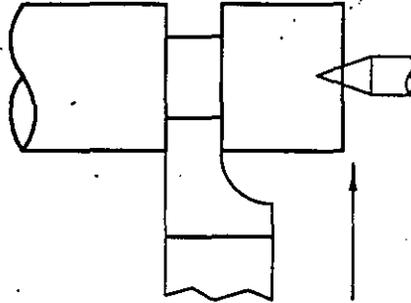


Figure 5.8 : Undercutting Operation.

5.9 External Threading

External threading is a process of producing a ridge of uniform section by cutting a continuous groove around the workpiece. This is done by taking successive light cuts with a threading toolbit the same shape as the thread form. In order to produce an accurate thread, it is important that the lathe, the cutting tool, and the workpiece be set up properly. This operation is illustrated in Figure 5.9.

This operation has been appointed by the proposed system for the EXTERNAL THREAD feature and the cutting tool for producing this feature is threading tool.

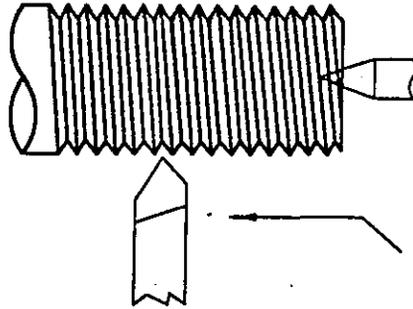


Figure 5.9 : External Threading Operation.

5.10 Drilling

A workpiece held in a chuck can be drilled quickly and accurately by a lathe. This is illustrated in Figure 5.10. The drill, held in a drill chuck or in the tailstock spindle, is brought against the revolving workpiece by turning the tailstock handreel. Drilling generally precedes the other lathe operations such as boring, reaming and tapping.

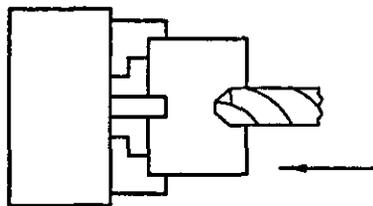


Figure 5.10 : Drilling Operation.

This operation is specified by the CYLINDER BORE, TAPER BORE and INTERNAL THREAD features. This operation is performed by using a tool called a drill.

5.11 Taper Boring

Taper boring is the process of cutting the internal diameter with taper forms on the workpiece as illustrated in Figure 5.11. This operation is appointed for the TAPER BORE feature and the tool that has been assigned by the prototype system in order to produce this feature is a boring tool.

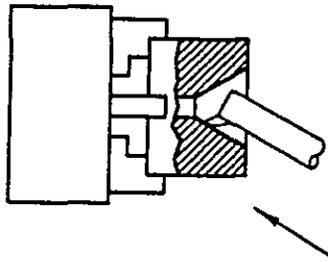


Figure 5.11 : Taper Boring Operation.

5.12 Internal Threading or Tapping

Most internal threads are cut with taps; however, there are times when a tap of a specific size is not available and the thread must be cut on a lathe. Internal threading, or cutting threads in a hole, is an operation performed on workpiece held in a chuck, collet or mounted on a faceplate. This operation is illustrated in Figure 5.12.

This operation is specified for the INTERNAL THREAD feature. The tool that has been assigned to produce this feature is internal threading tool.

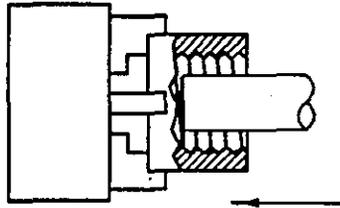


Figure 5.12 : Internal Threading Operation.

5.13 Boring

Boring is the process of enlarging a drilled hole with a single-point cutting tool as illustrated in Figure 5.13. When a hole is drilled in a workpiece in a lathe, the hole may not be concentric with the outside diameter. This occurs if the drill begins to wander due to hard spots or sand holes in the metal. If the hole location and size must be accurate, it is necessary to bore the hole so that the hole is concentric with the diameter of the workpiece. Boring is also used to finish an off-size hole for which no drill or reamer is available.

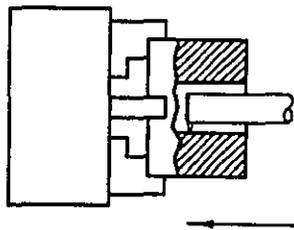


Figure 5.13 : Boring Operation.

This operation has been appointed by the prototype system for the CYLINDER BORE feature. Boring tool has been assigned to produce this feature.

5.14 Reaming

Reaming is the process of enlarging a drilled or bored hole to produce an accurately sized and shaped hole with a good surface finish. Figure 5.14 shows the reaming operation in order to produce the CYLINDER BORE feature. This operation is performed with a tool called a reamer.

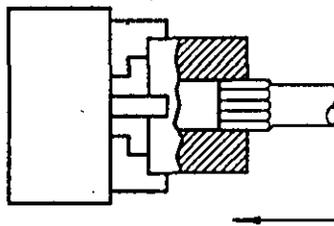


Figure 5.14 : Reaming Operation.

5.15 Summary

This chapter has discussed the machining operations which can be performed in producing rotational components. The operations that are used in the prototype system are based on the names of the features.

Chapter 6

CHAPTER 6

CADOPP SYSTEM SOFTWARE

6.1 Introduction

This chapter describes the feature-based CADOPP system which aims to provide facilities for design (geometrical specification) and outline process planning for rotational machined components.

CADOPP is an acronym of Computer-Aided Design and Outline Process Planning, and consists of several different manufacturing features which are based on different machining operations, that are turning, facing, knurling, threading, drilling, forming, chamfering, taper turning, undercutting, reaming, tapping, keyway cutting, slotting, milling, and taper boring.

The machining parameters such as operation name, machine tool and tool type are selected by the system automatically from the feature type in order to produce an outline process planning sheet.

6.2 System Description

Currently, CADOPP is oriented towards rotational machined components. The CADOPP system consists of a main program along with the subroutines of features and the modification modules. This program has been written to create CADOPP in FORTRAN-77 code, to run on an APOLLO - DN3500 workstation, under the AEGIS and UNIX operating systems.

The CADOPP software is an interactive program. The input data is entered into the system by means of a user-friendly interactive session, that allows for an easy dialogue between the user and the system. The user answers questions which are either of the multiple choice type or variable value type.

As stated before, the aims of the software are to allow the user to create the geometrical specification and an outline process plan for turned components.

The CADOPP software only allows machining from round bar stock material. The stock bar of material can therefore be defined geometrically by its length and diameter which are used to generate the cylindrical primitives in the solid model.

The material and the shape of the component are first described. The workpiece material can be drawn from the materials list or user-defined. The materials list is displayed on the screen and the user may select one for further consideration. If there is no material available from the materials list, the user chooses the "None of these" option and supplies the material type.

The user then interacts with the CADOPP software by input an appropriate values to describe the manufacturing features of the components. This will be discussed in section 6.8.

Before the concept of the prototype system is presented, it is convenient to review the proposed solid modeller that has been used as a tool in representing the solid geometry of an object and how the features are represented in the prototype system.

6.3 Solid Modeller

BOXER [PAFEC 1986], a true solid modeller based on Constructive Solid Geometry (CSG) representation is used in representing the solid model of an object. The model is built up by the user defining the type of primitives required, the key dimensions, its position relative to the origin (or relative to its current position) and its relationship with primitives already created.

The BOXER CSG has been selected to represent the output solid object model not only because a CSG representation provides an effective and compact structure to define a solid object, but also because the "DIFFERENCE" operation employed to generate the solid model is analogous to manufacturing processes.

In this solid modeller, the primitives shapes (as illustrated in Figure 3.4) are defined in a base coordinate system and are positioned

by a transformation (translation or rotation). For example, a BLOCK is defined by the distance of each side of the block along the respective axis. The syntax of specifying a BLOCK is as follows :

BLOCK (*X length, Y height, Z width*)

A cylinder is defined by its height and by the radius of the cylinder.

Syntax : **CYL** (*height, radius*)

A cone is defined by its length or height and by the minor-radius and major radius of the cone.

Syntax : **CONE** (*length, minor-radius, major-radius*)

A sphere is defined by its radius,

Syntax : **SPHERE** (*radius*)

A TORUS is defined by its minor-radius and major-radius.

Syntax : **TORUS** (*minor-radius, major-radius*)

All these primitives can be positioned and oriented by used of the "AT" qualifier. This takes the form :

primitives AT position

where position locates a primitives in relation to the BOXER coordinate system.

Position has two forms :

(*x, y, z*) or (*x, y, z, motion element list*)

where x, y, z are scalars giving the coordinates of the reference point of the primitive, and a motion element is as follows :

BLOCK (*L,H,W*) at (**MOVEX** = *x* , **MOVEY** = *y* , **MOVEZ** = *z*)

6.3.1 Definition Statement

Suppose that we wished to create an instance of the primitive "CYLINDER", measuring 50 units and 5 units in terms of its length or height and radius along X-axis respectively. The reference point for a cylinder is at the origin of the axis, i.e. (0,0,0). We could do it by the statement :

```
STOCK <- CYLINDER (50,5) AT (0,0,0,ROTY=90)
```

The word "STOCK" is the variable name which has been defined by the user, and is usually descriptive. The symbol "<-" is the assignment symbol for a solid object. This statement could be done in BOXER's text input mode.

The software for the text input mode is developed in the Mechanical Engineering Department at Leeds University by the Geometric Modelling Project [NONAME 1986].

Further explanations in the following sections describe how the solid model of an object can be constructed by using special statements in the text input mode.

6.3.2 SCOPES

When a shape of a model is frequently required, or occurs more than once with different dimensions, it is useful to "hide" parts of the definition away. BOXER provides "SCOPES" to achieve this.

A "SCOPE" can be considered as an area where objects and variables can be defined separately from the rest of the definitions and can be accessed from within the scope in which they are defined. Scope statements for each feature in the prototype system are illustrated in Appendix I :

Any statement typed subsequently belongs to this scope and can only be accessed directly when in this scope. All definitions will be stored under the scope last set by a scope statement.

There is a special scope, the variables of which can be accessed from any other scope - **SCOPE COMMON**. This is the scope set on entry to **BOXER**.

Objects and variables in **SCOPE COMMON** can be accessed by prefixing them with a circumflex "**^**". Based on the previous scope statements as in Appendix I, the **SCOPE COMMON** statement can be generated as follows :

```
SCOPE COMMON  
STOCK <- ^ MATERIAL(LENGTH,RADIUS)  
CYLIN1 <- ^ CYLINDER(LENGTH,RADIUS1,X1,Y1,Z1,$  
X2,Y2,Z2)  
TAPER1 <- ^ RIGHT_TAPER(LENGTH,RADIUS1,RADIUS2,$  
X1,Y1,Z1,X2,Y2,Z2)  
COMPONENT <- STOCK-CYLIN1-TAPER1
```

By using this **SCOPE COMMON** and the related scopes, the solid model of an object can be constructed in the **BOXER** solid modeller.

Appendix II illustrated how **SCOPE COMMON** and other scopes can be generated and how these scopes are used to create the model in the **BOXER** system.

6.4 Feature Representation in the Prototype System

In this prototype system, the technique that has been used in creating the geometry specification and an outline process plan of turned components is the designing with features approach of Destructive Solid Geometry (DSG). The features are represented in the DSG tree representation because this representation can be used directly by a feature-based manufacturing system [PERNG 1990].

For example, consider the process of turning a cylinder, tapering a taper and chamfering a chamfer on a stock of material as shown in Figure 6.1.

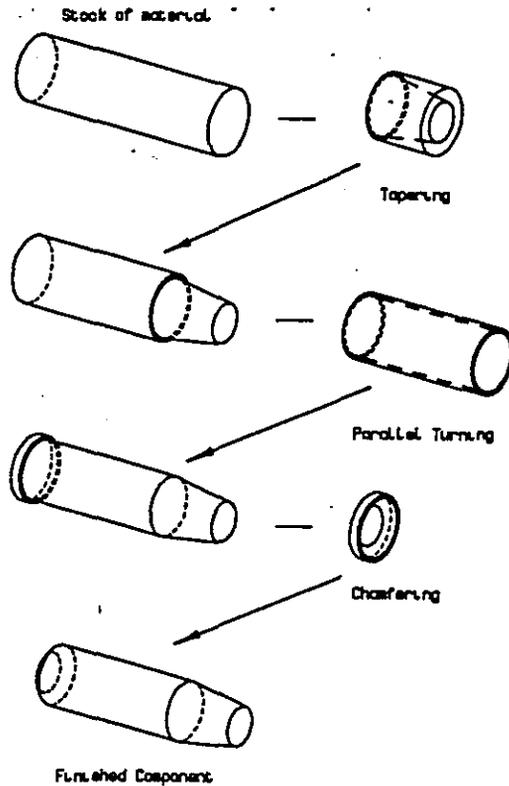


Figure 6.1 : An Example to Produce a Rotational Component Based on DSG Technique.

6.5 Prototype System Data Structure

The prototype data structure is quite similar to the CSG data structure, except that the "DIFFERENCE" operator in the operator node is used. Figure 6.2 shows us how the model can be created using DSG technique. Each feature corresponds to a cavity volume to be removed by machining. A cavity volume is the solid volume of material to be removed from the stock material by one or more material removal machining operations [ANSALDI,1988]. When a component is designed using a DSG model, there is an analogy between the modelling sequence and the machining sequence.

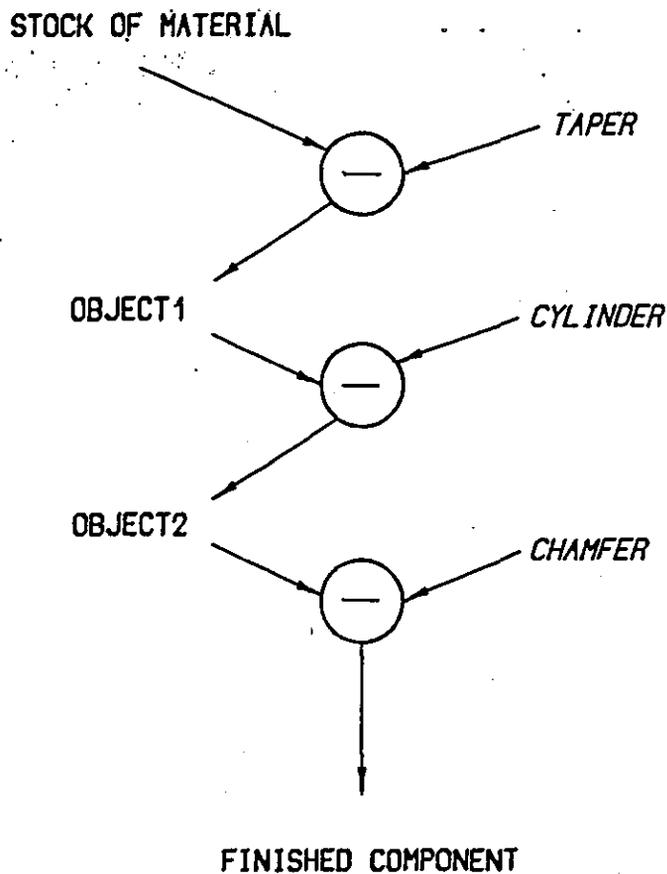


Figure 6.2 : Data Structure of DSG Model of an Example.

6.6 System Concept

The conceptual design of the CADOPP software is illustrated in Figure 6.3. It has the following specifications:

(i). Password - to enable only authorized users to have access to the system.

The user accesses the system by typing "CADOPP" and the system then prints "CADOPP" on the terminal screen followed by the first question :

" Enter password :"

If the user enters the wrong password, the system will give an error password statement and request the user to re-enter the password.

(ii). User information - to enable the user to know what the system is all about.

After the user has entered the password, he/she is offered user information.

" Do you need the user information ? (y/n) "

If the user is unfamiliar with the system, he/she will answer "y" and will be presented with information that describes the system itself. Appendix III illustrates the CADOPP user information.

(iii). System modules

Currently, the CADOPP system consists of four (4) modules. Each of these modules will be further discussed in the following sections.

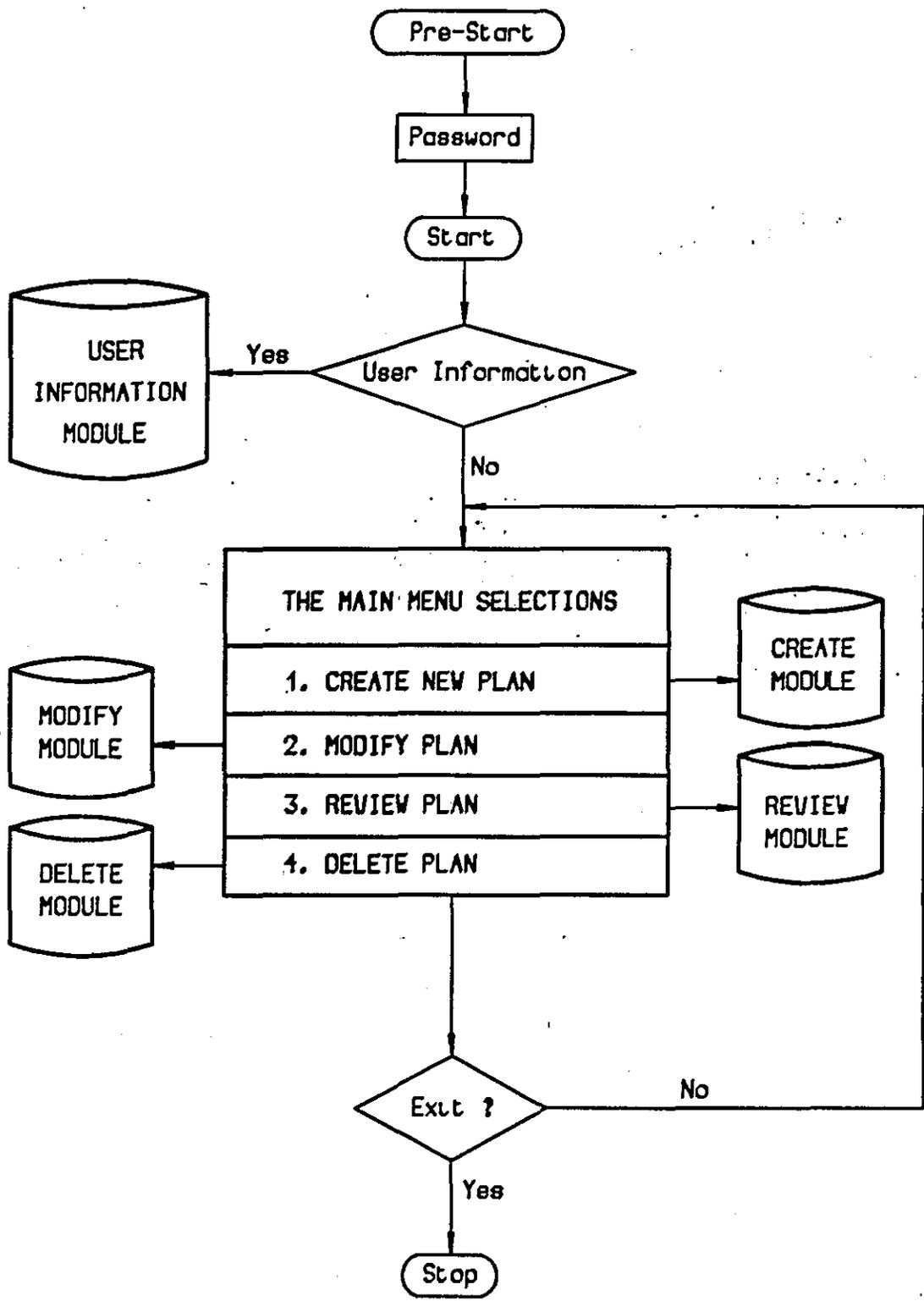


Figure 6.3 : The Conceptual Design of the CADOPP System.

6.7 CADOPP Interactive System Procedures

The CADOPP system has been designed as an interactive system so that the computer and the user can share the various tasks to provide the information for design and process planning.

Interaction with CADOPP can be divided into four distinct actions:

- (i). Creation of a New Design and Plan,
- (ii). Modification of an Existing Design and Plan,
- (iii). Review of an Existing Design and Plan,
- (iv). Deletion of an Existing Design and Plan.

All of these will be discussed in detail in the following sections.

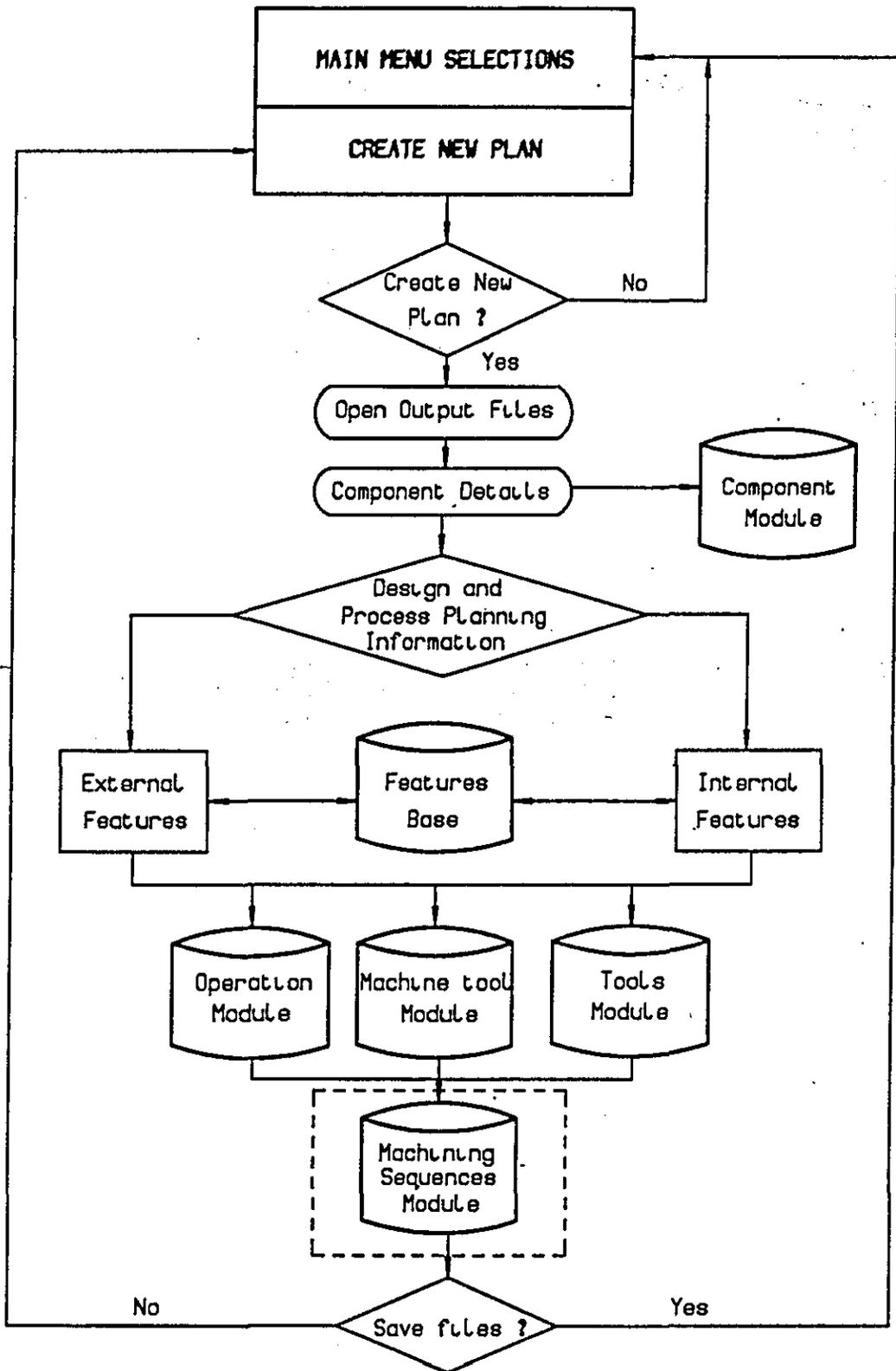
6.8 Interactive Creation of a New Design and Plan

This is the stage where the user specifies the component features parametrically. The user takes the sketch detail drawing of the rotational component. The user is first asked for information about the workpiece material and its dimensions in terms of length and diameter (in mm).

The workpiece material is normally selected from the list supplied by CADOPP, but facilities are provided for the user to supply an alternative textual description of the material.

Then the user has to decide in which order the feature should be machined. These features with their related information are described below.

All the input necessary for design and process planning is entered into the system by a comprehensive dialogue between the user and the computer. For every stage of input, the associated question which explains the input is prompted and the system waits for the entry of data. Figure 6.4 shows the create module structure of the system.



Dashed lines shown mean that they are not implemented in the system at the moment.

Figure 6.4 : Create Module Structure of the System.

6.8.1 Features Input

The input defines the dimensions and the locations of the features. Each feature has its own requirements. The input procedure starts by selecting one of the features below :

i. For axial external features :

- | | |
|-------------|-----------|
| 1. CYLINDER | 2. FACE |
| 3. TAPER | 4. KNURL |
| 5. PINION | 6. SPLINE |

ii. For surface external features :

- | | |
|----------------|-----------------|
| 1. CROSS-HOLE | 2. UNROUND-HOLE |
| 3. ACROSS-FLAT | 4. KEYWAY |

iii. For radial external features :

- | | |
|-------------|------------|
| 1. UNDERCUT | 2. FORMCUT |
| 3. ROUND | 4. CHAMFER |

iv. For axial internal features :

- | | |
|--------------------|---------------|
| 1. CYLINDER-BORE | 2. TAPER-BORE |
| 3. INTERNAL-THREAD | |

v. For surface internal features :

- | | |
|--------------------|-----------|
| 1. INTERNAL-KEYWAY | 2. RECESS |
|--------------------|-----------|

vi. For radial internal features :

- | | |
|-----------------|---------------------|
| 1. CENTRE-DRILL | 2. INTERNAL-CHAMFER |
|-----------------|---------------------|

After completing the input for each feature, the user may change the input values in case of misentering. Alternatively, input can be cancelled totally by issuing a "CANCEL" command.

The following example shows the dialogue between the user and the system for changing the feature parameters.

1: Enter the LENGTH of cylinder :

50

2: Enter the RADIUS of cylinder :

20

3: Enter the POSITION of cylinder :

090

The parameters for CYLINDER feature are as follows :

1. Length	= 50
2. Radius	= 20
3. Position	= 090

?: Any data to correct ? (y/n/cancel)

Y

?: Give the information number to correct :

1

1: Enter the LENGTH of cylinder :

60

The parameters for CYLINDER feature are as follows :

1. Length	= 60
2. Radius	= 20
3. Position	= 090

?: Anymore data to correct ? (y/n)

Y

?: Give the information number to correct :

2

2: Enter the RADIUS of cylinder :

22

The parameters for CYLINDER feature are as follows :

1. Length = 60
2. Radius = 22
3. Position = 090

?: Anymore data to correct ? (y/n)

n

The following sections describe the input for each feature.

6.8.1.1 Cylinder

Cylinders are processed by a turning operation on a lathe machine. The CADOPP assigned tool for the features is "Turning". The necessary inputs for cylinders are as follows :

1. LENGTH
2. DIAMETER
3. POSITION

6.8.1.2 Face

Faces are processed by a turning operation on a lathe machine. The tool used for the features is "Turning". The necessary inputs for faces are as follows :

1. LENGTH
2. DIAMETER
3. POSITION

6.8.1.3 Taper

Tapers can be divided into two types :

- i. External Taper,
- ii. Internal Taper.

Tapers are processed by a turning operation on a lathe machine. For internal taper features, a drilling operation must be done before the taper operation itself. The tool used for the features is "Turning". Both types of the taper have the same inputs. The necessary inputs for both tapers are as follows :

1. LENGTH
2. MINOR-DIAMETER
3. MAJOR-DIAMETER
4. POSITION

6.8.1.4 Knurl

Knurls are processed by a turning operation on a lathe machine using a special tool. The tool used for the features is "Knurling". The necessary inputs for knurls are as follows :

1. LENGTH
2. DIAMETER
3. TYPE OF KNURL (PATTERN)
4. POSITION

These features are represented as a cylinder feature in the geometrical specification system with the name as knurl feature. However, the information of these features can be used in producing an outline process plan sheet.

6.8.1.5 Thread

Threads can be divided into two types :

- i. External Thread,
- ii. Internal Thread.

Threads are processed by a turning operation on a lathe machine. The tool used for the features is "Threading". Both types of the thread have the same inputs as follows :

1. LENGTH
2. NOMINAL-DIAMETER
3. PITCH
4. TYPE OF THREAD (PATTERN)
5. POSITION

These features also are represented as a cylinder feature in the geometrical specification system with the name as thread feature. However, the information of these features can be used in producing an outline process plan sheet.

6.8.1.6 Pinion and Spline

The pinions and splines are processed by a milling operation on a milling machine. The tool used for the features is "Gear Cut". Both of them have the same inputs. The necessary inputs for pinions and splines are as follows :

1. LENGTH
2. MINOR-DIAMETER
3. MAJOR-DIAMETER
4. POSITION

These features are not implemented in the geometrical specification system. However, the information of these features can be used in producing an outline process plan sheet.

6.8.1.7 Cross Hole

Cross holes are processed by a drilling operation on lathe machine. The tool used for the features is "Drill". The necessary inputs for cross holes are as follows :

1. DIAMETER
2. POSITION

6.8.1.8 Unround Hole

Unround holes are processed by a milling operation on milling machine. The tool used for the features is "Mill". The necessary inputs for unround holes are as follows :

1. LENGTH
2. RADIUS
3. DEPTH
4. POSITION

These features are not implemented in the geometrical specification system. However, the information of these features can be used in producing an outline process plan sheet.

6.8.1.9 Across Flat

Across flats are processed by a milling operation on milling machine. The tool used for the features is "Mill". The necessary inputs for across flat are as follows :

1. LENGTH
2. WIDTH
3. DEPTH
4. POSITION

These features are not implemented in the geometrical specification system. However, the information of these features can be used in producing an outline process plan sheet.

6.8.1.10 Keyway

Keyways can be divided into two types :

- i. External Keyway,
- ii. Internal Keyway.

Keyways are processed by a milling operation on milling machine. To produce an internal keyway, a drilling operation must take place first. The tool used for internal keyways is a special tool "Broach". The tool to produce an external keyways is "Mill". The

necessary inputs for both keyways are as follows :

1. LENGTH
2. WIDTH
3. DEPTH
4. POSITION

These features are not implemented in the geometrical specification system. However, the information of these features can be used in producing an outline process plan sheet.

6.8.1.11 Undercut

Undercuts are processed by a turning operation on lathe machine. The tool used for the features is "End Cut". The necessary inputs for undercuts are as follows :

1. LENGTH
2. DIAMETER
3. POSITION

6.8.1.12 Formcut

Formcuts are processed by a turning operation on lathe machine. The tool used for the features is "End Cut". The necessary inputs for formcuts are as follows :

1. LENGTH
2. RADIUS
3. POSITION

These features are not implemented in the geometrical specification system. However, the information of these features can be used in producing an outline process plan sheet.

6.8.1.13 Round

Rounds are processed by a turning operation on lathe machine. The tool used for the features is "Form". The necessary inputs for rounds are as follows :

1. RADIUS
2. POSITION

These features are not implemented in the geometrical specification system. However, the information of these features can be used in producing an outline process plan sheet.

6.8.1.14 Chamfer

Chamfers can be divided into 2 types :

- i. External-Chamfer,
- ii. Internal-Chamfer.

Chamfers are processed by a turning operation on a lathe machine. Internal chamfers need to be drilled before the chamfering operation itself. The tool used for the features is "Forming". Both types of the chamfer have the same inputs. The necessary inputs for both chamfers are as follows :

1. LENGTH
2. MINOR-DIAMETER
3. MAJOR-DIAMETER
4. ANGLE
5. POSITION

6.8.1.15 Cylinder Bore

Cylinder bores are processed by a drilling operation on lathe machine. The tool used for the features is "Boring". The necessary inputs for cylinder bores are as follows :

1. LENGTH
2. DIAMETER
3. POSITION

6.8.1.16 Centre Drill

Centre drills are processed by a drilling operation on lathe machine. The tool used for the features is "Centre Drill". The

necessary inputs for centre drills are as follows :

1. LENGTH
2. DIAMETER
3. TYPE OF CENTRE DRILL (PATTERN)
4. POSITION

6.8.1.17 Recess

Recesses are processed by a reaming or boring operation on a lathe machine. The tool used for the features is a "Reamer". The necessary inputs for recesses are as follows :

1. LENGTH
2. DIAMETER
3. POSITION

This research work has adopted a volumetric feature-based strategy [CHANG,1990] in order to select the processes, and resources (machines, tools, etc). In other words, the geometric capability of the manufacturing process is described using the names of the volumetric features.

6.9 Modification of an Existing Design and Plan

After the geometric model output file and the process sheet, with all the necessary information for the manufacturing of the given part, are prepared by the CADOPP system, they can be modified manually by the user.

The system asks for the old and new name of the related files and the user has to type-in both filenames before the system asks for the type of modifications.

The user is allowed to delete existing processes, insert or add new processes or change the feature information. An interactive procedure to facilitate the user in selecting processes which he/she wants to manipulate have been developed. All these modes will be discussed in the following sections. Figure 6.5 shows the modify module structure of the system.

6.9.1 Deletion of Existing Processes

In this mode, a list of processes currently available are displayed for user selection. Those selected processes are deleted from the data files. Appendix IV illustrates an interactive delete of existing processes.

6.9.2 Insertion /Addition of New Processes

In this mode, the user can select processes from a list of machining operations and insert/add them to the new files. Currently, eighteen operations based on features are available for selection. Appendix V illustrates an interactive insert/add of new processes.

6.9.3 Change the Feature Information

In this mode, the user can select any parameters that he/she wants to change and input new value of that parameter. Appendix VI illustrates an interactive change the feature information.

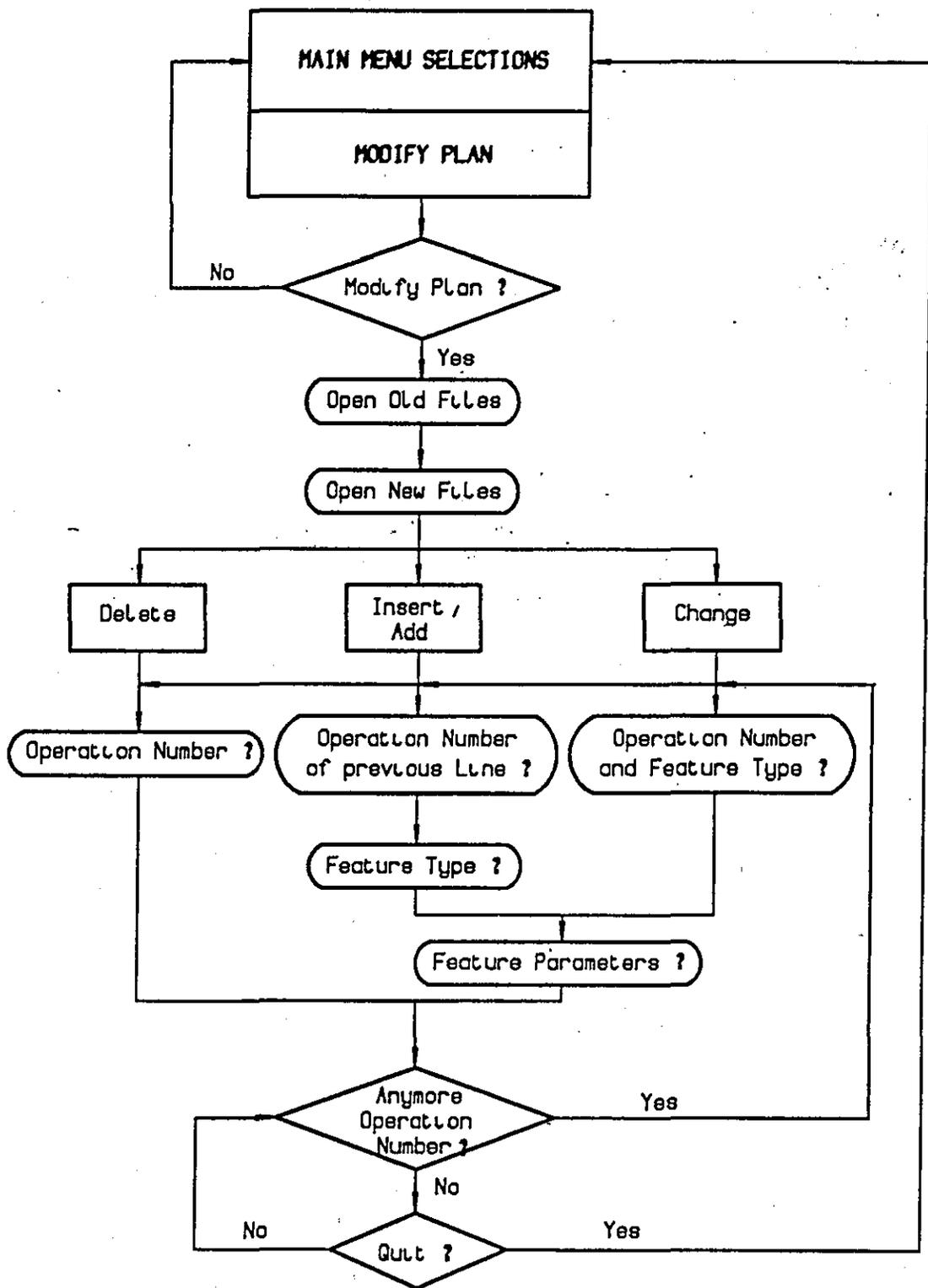


Figure 6.5 : Modify Module Structure of the System.

6.10 Review of an Existing Design and Plan

Both files can be reviewed by the user enter [R] key for review in the main menu selections of the system. The system will ask for the name of the file to be reviewed.

The system allows the user to review another file. Figure 6.6 shows the review module structure of the system.

6.11 Deletion of an Existing Design and Plan

Both files can be deleted by the user enter [D] key for delete in the main menu selections of the system.

The system allows the user to delete another file. Figure 6.7 shows the delete module structure of the system.

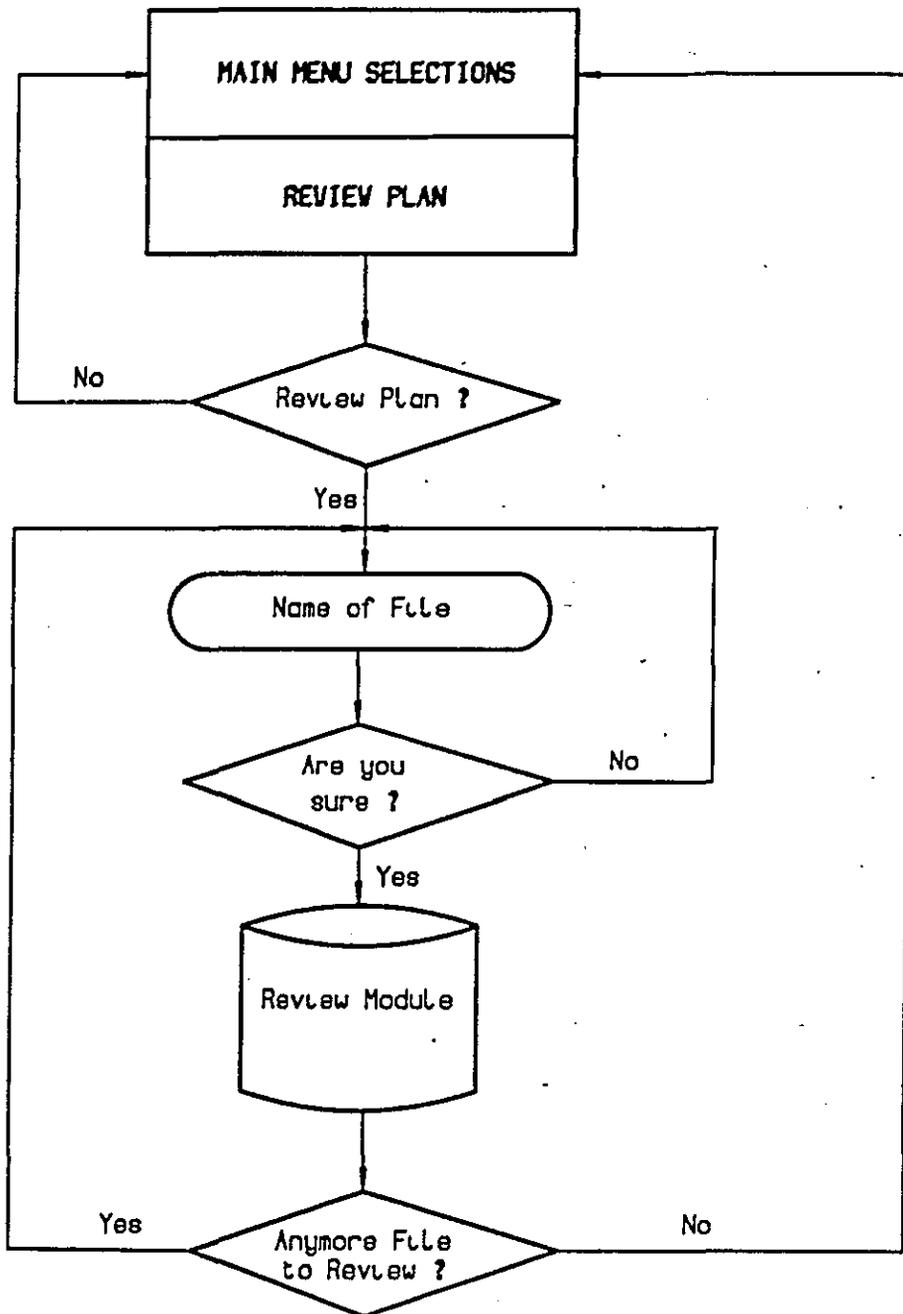


Figure 6.6 : Review Module Structure of the System.

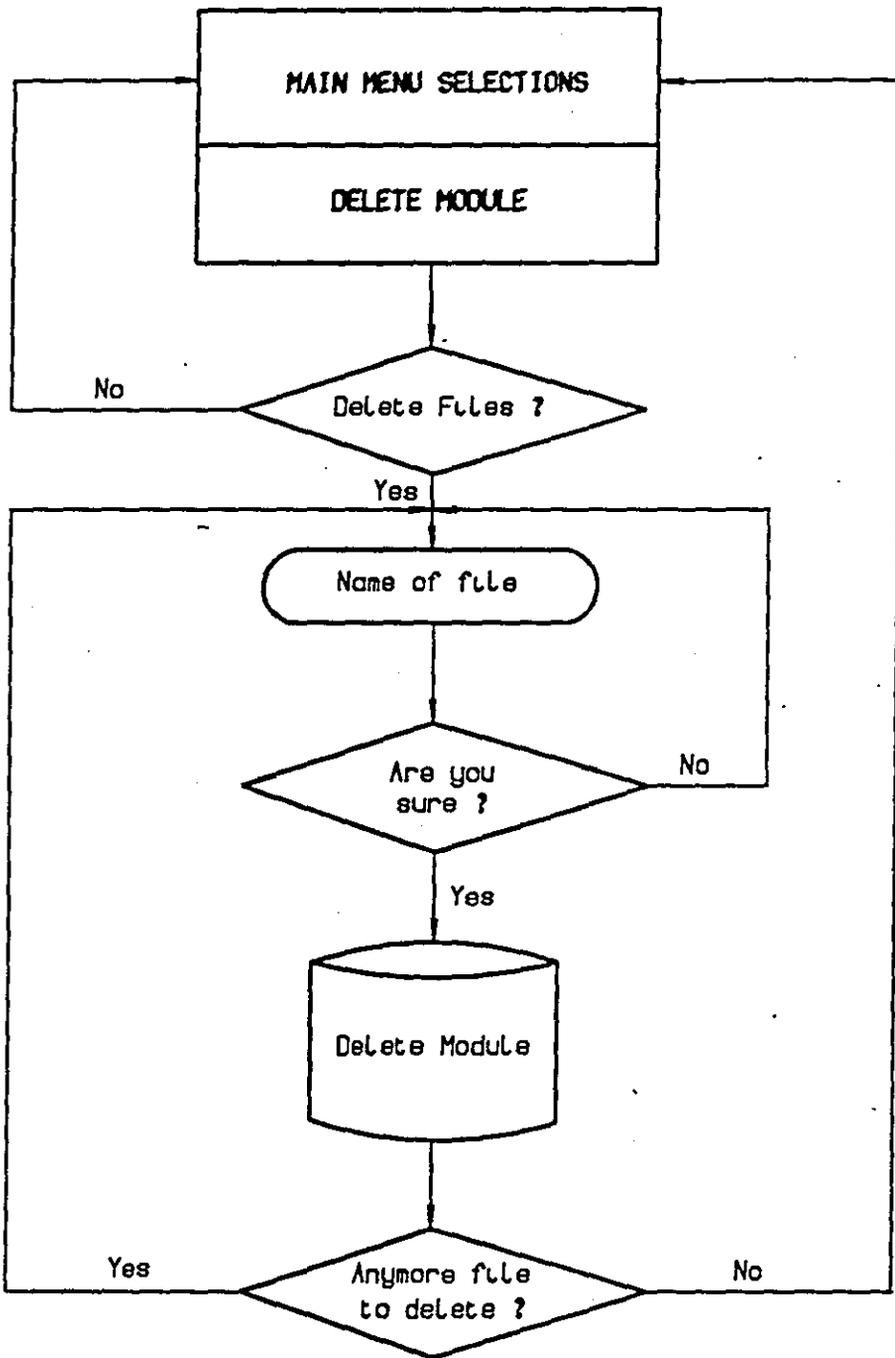


Figure 6.7 : Delete Module Structure of the System.

6.12 Summary

This chapter has explained in detail the CADOPP system software and representation of features in order to produce an outline process plan and the geometrical specification files.

The implementation of the system is presented in Chapter 7.

Chapter 7

CHAPTER 7

IMPLEMENTATION AND TEST RESULTS

7.1 Introduction

The development of the prototype system has been discussed in the previous chapter. In order to demonstrate the capabilities of the system, three samples of rotational components based on the use of simple geometrical shapes have been chosen. The results of using CADOPP on these samples are discussed in this chapter. The following sections give a demonstration of how the system works.

7.2 Sample 1.

The first sample belongs to the external features group and is illustrated in Figure 7.1.

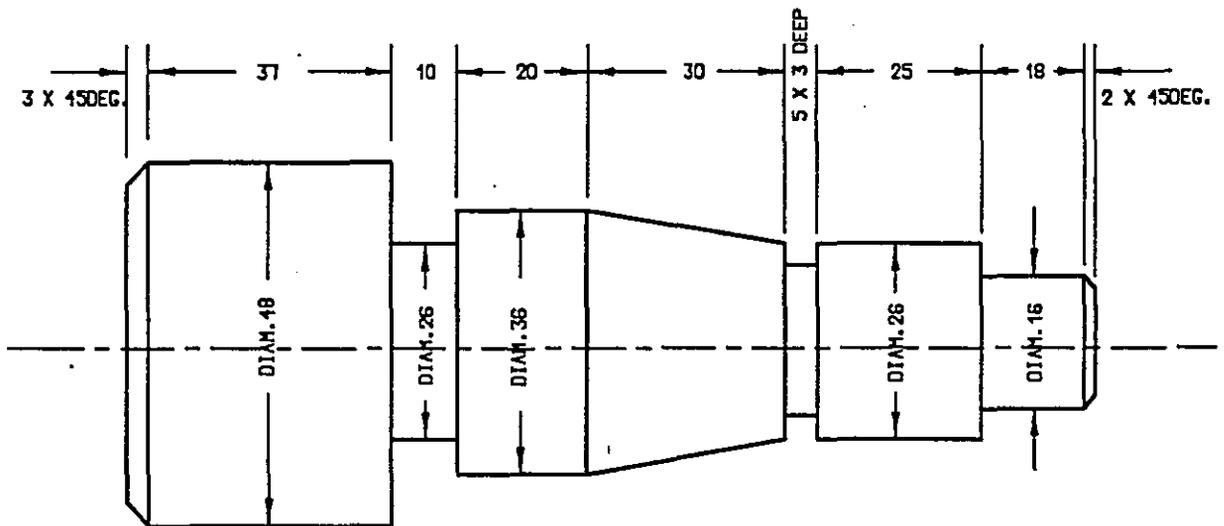


Figure 7.1 : Sample of External Features Component.

There are nine external features from the left end to the right end that need to be cut into this sample component. The features are 1 left-chamfer, 4 cylinders, 2 undercuts, 1 right-taper, and 1

right-chamfer. The sample is represented in the system as a cylindrical bar from which the cavities are subtracted.

The procedures adopted in order to produce the sample are described below.

Firstly, the user types "CADOPP" to gain access to the system. This will be followed by entering the password of the system. The system starts by asking for user information and THE MAIN MENU SELECTIONS OF THE SYSTEM is then displayed.

The user needs to enter [C] for creating new design and plan and must give the names of both output files required, that are the geometry specification and an outline process plan output files. The system requests the workpiece material and dimensions. The FEATURES MENU is then displayed for the user to select the features. The features should be selected from the right end to the left end. Since this sample has only external features, only the EXTERNAL FEATURES menu is chosen. The input of the dimensions and the location of the features is prompted for each feature type chosen.

When all the input of the features has been performed, the user needs to enter the [O] key to save the files and this will return the user to THE MAIN MENU SELECTIONS OF THE SYSTEM for further actions.

An interactive session in order to produce this sample of component is illustrated in Appendix VII.

7.3 Sample 2

The second sample belongs to the internal features group and this is illustrated in Figure 7.2. There are five internal features from the left end to the right end in order to produce this sample component. The features are 1 left-internal-chamfer, 2 cylinder-bores, 1 right-taper-bore and 1 right-internal-chamfer.

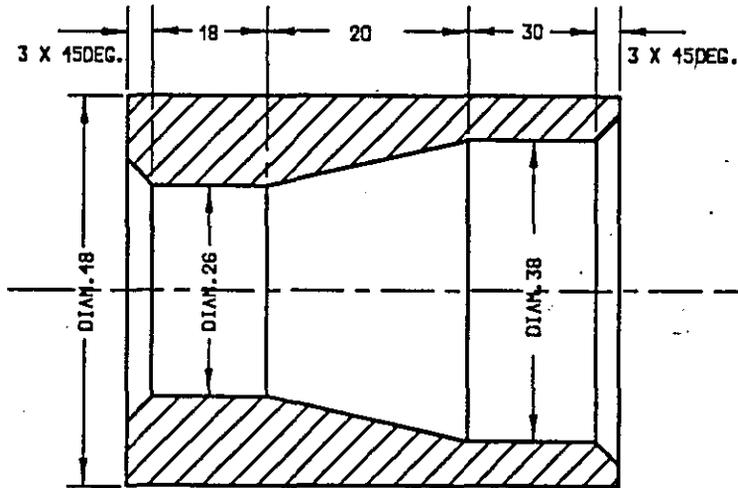


Figure 7.2 : Sample of Internal Features Component.

The procedures in producing this sample of component are similar with the previous sample except the INTERNAL FEATURES menu is chosen as this sample has only internal features. An interactive session to produce this sample of component is illustrated in Appendix VIII.

7.4 Sample 3

The third sample is a combination of the external and the internal features. This is illustrated in Figure 7.3. This sample consists of nine features from the left end to right end that need to be cut into this sample of component. The features are 1 left-chamfer, 2 cylinders, 1 right-chamfer, 2 undercuts, 1 external-thread, 1 cylinder-bore and 1 right-taper-bore.

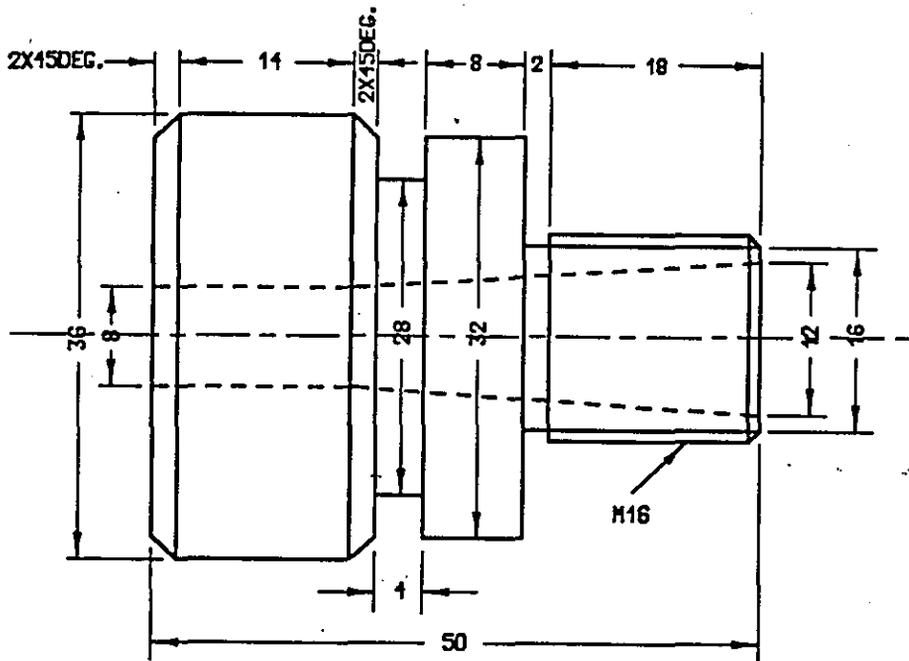


Figure 7.3 : Sample of Combination of External and Internal Features Component.

Because this sample is a combination of the external and internal features, therefore in producing this sample, firstly the user has to choose all the external features and then the internal features.

The procedures are as in Sample 1 and Sample 2, and an interactive session to produce this sample of component is illustrated in Appendix IX.

7.5 Test Results

From the above demonstrations, the CADOPP system can be used in producing the geometric specifications and outline process plans. Several components have been tested. The components tested were rotational components consisting of several external and internal features.

Sample 1 belongs to the external features group. The purpose of selecting this sample is to demonstrate how CADOPP generates the model of a component and a list of manufacturing processes for external features.

The corresponding geometric specification and an outline process plan files for the first sample which is carried out by the CADOPP system, are shown in Table 3 and Table 4 respectively.

```
SCOPE COMMON
STOCK<-^MATERIAL(150,25)
LCHM1<-^LEFT_CHAMFER(03,21,24,0,0,0,003,0,0)
CYLN1<-^CYLINDER(37,24,0,0,0,003,0,0)
UCUT1<-^UNDERCUT(10,13,0,0,0,040,0,0)
CYLN2<-^CYLINDER(20,18,0,0,0,050,0,0)
RTAP1<-^RIGHT_TAPER(30,13,18,0,0,0,070,0,0)
UCUT2<-^UNDERCUT(05,10,0,0,0,100,0,0)
CYLN3<-^CYLINDER(25,13,0,0,0,105,0,0)
CYLN4<-^CYLINDER(18,08,0,0,0,130,0,0)
RCHM1<-^RIGHT_CHAMFER(02,06,08,0,0,0,148,0,0)
COMPONENT<-STOCK-LCHM1-CYLN1-UCUT1-CYLN2-RTAP1-UCUT2-CYLN3$
-CYLN4-RCHM1
!
! MATERIAL
!
SCOPE MATERIAL
SOLID FAMILY MATERIAL(LENGTH,RADIUS):MATERIAL
MATERIAL<-CYL(LENGTH,RADIUS)AT(ROTY=90)
RADIUS=25
!
! CYLINDER
!
SCOPE CYLINDER
SOLID FAMILY CYLINDER(LENGTH,RADIUS1,X1,Y1,Z1,$
X2,Y2,Z2):CYLINDER
```

Table 3 : The Geometric Specification File of Sample 1.

```

OBJ1<-CYL(LENGTH,RADIUS) AT (ROTY=90)
OBJ2<-CYL(LENGTH,RADIUS1) AT (ROTY=90)
CYLINDER<-MOVE(OBJ1-OBJ2) BY (X2,0,0)
RADIUS=25
!
! RIGHT_TAPER
!
SCOPE RIGHT_TAPER
SOLID FAMILY RIGHT_TAPER(LENGTH,RADIUS1,$
RADIUS2,X1,Y1,Z1,X2,Y2,Z2):RIGHT_TAPER
OBJ1<-CYL(LENGTH,RADIUS) AT (ROTY=90)
OBJ2<-CONE(LENGTH,RADIUS1,RADIUS2) AT (ROTY=90)
RIGHT_TAPER<-MOVE(OBJ1-OBJ2) BY (X2,0,0)
RADIUS=25
!
! UNDERCUT
!
SCOPE UNDERCUT
SOLID FAMILY UNDERCUT(LENGTH,RADIUS1,X1,Y1,Z1,$
X2,Y2,Z2):UNDERCUT
OBJ1<-CYL(LENGTH,RADIUS) AT (ROTY=90)
OBJ2<-CYL(LENGTH,RADIUS1) AT (ROTY=90)
UNDERCUT<-MOVE(OBJ1-OBJ2) BY (X2,0,0)
RADIUS=25
!
! RIGHT_CHAMFER
!
SCOPE RIGHT_CHAMFER
SOLID FAMILY RIGHT_CHAMFER(LENGTH,RADIUS1,RADIUS2,$
X1,Y1,Z1,X2,Y2,Z2):RIGHT_CHAMFER
OBJ1<-CYL(LENGTH,RADIUS) AT (ROTY=90)
OBJ2<-CONE(LENGTH,RADIUS1,RADIUS2) AT (ROTY=90)
RIGHT_CHAMFER<-MOVE(OBJ1-OBJ2) BY (X2,0,0)
RADIUS=25
!
! LEFT_CHAMFER
!
SCOPE LEFT_CHAMFER
SOLID FAMILY LEFT_CHAMFER(LENGTH,RADIUS1,RADIUS2,$
X1,Y1,Z1,X2,Y2,Z2):LEFT_CHAMFER
OBJ1<-CYL(LENGTH,RADIUS) AT (ROTY=270)
OBJ2<-CONE(LENGTH,RADIUS1,RADIUS2) AT (ROTY=270)
LEFT_CHAMFER<-MOVE(OBJ1-OBJ2) BY (X2,0,0)
RADIUS=25

```

Table 3 : Continued.

Part material details : Medium Carbon Steel Round Bar

Geometry : A round bar of dimensions L150 x R25 (mm)

Op.no	Oper.name	Machine	Tool-type	Lgth	Rad	MinRad	MajRad	Pic	Wid	Dep	Ang	Pattn
00	Stock			150	25							
01	RChamfer	Lathe	Forming	02		06	08					45
02	Cylinder	Lathe	Turning	18	08							
03	Cylinder	Lathe	Turning	25	13							
04	Undercut	Lathe	End Cut	05	10							
05	RTaper	Lathe	Turning	30		13	18					
06	Cylinder	Lathe	Turning	20	18							
07	Undercut	Lathe	End Cut	10	13							
08	Cylinder	Lathe	Turning	37	24							
09	LChamfer	Lathe	Forming	03		21	24					45

Table 4 : An Outline Process Plan of Sample 1.

The geometry specification file includes the values of each feature with the reference co-ordinates. The model of a component is created from the equation that is based on the destructive solid geometry technique which has been developed in the system.

The outline process planning file include the operations which are based on feature types, the machine tools and the necessary tools which has been recommended by CADOPP. The corresponding attributes for each feature are all assigned by CADOPP, that is, diameter, length, pattern type, etc.

The criteria mechanisms have been established in the system. As mentioned earlier in Chapter 6, the dimensions of the stock material are first described. Then the feature's attributes are keyed. This system allows the user to re-enter the value of the features if that value is larger than the value of the stock material. CADOPP will automatically tell the user that the value is invalid.

The 3-D output drawing of Sample 1 is shown in Figure 7.4. This output model has been produced by transferring the file of geometric specification from BOXER to DOGS-2D because the BOXER does not have any printing facilities.

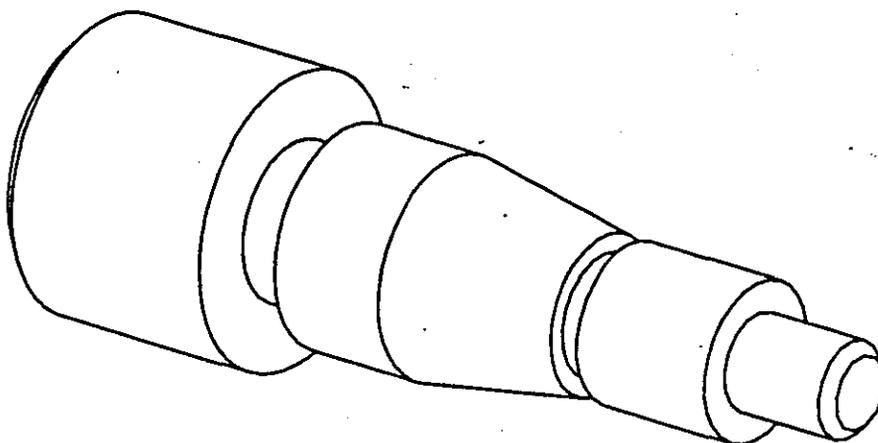


Figure 7.4: The 3-D Output Drawing of Sample 1.

Sample 2 belongs to the internal features group. The system has generated the model of the component and produced a list of manufacturing processes for the internal features. The internal features have been represented by a dotted lines as shown in Figure 7.5.

Table 5 and Table 6 respectively show the corresponding geometry specification and an outline process plan files for the second example which is carried out by the CADOPP system.

The output of geometric specification and outline process planning for this sample are similar with Sample 1 where the values of each features with their reference co-ordinates are presented.

The outline process planning file for Sample 2 consists of the operations in order to produce the sample. This clearly shown that the system was also capable of producing the output of geometric specification and outline process planning for internal features group.

```

SCOPE COMMON
STOCK<-^MATERIAL(074,25)
RICH1<-^R_INT_CHAMFER(03,19,22,0,0,0,074,0,0)
CBOR1<-^CYLINDER_BORE(20,19,0,0,0,051,0,0)
RITP1<-^R_TAP_BORE(30,13,19,0,0,0,051,0,0)
CBOR2<-^CYLINDER_BORE(18,13,0,0,0,003,0,0)
LICH1<-^L_INT_CHAMFER(03,13,16,0,0,0,000,0,0)
COMPONENT<-STOCK-RICH1-CBOR1-RITP1-CBOR2-LICH1
!
! MATERIAL
!
SCOPE MATERIAL
SOLID FAMILY MATERIAL(LENGTH,RADIUS):MATERIAL
MATERIAL<-CYL(LENGTH,RADIUS)AT(ROTY=90)
RADIUS=25
!
! CYLINDER_BORE
!
SCOPE CYLINDER_BORE
SOLID FAMILY CYLINDER_BORE(LENGTH,RADIUS1,$
X1,Y1,Z1,X2,Y2,Z2):CYLINDER_BORE

```

Table 5 : The Geometric Specification File of Sample 2.

```

OBJ1<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
CYLINDER_BORE<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=25
!
! R_TAP_BORE
!
SCOPE R_TAP_BORE
SOLID FAMILY R_TAP_BORE (LENGTH, RADIUS1, RADIUS2, $
X1, Y1, Z1, X2, Y2, Z2) :R_TAP_BORE
OBJ1<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=270)
R_TAP_BORE<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=25
!
! R_INT_CHAMFER
!
SCOPE R_INT_CHAMFER
SOLID FAMILY R_INT_CHAMFER (LENGTH, RADIUS1, $
RADIUS2, X1, Y1, Z1, X2, Y2, Z2) :R_INT_CHAMFER
OBJ1<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=270)
R_INT_CHAMFER<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=25
!
! L_INT_CHAMFER
!
SCOPE L_INT_CHAMFER
SOLID FAMILY L_INT_CHAMFER (LENGTH, RADIUS1, $
RADIUS2, X1, Y1, Z1, X2, Y2, Z2) :L_INT_CHAMFER
OBJ1<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=90)
L_INT_CHAMFER<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=25

```

Table 5 : Continued.

Part material details : Aluminium Round Bar

Geometry : A round bar of dimensions L074 x R25 (mm)

Op.no	Oper.name	Machine	Tool-type	Lgth	Rad	MinRad	MajRad	Pic	Wid	Dep	Ang	Pattern
00	Stock			074	25							
01	RlChamfer	Lathe	In.Chmfer	03		19	22				45	
02	Cyl.Bore	Lathe	Boring	20	19							
03	RTapBore	Lathe	Tap.Bore	30		13	19					
04	Cyl.Bore	Lathe	Boring	18	13							
05	LlChamfer	Lathe	In.Chmfer	03		13	16				45	

Table 6 : An Outline Process Plan of Sample 2.

The 3-D output drawing of Sample 2 is shown in Figure 7.5.

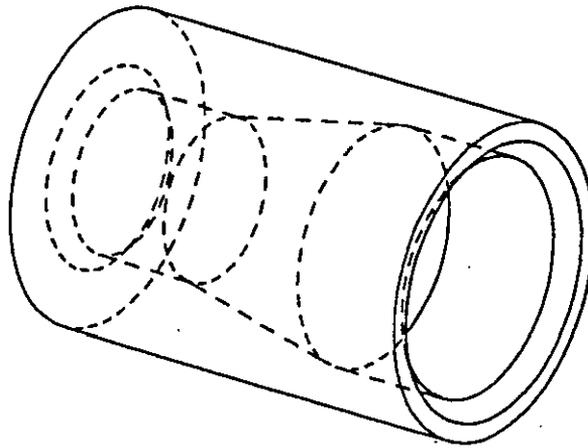


Figure 7.5 : The 3-D Output Drawing of Sample 2.

For sample 3, the component consists of both external and internal features in order to show that the system was also capable of processing both features.

In producing this sample, the procedure that has been taken is similar with the previous samples where the external feature input should be keyed first then followed by internal features.

The corresponding geometry specification and outline process plan files for the third sample of component which is carried out by the CADOPP system is shown in Table 7 and Table 8 respectively.

```

SCOPE COMMON
STOCK<-^MATERIAL(050,20)
THRD1<-^THREAD(18,08,0,0,0,032,0,0)
UCUT1<-^UNDERCUT(02,06,0,0,0,030,0,0)
CYLN1<-^CYLINDER(08,16,0,0,0,022,0,0)
UCUT2<-^UNDERCUT(04,14,0,0,0,018,0,0)
RCHM1<-^RIGHT_CHAMFER(02,16,18,0,0,0,016,0,0)
CYLN2<-^CYLINDER(14,18,0,0,0,002,0,0)
LCHM1<-^LEFT_CHAMFER(02,16,18,0,0,0,002,0,0)
RITP1<-^R_TAP_BORE(34,04,06,0,0,0,050,0,0)
CBOR1<-^CYLINDER_BORE(16,04,0,0,0,000,0,0)
COMPONENT<-STOCK-THRD1-UCUT1-CYLN1-UCUT2-RCHM1-CYLN2-LCHM1$
-RITP1-CBOR1
!
! MATERIAL
!
SCOPE MATERIAL
SOLID FAMILY MATERIAL(LENGTH,RADIUS):MATERIAL
MATERIAL<-CYL(LENGTH,RADIUS)AT(ROTY=90)
RADIUS=20
!
! CYLINDER
!
SCOPE CYLINDER
SOLID FAMILY CYLINDER(LENGTH,RADIUS1,X1,Y1,Z1,$
X2,Y2,Z2):CYLINDER
OBJ1<-CYL(LENGTH,RADIUS)AT(ROTY=90)
OBJ2<-CYL(LENGTH,RADIUS1)AT(ROTY=90)
CYLINDER<-MOVE(OBJ1-OBJ2)BY(X2,0,0)
RADIUS=20
!
! THREAD
!

```

Table 7 : The Geometric Specification File of Sample 3.

```

SCOPE THREAD
SOLID FAMILY THREAD (LENGTH, RADIUS1, X1, Y1, Z1, $
X2, Y2, Z2) :THREAD
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
THREAD<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=20
!
! UNDERCUT
!
SCOPE UNDERCUT
SOLID FAMILY UNDERCUT (LENGTH, RADIUS1, X1, Y1, Z1, $
X2, Y2, Z2) :UNDERCUT
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
UNDERCUT<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=20
!
! RIGHT_CHAMFER
!
SCOPE RIGHT_CHAMFER
SOLID FAMILY RIGHT_CHAMFER (LENGTH, RADIUS1, RADIUS2, $
X1, Y1, Z1, X2, Y2, Z2) :RIGHT_CHAMFER
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=90)
RIGHT_CHAMFER<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=20
!
! LEFT_CHAMFER
!
SCOPE LEFT_CHAMFER
SOLID FAMILY LEFT_CHAMFER (LENGTH, RADIUS1, RADIUS2, $
X1, Y1, Z1, X2, Y2, Z2) :LEFT_CHAMFER
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=270)
OBJ2<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=270)
LEFT_CHAMFER<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=20
!
! CYLINDER_BORE
!
SCOPE CYLINDER_BORE
SOLID FAMILY CYLINDER_BORE (LENGTH, RADIUS1, $
X1, Y1, Z1, X2, Y2, Z2) :CYLINDER_BORE
OBJ1<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
CYLINDER_BORE<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=20

```

Table 7 : Continued.

```

! R_TAP_BORE
!
SCOPE R_TAP_BORE
SOLID FAMILY R_TAP_BORE (LENGTH, RADIUS1, RADIUS2, $
X1, Y1, Z1, X2, Y2, Z2) : R_TAP_BORE
OBJ1 <- CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=270)
R_TAP_BORE <- MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=20

```

Table 7 : Continued.

Part material details : Brass Round Bar

Geometry : A round bar of dimensions L050 x R20 (mm)

Op.no	Oper.name	Machine	Tool-type	Lgth	Rad	MinRad	MajRad	Pic	Wid	Dep	Ang	Pattern
00	Stock			050	20							
01	Ex.Thread	Lathe	Ex.Thread	18	08			04				Acme
02	Undercut	Lathe	End Cut	02	06							
03	Cylinder	Lathe	Turning	08	16							
04	Undercut	Lathe	End Cut	04	14							
05	RChamfer	Lathe	Forming	02		16	18				45	
06	Cylinder	Lathe	Turning	14	18							
07	LChamfer	Lathe	Forming	02		16	18				45	
08	RTapBore	Lathe	Tap.Bore	34		04	06					
09	Cyl.Bore	Lathe	Boring	16	04							

Table 8 : An Outline Process Plan of Sample 3.

The 3-D output drawing of the Sample 3 is shown in Figure 7.6.

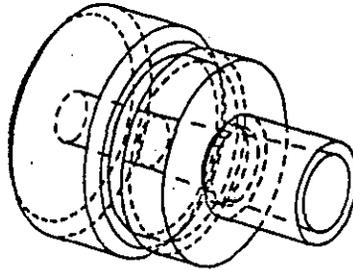


Figure 7.6 : The 3-D Output Drawing of Sample 3.

The system has been designed to allow the input related to the features to be modified if an error is made while keying the input data. Also, alternatively the current feature can be cancelled.

As mentioned in the previous chapter, after the geometric specification and outline process planning files have been produced, the system allows the user to modify the information manually. These modifications can be done in three ways that are:

- (i). delete the existing processes,
- (ii). insert new processes,
- (iii). change feature's information.

In the modification module, firstly the user needs to type-in [M] for modifying files. The system will request whether the modification is needed or not. After the user states that the modification is needed, the system will ask name of files that need to be modified. Then the user needs to select type of modifications whether to delete the existing processes, insert new processes or change the information of the features.

If the user needs to delete the operation, the system will ask which operation number that is needed to be deleted. After the operation number is entered, a statement of the operation number that has been deleted is displayed.

The same procedure of modification for selecting the insert new processes is followed. The user selects the insert new processes option and is asked to give the operation number of the previous line. Then the type of features is displayed for user to select. This is followed by enter the feature's attributes. After all the input has been entered, a list of parameters of that features is displayed.

Thirdly is the modification by changing the information of the feature. The same steps should be taken before option change the information is selected. This is followed by asking the user which operation number's and what type of feature need to be changed. The user needs to choose and key-in the feature parameters. After changes have been made, this will bring the user to the main menu selections of the CADOPP system.

7.6 Summary

Several rotational machined components were tested. The components tested comprising of several external and internal features with different dimensions. The reason for trying out CADOPP on the samples was to provide some evidence that the system and the concepts behind it actually work. The above demonstrations have shown that the CADOPP system can be used as a basic system to integrate the design and process planning where this system can concurrently produce the geometric specification and outline process plan.

The system was able to recommend the machine tools and the cutting tool for all the operations based on the type of the features.

Chapter 8

CHAPTER 8

CONCLUSIONS AND FUTURE RESEARCH

8.1 Introduction

The system which has been described in the previous chapter in order to integrate the design and process planning has been developed and tested. The system used an approach of designing with features where the features have been used as a means of linking the geometry and machining. The software of the system has been implemented in Fortran-77 programming language and in a subroutine manner. The input to the system is information about features' attributes. The output from the system are the output files of geometric specification, that is the model of a component, and an outline process plan. This chapter concludes the results from the research and the future research.

8.2 Conclusions

The research undertaken was to illustrate an approach that the designer can use to design certain rotational machined components in terms of manufacturing features and simultaneously capture this information which is suitable for outline process planning.

The capabilities of the system has been discussed in the previous chapter. It seems that it was capable of producing the model of a components and a list of manufacturing processes to manufacture the simple rotational components. The performance of the system in producing the model was fairly fast compared with existing CAD solid modeller.

This work has shown that the interactive dialogue used in the CADOPP system is convenient for the user. The interactive programming has been shown to reduce the skill required because the user is prompted by the dialogue mode. The sharing of the tasks by the computer and the user increases the overall performance and the job satisfaction of the user.

This system is relatively easy to use and its feature enhances its userfriendliness. The user only needs to key-in the input data of the feature and select only the required item on the screen.

The data for the recommended operation types, machine tools and cutting tools was found to be reasonable at the moment.

The system produces the geometric specification of the component where it can be applied for the finite element analysis. However, in producing an outline process planning file, its still depend on the skill of the user to determine the sequence of operations.

The present research work is only part of an overall effort to link the design function with process planning and also to the manufacturing function of machining the component.

The results from the system implementation have clearly shown that the integration between both design and process planning can be done by using features. In order to model a component, the system has provide the answers about features and their characteristics. This is an advantage of using the designing with features approach. The relationship between features and manufacturing processes in planning strategy has been used in creating the process planning sheet.

However, this approach has arise a constraints in determining a set of features with the manufacturing method and sequence of manufacturing processes in order to create the process planning. In order to produce the geometric specification of the rotational machined components and outline process planning, it should be considered that this is an integrated activity which can be done by one person.

Even this system is limited to the design and manufacturing problems, it has explained an overall concept of the approach that has been used in integrating the design and process planning.

So that, the major contributions to the research work are to illustrate the concept of using the destructive solid geometry technique of designing with features approach for rotational machined

components in order to simultaneously capture the designer intention with the list of manufacturing processes and the modification of the information.

8.3 Future Research

This research has demonstrated the use of manufacturing features in designing the geometric specification and outline process planning. However, this system is not a complete system for integrating the design and manufacturing function through process planning. A lot of effort needs to be done on CADOPP before it can be called a system which can be used fully in integrating the design and process planning.

The preliminary work had already been done to explore the concept of designing with features in linking both functions. These include developing the program which is suitable for creating the model in the solid modeller and producing a list of manufacturing processes in terms of outline process planning, modifying the information and implementing the system.

However, the application area was limited to certain types of features for rotational machined components in producing the model of a component in the solid modeller. Future research is recommended to cover all types of features, especially the non-rotational features such as keyways, unround holes, slots and so forth.

The CADOPP system is far from complete particularly in the process planning where the processes are sequence based on the features that have been chosen. In other words, the method of sequencing the processes is based on human expertise. This can be extended by applying an expert to the system in order to automatically sequence the operations or processes for manufacturing purposes based on the intelligent knowledge.

The present system does not have the capability to incorporate the machining parameters such as speed, feed, depth of cut and width of cut for each machining operation. This could be done by developing an algorithm which will determine the parameters needed. By doing

this, it will cut down the time spent by the user and also this will increase the productivity.

A designing with features approach is seen as the direction to integrate the design and manufacturing through process planning. The "front-end" system that has been developed can be improved by adding more related functions and features, and the following are suggested:-

(i). Developing a menu driven system including graphics and more explanations or guides with mouse support which will create a more user friendly environment,

(ii). Developing an algorithm which will calculate the approximately machining times of each machining operation,

(iii). Developing a program which will provide the tolerancing and surface finish information which are important for process planning.

At the end, it is hoped that the concept of the approach presented in this research gives a clear illustration to someone who is interested to contribute to the development of integrating the design and process planning using the designing with feature approach.

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Appendices

Appendix I - Scope Statement for Each Feature.

For stock of material :

```
SCOPE MATERIAL
SOLID FAMILY MATERIAL (LENGTH, RADIUS) : MATERIAL
MATERIAL<-CYL (LENGTH, RADIUS) AT (ROTY=90)
LENGTH=150
RADIUS=25
```

For cylinder feature :

```
SCOPE CYLINDER
SOLID FAMILY CYLINDER (LENGTH, RADIUS1, X1, Y1, Z1, $
X2, Y2, Z2) : CYLINDER
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
CYLINDER<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For face feature :

```
SCOPE FACE
SOLID FAMILY FACE (LENGTH, RADIUS, X1, Y1, Z1, X2, Y2, $
Z2) : FACE
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
FACE<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=25
```

For right-taper feature :

```
SCOPE RIGHT_TAPER
SOLID FAMILY RIGHT_TAPER (LENGTH, RADIUS1, $
RADIUS2, X1, Y1, Z1, X2, Y2, Z2) : RIGHT_TAPER
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=90)
RIGHT_TAPER<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For left-taper feature :

```
SCOPE LEFT_TAPER
SOLID FAMILY LEFT_TAPER (LENGTH, RADIUS1, $
RADIUS2, X1, Y1, Z1, X2, Y2, Z2) : LEFT_TAPER
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=270)
OBJ2<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=270)
LEFT_TAPER<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For knurl feature :

```
SCOPE KNURL
SOLID FAMILY KNURL (LENGTH, RADIUS1, X1, Y1, Z1, $
X2, Y2, Z2) : KNURL
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
KNURL<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

(continued)

For thread feature :

```
SCOPE THREAD
SOLID FAMILY THREAD (LENGTH, RADIUS1, X1, Y1, Z1, $
X2, Y2, Z2) :THREAD
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
THREAD<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For pinion feature :

```
SCOPE PINION
SOLID FAMILY PINION (LENGTH, RADIUS1, RADIUS2, X1, $
Y1, Z1, X2, Y2, Z2) :PINION
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=90)
PINION<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For spline feature :

```
SCOPE SPLINE
SOLID FAMILY SPLINE (LENGTH, RADIUS1, RADIUS2, X1, $
Y1, Z1, X2, Y2, Z2) :SPLINE
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=90)
SPLINE<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For cross-hole feature :

```
SCOPE CROSS_HOLE
SOLID FAMILY CROSS_HOLE (LENGTH, RADIUS1, $
X1, Y1, Z1, X2, Y2, Z2) :CROSS_HOLE
OBJ1<-CYL (LENGTH, RADIUS1) AT (ROTX=90)
CROSS_HOLE<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=25
```

For unround-hole feature :

```
SCOPE UNROUND_HOLE
SOLID FAMILY UNROUND_HOLE (LENGTH, RADIUS1, X1, Y1, $
Z1, X2, Y2, Z2) :UNROUND_HOLE
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
UNROUND_HOLE<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For across-flat feature :

```
SCOPE ACROSS_FLAT
SOLID FAMILY ACROSS_FLAT (X, Y, Z, XC, YC, ZC, RX, RY, RZ, $
IROT) :ACROSS_FLAT
OBJ1<-BLOCK (ABS (X) , ABS (Y) , ABS (Z) )
ACROSS_FLAT<-MOVE (OBJ1) BY (XC, YC, ZC)
```

For keyway feature :

(continued)

```
SCOPE KEYWAY
SOLID FAMILY KEYWAY (X, Y, Z, XC, YC, ZC, RX, RY, RZ, $
IROT) :KEYWAY
OBJ1<-BLOCK (ABS (X) , ABS (Y) , ABS (Z) )
KEYWAY<-MOVE (OBJ1) BY (XC, YC, ZC)
RADIUS=25
```

For undercut feature :

```
SCOPE UNDERCUT
SOLID FAMILY UNDERCUT (LENGTH, RADIUS1, X1, Y1, Z1, $
X2, Y2, Z2) :UNDERCUT
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
UNDERCUT<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For formcut feature :

```
SCOPE FORMCUT
SOLID FAMILY FORMCUT (R1, R2, XC, YC, ZC, RX, RY, RZ, IROT) $
:FORMCUT
OBJ1<-TORUS (RADIUS2, RADIUS1) AT (ROTY=90)
FORMCUT<-MOVE (OBJ1) BY (XC, YC, ZC)
RADIUS=25
```

For round feature :

```
SCOPE ROUND
SOLID FAMILY ROUND (R1, R2, XC, YC, ZC, RX, RY, RZ, IROT) $
:ROUND
OBJ1<-TORUS (RADIUS2, RADIUS1) AT (ROTY=90)
ROUND<-MOVE (OBJ1) BY (XC, YC, ZC)
RADIUS=25
```

For right-chamfer feature :

```
SCOPE RIGHT_CHAMFER
SOLID FAMILY RIGHT_CHAMFER (LENGTH, RADIUS1, RADIUS2, $
X1, Y1, Z1, X2, Y2, Z2) :RIGHT_CHAMFER
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=90)
RIGHT_CHAMFER<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For left-chamfer feature :

```
SCOPE LEFT_CHAMFER
SOLID FAMILY LEFT_CHAMFER (LENGTH, RADIUS1, RADIUS2, $
X1, Y1, Z1, X2, Y2, Z2) :LEFT_CHAMFER
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=270)
OBJ2<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=270)
LEFT_CHAMFER<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For cylinder-bore feature :

```
SCOPE CYLINDER_BORE
SOLID FAMILY CYLINDER_BORE (LENGTH, RADIUS1, $
```

(continued)

```
X1, Y1, Z1, X2, Y2, Z2) : CYLINDER_BORE
OBJ1<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
CYLINDER_BORE<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=25
```

For right-taper-bore feature :

```
SCOPE R_TAP_BORE
SOLID FAMILY R_TAP_BORE (LENGTH, RADIUS1, RADIUS2, $
X1, Y1, Z1, X2, Y2, Z2) : R_TAP_BORE
OBJ1<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=90)
R_TAP_BORE<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=25
```

For left-taper-bore feature :

```
SCOPE L_TAP_BORE
SOLID FAMILY L_TAP_BORE (LENGTH, RADIUS1, RADIUS2, $
X1, Y1, Z1, X2, Y2, Z2) : L_TAP_BORE
OBJ1<-CONE (LENGTH, RADIUS1, RADIUS2) AT (ROTY=270)
L_TAP_BORE<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=25
```

For internal-thread feature :

```
SCOPE INTERNAL_THREAD
SOLID FAMILY INTERNAL_THREAD (LENGTH, RADIUS1, $
X1, Y1, Z1, X2, Y2, Z2) : INTERNAL_THREAD
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
INTERNAL_THREAD<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For internal-keyway feature :

```
SCOPE INTERNAL_KEYWAY
SOLID FAMILY INTERNAL_KEYWAY (X, Y, Z, XC, YC, ZC, $
RX, RY, RZ, IROT) : INTERNAL_KEYWAY
OBJ1<-BLOCK (ABS (X), ABS (Y), ABS (Z))
INTERNAL_KEYWAY<-MOVE (OBJ1) BY (XC, YC, ZC)
RADIUS=25
```

For recess feature :

```
SCOPE RECESS
SOLID FAMILY RECESS (LENGTH, RADIUS1, X1, Y1, Z1, $
X2, Y2, Z2) : RECESS
OBJ1<-CYL (LENGTH, RADIUS) AT (ROTY=90)
OBJ2<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
RECESS<-MOVE (OBJ1-OBJ2) BY (X2, 0, 0)
RADIUS=25
```

For centre-drill feature :

```
SCOPE CENTRE_DRILL
SOLID FAMILY CENTRE_DRILL (LENGTH, RADIUS1, $
X1, Y1, Z1, X2, Y2, Z2) : CENTRE_DRILL
OBJ1<-CYL (LENGTH, RADIUS1) AT (ROTY=90)
CENTRE_DRILL<-MOVE (OBJ1) BY (X2, 0, 0)
RADIUS=25
```

(continued)

For right-internal-chamfer feature :

```
SCOPE R_INT_CHAMFER
SOLID FAMILY R_INT_CHAMFER(LENGTH,RADIUS1,$
RADIUS2,X1,Y1,Z1,X2,Y2,Z2):R_INT_CHAMFER
OBJ1<-CONE(LENGTH,RADIUS1,RADIUS2)AT(ROTY=90)
R_INT_CHAMFER<-MOVE(OBJ1)BY(X2,0,0)
RADIUS=25
```

For left-internal-chamfer feature :

```
SCOPE L_INT_CHAMFER
SOLID FAMILY L_INT_CHAMFER(LENGTH,RADIUS1,$
RADIUS2,X1,Y1,Z1,X2,Y2,Z2):L_INT_CHAMFER
OBJ1<-CONE(LENGTH,RADIUS1,RADIUS2)AT(ROTY=270)
L_INT_CHAMFER<-MOVE(OBJ1)BY(X2,0,0)
RADIUS=25
```

Appendix II - SCOPE COMMON and other Scopes.

```
SCOPE COMMON
STOCK<-^MATERIAL(150,25)
LCHM1<-^LEFT_CHAMFER(03,21,24,0,0,0,003,0,0)
CYLN1<-^CYLINDER(37,24,0,0,0,003,0,0)
UCUT1<-^UNDERCUT(10,13,0,0,0,040,0,0)
CYLN2<-^CYLINDER(20,18,0,0,0,050,0,0)
RTAP1<-^RIGHT_TAPER(30,13,18,0,0,0,070,0,0)
UCUT2<-^UNDERCUT(05,10,0,0,0,100,0,0)
CYLN3<-^CYLINDER(25,13,0,0,0,105,0,0)
CYLN4<-^CYLINDER(18,08,0,0,0,130,0,0)
RCHM1<-^RIGHT_CHAMFER(02,06,08,0,0,0,148,0,0)
COMPONENT<-STOCK-LCHM1-CYLN1-UCUT1-CYLN2-RTAP1-UCUT2-CYLN3$
-CYLN4-RCHM1
!
! MATERIAL
!
SCOPE MATERIAL
SOLID FAMILY MATERIAL(LENGTH,RADIUS):MATERIAL
MATERIAL<-CYL(LENGTH,RADIUS) AT (ROTY=90)
RADIUS=25
!
! CYLINDER
!
SCOPE CYLINDER
SOLID FAMILY CYLINDER(LENGTH,RADIUS1,X1,Y1,Z1,$
X2,Y2,Z2):CYLINDER
OBJ1<-CYL(LENGTH,RADIUS) AT (ROTY=90)
OBJ2<-CYL(LENGTH,RADIUS1) AT (ROTY=90)
CYLINDER<-MOVE(OBJ1-OBJ2) BY (X2,0,0)
RADIUS=25
!
! RIGHT_TAPER
!
SCOPE RIGHT_TAPER
SOLID FAMILY RIGHT_TAPER(LENGTH,RADIUS1,$
RADIUS2,X1,Y1,Z1,X2,Y2,Z2):RIGHT_TAPER
OBJ1<-CYL(LENGTH,RADIUS) AT (ROTY=90)
OBJ2<-CONE(LENGTH,RADIUS1,RADIUS2) AT (ROTY=90)
RIGHT_TAPER<-MOVE(OBJ1-OBJ2) BY (X2,0,0)
RADIUS=25
!
! UNDERCUT
!
SCOPE UNDERCUT
SOLID FAMILY UNDERCUT(LENGTH,RADIUS1,X1,Y1,Z1,$
X2,Y2,Z2):UNDERCUT
OBJ1<-CYL(LENGTH,RADIUS) AT (ROTY=90)
OBJ2<-CYL(LENGTH,RADIUS1) AT (ROTY=90)
UNDERCUT<-MOVE(OBJ1-OBJ2) BY (X2,0,0)
RADIUS=25
```

(continued)

```
!  
! RIGHT_CHAMFER  
!  
SCOPE RIGHT_CHAMFER  
SOLID FAMILY RIGHT_CHAMFER(LENGTH,RADIUS1,RADIUS2,$  
X1,Y1,Z1,X2,Y2,Z2):RIGHT_CHAMFER  
OBJ1<-CYL(LENGTH,RADIUS)AT(ROTY=90)  
OBJ2<-CONE(LENGTH,RADIUS1,RADIUS2)AT(ROTY=90)  
RIGHT_CHAMFER<-MOVE(OBJ1-OBJ2)BY(X2,0,0)  
RADIUS=25  
!  
! LEFT_CHAMFER  
!  
SCOPE LEFT_CHAMFER  
SOLID FAMILY LEFT_CHAMFER(LENGTH,RADIUS1,RADIUS2,$  
X1,Y1,Z1,X2,Y2,Z2):LEFT_CHAMFER  
OBJ1<-CYL(LENGTH,RADIUS)AT(ROTY=270)  
OBJ2<-CONE(LENGTH,RADIUS1,RADIUS2)AT(ROTY=270)  
LEFT_CHAMFER<-MOVE(OBJ1-OBJ2)BY(X2,0,0)  
RADIUS=25
```

Appendix III - CADOPP User Information.

```

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%           =====           %%
%           THIS INFORMATION IS FOR THE USER TO           %%
%           KNOW WHAT IS THE SYSTEM ALL ABOUT.           %%
%           =====           %%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

```

1. The CADOPP system is designed to produce the outline process planning and geometric specification system for the rotational components.
2. All the input related to the features for process planning is entered into the system by a dialogue between the user and the computer.
3. The user initially describes the component blank and then interacts with the software to define the machined features.
4. Machined features are classified into 2 types:
 - a). External feature.
 - b). Internal feature.and subclassified into other 3 types:
 - i). Axial.
 - ii). Surface.
 - iii). Radial.
5. The location of the features is referred to the one end point at the origin, along the positive X-axis.
6. Enter the appropriate data when the system asks.

```

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%           =====           %%
%           THE INFORMATION OF TURNING OPERATIONS           %%
%           =====           %%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

```

1. Turning operations can be divided into two classes:
 - a. The operations for machining between centers.
 - b. The operations for machining in a chuck.
2. Finish turning should be done after rough turning.
3. Depth of rough cut will be from 0.750 to 25.4 mm.
Depth of finish cut will be from 0.125 to 0.500 mm.

(continued)

4. Thread cutting and knurling operations should be done after rough turning.
5. Reaming, boring and tapping operations should be done after drilling.
6. Drilling should be done after centre drilling.
7. Reaming should be done for hole diameter 3 to 40 mm.
8. Boring should be done for hole larger than 40 mm.
9. Parting-off is the last operation.
10. If $L/D \geq 3$, workpiece should be supported by a steady rest or tailstock center.

Appendix IV - An Interactive Delete of Existing Processes.

```

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%
%          *****
%
%          THE MAIN MENU SELECTIONS
%
%          OF THE CADOPP SYSTEM
%
%          *****
%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%
%
%          [C]REATE NEW FILES
%
%          [M]ODIFY FILES
%
%          [R]EVIEW FILES
%
%          [D]ELETE FILES
%
%          [E]XIT THE SYSTEM
%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

```

?: Select desired option :
or enter [E] to exit the system.

m

?: Do you want to make any MODIFICATIONS ? (y/n)

y

?: Enter the geometry specification filename to MODIFY :
box1

?: Enter the NEW geometry specification filename :
box1.mod

?: Enter the outline process plan filename to MODIFY :
oppl

?: Enter the NEW outline process plan filename :
oppl.mod

MODIFY PLAN CAN BE DONE IN THREE (3) WAYS.

=====

- [1]. Delete the existing processes.
- [2]. Insert new processes.
- [3]. Change the information.

?: Select the type of modifications : [1] - [3].

1

(continued)

```
*****  
% ===== %  
%   WELCOME TO THE DELETE INFORMATION MODULE   %  
% ===== %  
*****
```

?: Which OPERATION NUMBER do you want to delete ?

5

OPERATION NO. 5 HAS BEEN DELETED.

?: Do you want to delete anymore operations ? (y/n)

n.

?: Do you want to quit the module ? (y/n)

y

Appendix V - An Interactive Insert/Add of New Processes.

```

#####
#####
%                *****                %
%                THE MAIN MENU SELECTIONS                %
%                OF THE CADOPP SYSTEM                    %
%                *****                %
#####
#####
#####
%
%
%                [C]REATE NEW FILES                    %
%
%                [M]ODIFY FILES                        %
%
%                [R]EVIEW FILES                        %
%
%                [D]ELETE FILES                        %
%
%                [E]XIT THE SYSTEM                    %
%
%
#####

```

?: Select desired option :
or enter [E] to exit the system.

m

?: Do you want to make any MODIFICATIONS ? (y/n)

y

?: Enter the geometry specification filename to MODIFY :
box2

?: Enter the NEW geometry specification filename :
box2.mod

?: Enter the outline process plan filename to MODIFY :
opp2

?: Enter the NEW outline process plan filename :
opp2.mod

MODIFY PLAN CAN BE DONE IN THREE (3) WAYS.

=====

- [1]. Delete the existing processes.
- [2]. Insert new processes.
- [3]. Change the information.

?: Select the type of modifications : [1] - [3].

2

(continued)

```
#####  
% ===== %  
% WELCOME TO THE INSERT INFORMATION MODULE %  
% ===== %  
#####
```

? : Give the OPERATION NUMBER of the previous line :
5

TYPE OF FEATURES FOR INSERT MODULE
=====

- | | |
|---------------------|-----------------------|
| 1. Cylinder. | 12. Undercut. |
| 2. Face. | 13. Formcut. |
| 3. Taper. | 14. Round. |
| 4. Knurl. | 15. Chamfer. |
| 5. External-Thread. | 16. Cylinder-Bore. |
| 6. Pinion. | 17. Taper-Bore. |
| 7. Spline. | 18. Internal-Thread. |
| 8. Cross-Hole. | 19. Internal-Keyway. |
| 9. Unround-Hole. | 20. Recess. |
| 10. Across-Flat. | 21. Centre-Drill. |
| 11. Keyway. | 22. Internal-Chamfer. |

? : Select desired feature to insert : [1] - [22]
4

1: Enter the LENGTH of knurl feature :
(2-digits)
10

2: Enter the RADIUS of knurl feature :
(2-digits)
18

PATTERNS OF KNURL AVAILABLE
=====

- [1]. Straight
- [2]. Diamond

? : Enter only selections listed : [1] or [2]
1

Straight pattern can be divided into 3 types.

- [1]. Fine straight
- [2]. Medium straight
- [3]. Course straight

3: Enter only selections listed : [1] - [3]
2

(continued)

4: Enter the POSITION of knurl feature :
(3-digits)

100

The parameters for Knurl feature are as follows :

1. Operation name = Knurl
2. Machine tool = Lathe
3. Tool type = Knurling
4. Length = 10
5. Radius = 18
6. Minor-Radius =
7. Major-Radius =
8. Pitch =
9. Width =
10. Depth =
11. Angle =
12. Pattern = MediumS
13. Position = 100

?: Any data to correct ? (y/n)

n

?: Do you want to insert anymore processes ? (y/n)

n

?: Do you want to quit the module ? (y/n)

y

Appendix VI - An Interactive Change the Feature Information.

```

*****
THE MAIN MENU SELECTIONS
OF THE CADOPP SYSTEM
*****

[C]REATE NEW FILES
[M]ODIFY FILES
[R]EVIEW FILES
[D]ELETE FILES
[E]XIT THE SYSTEM
*****
```

?: Select desired option :
or enter [E] to exit the system.

m

?: Do you want to make any MODIFICATIONS ? (y/n)

y

?: Enter the geometry specification filename to MODIFY :
box3

?: Enter the NEW geometry specification filename :
box3.mod

?: Enter the outline process plan filename to MODIFY :
opp3

?: Enter the NEW outline process plan filename :
opp3.mod

MODIFY PLAN CAN BE DONE IN THREE (3) WAYS.

- =====
- [1]. Delete the existing processes.
 - [2]. Insert new processes.
 - [3]. Change the information.

?: Select the type of modifications : [1] - [3].

3

(continued)

```
*****  
% ===== %  
% WELCOME TO THE CHANGE INFORMATION MODULE %  
% ===== %  
*****
```

? : Which OPERATION NUMBER's information to change ?
5

? : AND what type of feature it is ?

- | | |
|---------------------|-----------------------|
| 1. Cylinder. | 12. Undercut. |
| 2. Face. | 13. Formcut. |
| 3. Taper. | 14. Round. |
| 4. Knurl. | 15. Chamfer. |
| 5. External-Thread. | 16. Cylinder-Bore. |
| 6. Pinion. | 17. ITaper-Bore. |
| 7. Spline. | 18. Internal-Thread. |
| 8. Cross-Hole. | 19. Internal-Keyway. |
| 9. Unround-Hole. | 20. Recess. |
| 10. Across-Flat. | 21. Centre-Drill. |
| 11. Keyway. | 22. Internal-Chamfer. |

? : Select desired option : [1] - [22]
12

? : Which feature parameters do you want to change ?

- [01]. Length.
- [02]. Radius.
- [03]. Minor-radius.
- [04]. Major-radius.
- [05]. Pitch.
- [06]. Width.
- [07]. Depth.
- [08]. Angle.
- [09]. Pattern.
- [10]. Position.

? : Select desired option : [1] - [10]
or enter [0] to end specification.
01

? : Enter the new length of the feature :
(2-digits)
05

? : Which feature parameters do you want to change ?

(continued)

- [01]. Length.
- [02]. Radius.
- [03]. Minor-radius.
- [04]. Major-radius.
- [05]. Pitch.
- [06]. Width.
- [07]. Depth.
- [08]. Angle.
- [09]. Pattern.
- [10]. Position.

?: Select desired option : [1] - [10] .
or enter [0] to end specification.

0

?: Do you want to change any OPERATION NUMBER's information ? (y/n)

n

?: Do you want to quit the module ? (y/n)

y

Appendix VII - An Interactive Session of Sample 1.

\$ CADOPP

```

#####
#####
%                !!!!!!!!!!!!!!!!!!!!!!!                %%
%                ! WELCOME TO THE SYSTEM !                %%
%                !!!!!!!!!!!!!!!!!!!!!!!                %%
#####
#####
%
%                =====
%                DEPARTMENT OF MANUFACTURING ENGINEERING    %%
%                =====
%                LOUGHBOROUGH UNIVERSITY OF TECHNOLOGY      %%
%                =====
%
%                CCCCC      A      DDDDD      OOOO      PPPPP      PPPPP    %%
%                CCCCC      A A     DDDDD D     OOOO O PPPPP P PPPPP P    %%
%                CC        A A A   DD   DD   OO   OO PP   PP PP   PP    %%
%                CC        A   A A DD   DD   OO   OO PP   PP PP   PP    %%
%                CC        AA    AA DD   DD   OO   OO PP   PP PP   PP    %%
%                CC        AAAAAAA DD   DD   OO   OO PPPPP PPPPP    %%
%                CC        AAAAAAA DD   DD   OO   OO PPPP  PPPP    %%
%                CC        AA    AA DD   DD   OO   OO PP   PP    %%
%                CC        AA    AA DD   DD   OO   OO PP   PP    %%
%                C CCCCC AA    AA DDDDD   O OOOO PP   PP    %%
%                CCCCC A     A  DDDDD     OOOO  P     P    %%
%
%                =====
%                COMPUTER-AIDED DESIGN AND                    %%
%                OUTLINE PROCESS PLANNING SYSTEM              %%
%                =====
%
#####

```

?: Enter password:
cadopp

?: Do you need the user information ? (y/n)
n

?: Do you wish to start the system now ? (y/n)
y

```

#####
#####
%                *****                                %%
%                THE MAIN MENU SELECTIONS                %%
%                OF THE CADOPP SYSTEM                    %%
%                *****                                %%
#####
#####
%
%                [C]REATE NEW FILES                      %%
%
%                [M]ODIFY FILES                          %%
%
%                [R]EVIEW FILES                          %%
%
%                [D]ELETE FILES                          %%

```


(continued)

3

?: Strike a [RETURN] key for feature menu selection :

```

#####
#####
% ***** %
% THE FEATURE MENU SELECTIONS OF THE SYSTEM %%
% FOR DESIGN AND PROCESS PLANNING %%
% ***** %
#####

```

Features are classified into two (2) types :

- [01]. EXTERNAL FEATURES.
- [02]. INTERNAL FEATURES.

?: SELECT feature option : [01] or [02]

1

```

#####
#####
% ===== %%
% EXTERNAL FEATURES TYPE %%
% ===== %%
#####

```

External features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15], or ENTER [16] for internal features, or ENTER [00] to end the operation.

15

Which type of chamfer for chamfer feature ?

- [1]. Right-chamfer
- [2]. Left-chamfer

?: Enter only selections listed : [1] - [2]

(continued)

1

4

```

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% ----- %%
% Data input for RIGHT-CHAMFER feature that %%
% is required for design and process planning. %%
% ----- %%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

```

1: Enter the LENGTH of right-chamfer :
(2-digits)

02

2: Enter the MINOR-RADIUS of right-chamfer :
(2-digits)

06

3: Enter the MAJOR-RADIUS of right-chamfer :
(2-digits)

08

4: Enter the ANGLE of right-chamfer :
(2-digits)

45

5: Enter the POSITION of right-chamfer :
(3-digits)

148

The parameters for RIGHT-CHAMFER feature are as follows :

- 1. Length = 02
- 2. Minor-Radius = 06
- 3. Major-Radius = 08
- 4. Angle = 45
- 5. Position = 148

?: Any data to correct ? (y/n/cancel)

n

```

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% ===== %%
% EXTERNAL FEATURES TYPE %%
% ===== %%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

```

External features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

(continued)

5

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.
- => [16]. INTERNAL FEATURES.

? : SELECT external feature type : [01] - [15],
 or ENTER [16] for internal features,
 or ENTER [00] to end the operation.

01

```

*****
*****
% ----- %%
% Data input for CYLINDER feature that is %%
% required for design and process planning. %%
% ----- %%
*****

```

1: Enter the LENGTH of cylinder :
 (2-digits)

18

2: Enter the RADIUS of cylinder :
 (2-digits)

08

3: Enter the POSITION of cylinder :
 (3-digits)

130

The parameters for CYLINDER feature are as follows :

- 1. Length = 18
- 2. Radius = 08
- 3. Position = 130

? : Any data to correct ? (y/n/cancel)

n

```

*****
*****
% ===== %%
% EXTERNAL FEATURES TYPE %%
% ===== %%
*****

```

External features are subclassified
 into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.
- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.

(continued)

6

- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.
- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.
- => [16]. INTERNAL FEATURES.

? : SELECT external feature type : [01] - [15],
 or ENTER [16] for internal features,
 or ENTER [00] to end the operation.

01

```

*****
*****
% ----- %%
% Data input for CYLINDER feature that is %%
% required for design and process planning. %%
% ----- %%
*****

```

1: Enter the LENGTH of cylinder :
 (2-digits)

25

2: Enter the RADIUS of cylinder :
 (2-digits)

13

3: Enter the POSITION of cylinder :
 (3-digits)

105

The parameters for CYLINDER feature are as follows :

- 1. Length = 25
- 2. Radius = 13
- 3. Position = 105

? : Any data to correct ? (y/n/cancel)

n

```

*****
*****
% ===== %%
% EXTERNAL FEATURES TYPE %%
% ===== %%
*****

```

External features are subclassified
 into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

(continued)

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

? : SELECT external feature type : [01] - [15],
 or ENTER [16] for internal features,
 or ENTER [00] to end the operation.

12

```

#####
#####
% ----- %%
%   Data input for UNDERCUT feature that is   %%
%   required for design and process planning. %%
% ----- %%
#####

```

1: Enter the LENGTH of undercut :
 (2-digits)

05

2: Enter the RADIUS of undercut :
 (2-digits)

10

3: Enter the POSITION of undercut :
 (3-digits)

100

The parameters for UNDERCUT feature are as follows :

- 1. Length = 05
- 2. Radius = 10
- 3. Position = 100

? : Any data to correct ? (y/n/cancel)

n

```

#####
#####
% ===== %%
%   EXTERNAL FEATURES TYPE   %%
% ===== %%
#####

```

External features are subclassified
 into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.

(continued)

=> [07]. SPLINE.

8

B. SURFACE. => [08]. CROSS-HOLE.
=> [09]. UNROUND-HOLE.
=> [10]. ACROSS-FLAT.
=> [11]. KEYWAY.

C. RADIAL. => [12]. UNDERCUT.
=> [13]. FORMCUT.
=> [14]. ROUND.
=> [15]. CHAMFER.
=> [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15],
or ENTER [16] for internal features,
or ENTER [00] to end the operation.

03

Which type of taper for taper feature ?

[1]. Right-taper
[2]. Left-taper

?: Enter only selections listed : [1] - [2]

1

```

*****
*****
% ----- %%
% Data input for RIGHT-TAPER feature that is %%
% required for design and process planning. %%
% ----- %%
*****

```

1: Enter the LENGTH of right-taper :
(2-digits)

30

2: Enter the MINOR-RADIUS of right-taper :
(2-digits)

13

3. Enter the MAJOR-RADIUS of right-taper :
(2-digits)

18

4: Enter the POSITION of right-taper :
(3-digits)

070

The parameters for RIGHT-TAPER feature are as follows :

1. Length = 30
2. Minor-Radius = 13
3. Major-Radius = 18
4. Position = 070

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****

```

(continued)

9

```

% ===== %%
% EXTERNAL FEATURES TYPE %%
% ===== %%
%*****%

```

External features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES..

?: SELECT external feature type : [01] - [15],
or ENTER [16] for internal features,
or ENTER [00] to end the operation.

01

```

%*****%
%*****%
% ----- %%
% Data input for CYLINDER feature that is %%
% required for design and process planning. %%
% ----- %%
%*****%

```

1: Enter the LENGTH of cylinder :
(2-digits)

20

2: Enter the RADIUS of cylinder :
(2-digits)

18

3: Enter the POSITION of cylinder :
(3-digits)

050

The parameters for CYLINDER feature are as follows :

- 1. Length = 20
- 2. Radius = 18
- 3. Position = 050

?: Any data to correct ? (y/n/cancel)

n

(continued)

10

```

*****
*****
%      =====      %%
%      EXTERNAL FEATURES TYPE      %%
%      =====      %%
*****

```

External features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. _RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15],
 or ENTER [16] for internal features,
 or ENTER [00] to end the operation.

12

```

*****
*****
%      -----      %%
%      Data input for UNDERCUT feature that is %%
%      required for design and process planning. %%
%      -----      %%
*****

```

1: Enter the LENGTH of undercut :
 (2-digits)

10

2: Enter the RADIUS of undercut :
 (2-digits)

13

3: Enter the POSITION of undercut :
 (3-digits)

040

The parameters for UNDERCUT feature are as follows :

- 1. Length = 10
- 2. Radius = 13
- 3. Position = 040

?: Any data to correct ? (y/n/cancel)

(continued)

n

11

```

*****
*****
%      =====      %%
%      EXTERNAL FEATURES TYPE      %%
%      =====      %%
*****

```

External features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15],
 or ENTER [16] for internal features,
 or ENTER [00] to end the operation.

01

```

*****
*****
%      -----      %%
%      Data input for CYLINDER feature that is      %%
%      required for design and process planning.      %%
%      -----      %%
*****

```

1: Enter the LENGTH of cylinder :
 (2-digits)

37

2: Enter the RADIUS of cylinder :
 (2-digits)

24

3: Enter the POSITION of cylinder :
 (3-digits)

003

The parameters for CYLINDER feature are as follows :

- 1. Length = 37
- 2. Radius = 24
- 3. Position = 003

(continued)

12

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****
% ----- %%
%   EXTERNAL FEATURES TYPE   %%
%   ----- %%
*****

```

External features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15], or ENTER [16] for internal features, or ENTER [00] to end the operation.

15

Which type of chamfer for chamfer feature ?

- [1]. Right-chamfer
- [2]. Left-chamfer

?: Enter only selections listed : [1] - [2]

2

```

*****
*****
% ----- %%
%   Data input for LEFT-CHAMFER feature that is %%
%   required for design and process planning. %%
%   ----- %%
*****

```

1: Enter the LENGTH of left-chamfer : (2-digits)

03

2: Enter the MINOR-RADIUS of left-chamfer : (2-digits)

21

(continued)

13

3: Enter the MAJOR-RADIUS of left-chamfer :
(2-digits)

24

4: Enter the ANGLE of left-chamfer :
(2-digits)

45

5: Enter the POSITION of left-chamfer :
(3-digits)

003

The parameters for LEFT-CHAMFER feature are as follows :

- 1. Length = 03
- 2. Minor-Radius = 21
- 3. Major-Radius = 24
- 4. Angle = 45
- 5. Position = 003

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****
%      =====      %%
%      EXTERNAL FEATURES TYPE      %%
%      =====      %%
*****

```

External features are subclassified
into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15],
or ENTER [16] for internal features,
or ENTER [00] to end the operation.

00

?: Do you want to SAVE the files ? (y/n)

y

(continued)

?: Strike a [RETURN] key for feature menu selection :

```

*****
*****
% ***** %
% THE FEATURE MENU SELECTIONS OF THE SYSTEM %%
% FOR DESIGN AND PROCESS PLANNING %%
% ***** %
*****

```

Features are classified into two (2) types :

- [01]. EXTERNAL FEATURES.
- [02]. INTERNAL FEATURES.

?: SELECT feature option : [01] or [02]
02

```

*****
*****
% ===== %%
% INTERNAL FEATURES TYPE %%
% ===== %%
*****

```

Internal features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER-BORE.
- => [02]. TAPER-BORE.
- => [03]. INTERNAL-THREAD.

- B. SURFACE. => [04]. INTERNAL-KEYWAY.
- => [05]. RECESS.

- C. RADIAL. => [06]. CENTRE-DRILL.
- => [07]. INTERNAL-CHAMFER.

- => [08]. EXTERNAL FEATURES.

?: SELECT internal feature type : [01] - [07],
or ENTER [08] for external features,
or ENTER [00] to end the operation.
07

Which type of chamfer for internal-chamfer feature ?

- [1]. Right-internal-chamfer
- [2]. Left-internal-chamfer

?: Enter only selections listed : [1] - [2]
1

```

*****
*****
% ----- %%
% Data input for RIGHT-INTERNAL-CHAMFER feature %%
% that is required for design and process planning. %%
% ----- %%

```

(continued)

1: Enter the LENGTH of right-internal-chamfer :
(2-digits)

03

2: Enter the MINOR-RADIUS of right-internal-chamfer :
(2-digits)

19

3: Enter the MAJOR-RADIUS of right-internal-chamfer :
(2-digits)

22

4: Enter the ANGLE of right-internal-chamfer :
(2-digits)

45

5: Enter the POSITION of right-internal-chamfer :
(3-digits)

074

The parameters for RIGHT-INTERNAL-CHAMFER feature are as follows :

- 1. Length = 03
- 2. Minor-Radius = 19
- 3. Major-Radius = 22
- 4. Angle = 45
- 5. Position = 074

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****
%      =====      %%
%      INTERNAL FEATURES TYPE      %%
%      =====      %%
*****

```

Internal features are subclassified
into three (3) types :

- A. AXIAL. => [01]. CYLINDER-BORE.
- => [02]. TAPER-BORE.
- => [03]. INTERNAL-THREAD.

- B. SURFACE. => [04]. INTERNAL-KEYWAY.
- => [05]. RECESS.

- C. RADIAL. => [06]. CENTRE-DRILL.
- => [07]. INTERNAL-CHAMFER.

- => [08]. EXTERNAL FEATURES.

?: SELECT internal feature type : [01] - [07],
or ENTER [08] for external features,
or ENTER [00] to end the operation.

01

(continued)

```

% ----- %%
% Data input for CYLINDER-BORE feature that %%
% is required for design and process planning. %%
% ----- %%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

```

1: Enter the LENGTH of cylinder-bore :
(2-digits)

20

2: Enter the RADIUS of cylinder-bore :
(2-digits)

19

3: Enter the POSITION of cylinder-bore :
(3-digits)

051

The parameters for CYLINDER-BORE feature are as follows :

```

1. Length      = 20
2. Radius      = 19
3. Position    = 051

```

?: Any data to correct ? (y/n/cancel)

n

```

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% ===== %%
% INTERNAL FEATURES TYPE %%
% ===== %%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

```

Internal features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER-BORE.
- => [02]. TAPER-BORE.
- => [03]. INTERNAL-THREAD.

- B. SURFACE. => [04]. INTERNAL-KEYWAY.
- => [05]. RECESS.

- C. RADIAL. => [06]. CENTRE-DRILL.
- => [07]. INTERNAL-CHAMFER.

- => [08]. EXTERNAL FEATURES.

?: SELECT internal feature type : [01] - [07],
or ENTER [08] for external features,
or ENTER [00] to end the operation.

02

Which type of taper for taper-bore feature ?

- [1]. Right-taper-bore
- [2]. Left-taper-bore

?: Enter only selections listed : [1] - [2]

1

(continued)

```

*****
*****
% ----- %%
% Data input for RIGHT-TAPER-BORE feature that %%
% is required for design and process planning. %%
% ----- %%
*****

```

1: Enter the LENGTH of right-taper-bore :
(2-digits)

30

2: Enter the MINOR-RADIUS of right-taper-bore :
(2-digits)

13

3: Enter the MAJOR-RADIUS of right-taper-bore :
(2-digits)

19

4: Enter the POSITION of right-taper-bore :
(3-digits)

051

The parameters for RIGHT-TAPER-BORE feature are as follows :

- 1. Length = 30
- 2. Minor-Radius = 13
- 3. Major-Radius = 19
- 4. Position = 051

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****
% ===== %%
% INTERNAL FEATURES TYPE %%
% ===== %%
*****

```

Internal features are subclassified
into three (3) types :

- A. AXIAL. => [01]. CYLINDER-BORE.
- => [02]. TAPER-BORE.
- => [03]. INTERNAL-THREAD.

- B. SURFACE. => [04]. INTERNAL-KEYWAY.
- => [05]. RECESS.

- C. RADIAL. => [06]. CENTRE-DRILL.
- => [07]. INTERNAL-CHAMFER.

- => [08]. EXTERNAL FEATURES.

?: SELECT internal feature type : [01] - [07],
or ENTER [08] for external features,
or ENTER [00] to end the operation.

01

```
*****
```

(continued)

```

#####
% ----- %%
%   Data input for CYLINDER-BORE feature that   %%
%   is required for design and process planning. %%
%   ----- %%
#####

```

1: Enter the LENGTH of cylinder-bore :
(2-digits)

18

2: Enter the RADIUS of cylinder-bore :
(2-digits)

13

3: Enter the POSITION of cylinder-bore :
(3-digits)

003

The parameters for CYLINDER-BORE feature are as follows :

- 1. Length = 18
- 2. Radius = 13
- 3. Position = 003

?: Any data to correct ? (y/n/cancel)

n

```

#####
#####
%   ===== %%
%   INTERNAL FEATURES TYPE   %%
%   ===== %%
#####

```

Internal features are subclassified
into three (3) types :

- A. AXIAL. => [01]. CYLINDER-BORE.
- => [02]. TAPER-BORE.
- => [03]. INTERNAL-THREAD.

- B. SURFACE. => [04]. INTERNAL-KEYWAY.
- => [05]. RECESS.

- C. RADIAL. => [06]. CENTRE-DRILL.
- => [07]. INTERNAL-CHAMFER.

- => [08]. EXTERNAL FEATURES.

?: SELECT internal feature type : [01] - [07],
or ENTER [08] for external features,
or ENTER [00] to end the operation.

07

Which type of chamfer for internal-chamfer feature ?

- [1]. Right-internal-chamfer
- [2]. Left-internal-chamfer

?: Enter only selections listed : [1] - [2]

2

(continued)

```

*****
*****
% ----- %%
% Data input for LEFT-INTERNAL-CHAMFER feature that %%
% is required for design and process planning. %%
% ----- %%
*****

```

- 1: Enter the LENGTH of left-internal-chamfer :
(2-digits)
03
- 2: Enter the MINOR-RADIUS of left-internal-chamfer :
(2-digits)
13
- 3: Enter the MAJOR-RADIUS of left-internal-chamfer :
(2-digits)
16
- 4: Enter the ANGLE of left-internal-chamfer :
(2-digits)
45
- 5: Enter the POSITION of left-internal-chamfer :
(3-digits)
000

The parameters for LEFT-INTERNAL-CHAMFER feature are as follows :

- 1. Length = 03
- 2. Minor-Radius = 13
- 3. Major-Radius = 16
- 4. Angle = 45
- 5. Position = 000

? : Any data to correct ? (y/n/cancel)
n

```

*****
*****
% ===== %%
% INTERNAL FEATURES TYPE %%
% ===== %%
*****

```

Internal features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER-BORE.
=> [02]. TAPER-BORE.
=> [03]. INTERNAL-THREAD.
- B. SURFACE. => [04]. INTERNAL-KEYWAY.
=> [05]. RECESS.
- C. RADIAL. => [06]. CENTRE-DRILL.
=> [07]. INTERNAL-CHAMFER.
=> [08]. EXTERNAL FEATURES.

(continued)

```

%                                                     %%
%               [E]XIT THE SYSTEM                     %%
%                                                     %%
%                                                     %%
%                                                     %%
%*****%

```

? : Select desired option :
 or enter [E] to exit the system.

c

? : Do you want to CREATE a new files ? (y/n)

y.

```

%*****%
%*****%
%*****%
% *****%
% In order to create the geometry specification and %%
% an outline process planning files, make sure %%
% you enter the output filenames. %%
% *****%
%*****%

```

? : Enter the geometry specification filename :

box3

? : Enter the outline process plan filename :

opp3

```

%*****%
%*****%
% *****%
% GENERAL INFORMATION ON COMPONENT %%
% *****%
%*****%

```

ROUND BAR STOCK MATERIALS AVAILABLE

- [1]. Medium Carbon Steel Round Bar.
- [2]. Mild Steel Round Bar.
- [3]. Aluminium Round Bar.
- [4]. Cast Iron Round Bar.
- [5]. Brass Round Bar.
- [6]. None of these.

? : Enter only selections listed : [1] - [6]

5

? : Enter the LENGTH of round bar (in mm) :
 (3-digits only)

050

? : Enter the RADIUS of round bar (in mm) :
 (2-digits only)

20

```

%*****%
%*****%
% *****%
% DESIGN AND PROCESS PLANNING INFORMATION %%
% *****%
%*****%

```

(continued)

?: Strike a [RETURN] key for feature menu selection :

```
#####  
#####  
% ***** %  
% THE FEATURE MENU SELECTIONS OF THE SYSTEM %%  
% FOR DESIGN AND PROCESS PLANNING %%  
% ***** %  
#####
```

Features are classified into two (2) types :

[01]. EXTERNAL FEATURES.

[02]. INTERNAL FEATURES.

?: SELECT feature option : [01] or [02]

01

```
#####  
#####  
% ===== %%  
% EXTERNAL FEATURES TYPE %%  
% ===== %%  
#####
```

External features are subclassified
into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15],
or ENTER [16] for internal features,
or ENTER [00] to end the operation.

05

```
#####  
#####  
% ----- %%  
% Data input for EXTERNAL-THREAD feature that %%  
% is required for design and process planning. %%  
% ----- %%
```

(continued)

1: Enter the LENGTH of external-thread :
(2-digits)

18

2: Enter the NOMINAL-RADIUS of external-thread :
(2-digits)

08

3: Enter the PITCH of external-thread :
(2-digits)

04

TYPES OF THREAD AVAILABLE

=====

- [1]. V-Thread
- [2]. Acme
- [3]. Whitworth
- [4]. Square
- [5]. Unified
- [6]. None of these

4: Enter only selections listed : [1] - [6]

2

5: Enter the POSITION of thread :
(3-digits)

032

The parameters for THREAD feature are as follows :

- 1. Length = 18
- 2. Nominal-Radius = 08
- 3. Pitch = 04
- 4. Pattern = Acme
- 5. Position = 032

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****
%      =====      %%
%      EXTERNAL FEATURES TYPE      %%
%      =====      %%
*****

```

External features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.

(continued)

- => [11]. KEYWAY.
- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.
- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15],
 or ENTER [16] for internal features,
 or ENTER [00] to end the operation.

12

```

#####
#####
% ----- %%
%   Data input for UNDERCUT feature that is   %%
%   required for design and process planning. %%
%   ----- %%
#####

```

1: Enter the LENGTH of undercut :
 (2-digits)

02

2: Enter the RADIUS of undercut :
 (2-digits)

06

3: Enter the POSITION of undercut :
 (3-digits)

030

The parameters for UNDERCUT feature are as follows :

- 1. Length = 02
- 2. Radius = 06
- 3. Position = 030

?: Any data to correct ? (y/n/cancel)

n

```

#####
#####
% ===== %%
%   EXTERNAL FEATURES TYPE   %%
%   ===== %%
#####

```

External features are subclassified
 into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.

(continued)

- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.
- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15],
 or ENTER [16] for internal features,
 or ENTER [00] to end the operation.

01

```

*****
*****
% ----- %%
% Data input for CYLINDER feature that is %%
% required for design and process planning. %%
% ----- %%
*****

```

1: Enter the LENGTH of cylinder :
 (2-digits)

08

2: Enter the RADIUS of cylinder :
 (2-digits)

16

3: Enter the POSITION of cylinder :
 (3-digits)

022

The parameters for CYLINDER feature are as follows :

- 1. Length = 08
- 2. Radius = 16
- 3. Position = 022

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****
% ===== %%
% EXTERNAL FEATURES TYPE %%
% ===== %%
*****

```

External features are subclassified
 into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

(continued)

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15],
 or ENTER [16] for internal features,
 or ENTER [00] to end the operation.

12

```

#####
#####
% ----- %%
%   Data input for UNDERCUT feature that is   %%
%   required for design and process planning. %%
% ----- %%
#####

```

1: Enter the LENGTH of undercut :
 (2-digits)

04

2: Enter the RADIUS of undercut :
 (2-digits)

14

3: Enter the POSITION of undercut :
 (3-digits)

018

The parameters for UNDERCUT feature are as follows :

- 1. Length = 04
- 2. Radius = 14
- 3. Position = 018

?: Any data to correct ? (y/n/cancel)

n

```

#####
#####
% ===== %%
%   EXTERNAL FEATURES TYPE   %%
% ===== %%
#####

```

External features are subclassified
 into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.

(continued)

- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

? : SELECT external feature type : [01] - [15],
or ENTER [16] for internal features,
or ENTER [00] to end the operation.

15

Which type of chamfer for chamfer feature ?

- [1]. Right-chamfer
- [2]. Left-chamfer

? : Enter only selections listed : [1] - [2]

1

```
*****  
*****  
% ----- %%  
% Data input for RIGHT-CHAMFER feature that %%  
% is required for design and process planning. %%  
% ----- %%  
*****
```

1: Enter the LENGTH of right-chamfer :
(2-digits)

02

2: Enter the MINOR-RADIUS of right-chamfer :
(2-digits)

16

3: Enter the MAJOR-RADIUS of right-chamfer :
(2-digits)

18

4: Enter the ANGLE of right-chamfer :
(2-digits)

45

5: Enter the POSITION of right-chamfer :
(3-digits)

016

The parameters for RIGHT-CHAMFER feature are as follows :

- 1. Length = 02
- 2. Minor-Radius = 16
- 3. Major-Radius = 18
- 4. Angle = 45
- 5. Position = 016

(continued)

?: Any data to correct ? (y/n/cancel)

n

```
#####  
#####  
%      =====      %%  
%      EXTERNAL FEATURES TYPE      %%  
%      =====      %%  
#####
```

External features are subclassified
into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
 => [02]. FACE.
 => [03]. TAPER.
 => [04]. KNURL.
 => [05]. THREAD.
 => [06]. PINION.
 => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
 => [09]. UNROUND-HOLE.
 => [10]. ACROSS-FLAT.
 => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
 => [13]. FORMCUT.
 => [14]. ROUND.
 => [15]. CHAMFER.

 => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15],
or ENTER [16] for internal features,
or ENTER [00] to end the operation.

01

```
#####  
#####  
%      =====      %%  
%      Data input for CYLINDER feature that is      %%  
%      required for design and process planning.      %%  
%      =====      %%  
#####
```

1: Enter the LENGTH of cylinder :
(2-digits)

14

2: Enter the RADIUS of cylinder :
(2-digits)

18

3: Enter the POSITION of cylinder :
(3-digits)

002

The parameters for CYLINDER feature are as follows :

- 1. Length = 14
- 2. Radius = 18

(continued)

3. Position = 002

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****
%      =====      %%
%      EXTERNAL FEATURES TYPE      %%
%      =====      %%
*****

```

External features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE..

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15], or ENTER [16] for internal features, or ENTER [00] to end the operation.

15

Which type of chamfer for chamfer feature,?

- [1]. Right-chamfer
- [2]. Left-chamfer

?: Enter only selections listed : [1] - [2]

2

```

*****
*****
%      -----      %%
%      Data input for LEFT-CHAMFER feature that is %%
%      required for design and process planning.  %%
%      -----      %%
*****

```

1: Enter the LENGTH of left-chamfer : (2-digits)

02

2: Enter the MINOR-RADIUS of left-chamfer : (2-digits)

(continued)

16

3: Enter the MAJOR-RADIUS of left-chamfer :
(2-digits)

18

4: Enter the ANGLE of left-chamfer :
(2-digits)

45

5: Enter the POSITION of left-chamfer :
(3-digits)

002

The parameters for LEFT-CHAMFER feature are as follows :

- 1. Length = 02
- 2. Minor-Radius = 16
- 3. Major-Radius = 18
- 4. Angle = 45
- 5. Position = 002

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****
*          *****          *
*          EXTERNAL FEATURES TYPE          *
*          *****          *
*****

```

External features are subclassified
into three (3) types :

- A. AXIAL. => [01]. CYLINDER.
- => [02]. FACE.
- => [03]. TAPER.
- => [04]. KNURL.
- => [05]. THREAD.
- => [06]. PINION.
- => [07]. SPLINE.

- B. SURFACE. => [08]. CROSS-HOLE.
- => [09]. UNROUND-HOLE.
- => [10]. ACROSS-FLAT.
- => [11]. KEYWAY.

- C. RADIAL. => [12]. UNDERCUT.
- => [13]. FORMCUT.
- => [14]. ROUND.
- => [15]. CHAMFER.

- => [16]. INTERNAL FEATURES.

?: SELECT external feature type : [01] - [15],
or ENTER [16] for internal features,
or ENTER [00] to end the operation.

16

```

*****
*****
*          *****          *

```

(continued)

```

%      INTERNAL FEATURES TYPE      %%
%      =====                %%
%      ~~~~~

```

Internal features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER-BORE.
- => [02]. TAPER-BORE.
- => [03]. INTERNAL-THREAD.

- B. SURFACE. => [04]. INTERNAL-KEYWAY.
- => [05]. RECESS.

- C. RADIAL. => [06]. CENTRE-DRILL.
- => [07]. INTERNAL-CHAMFER.

- => [08]. EXTERNAL FEATURES.

?: SELECT internal feature type : [01] - [07], or ENTER [08] for external features, or ENTER [00] to end the operation.

02

Which type of taper for taper-bore feature ?

- [1]. Right-taper-bore
- [2]. Left-taper-bore

?: Enter only selections listed : [1] - [2]

1

```

%      ~~~~~                %%
%      Data input for RIGHT-TAPER-BORE feature that %%
%      is required for design and process planning. %%
%      ~~~~~                %%

```

1: Enter the LENGTH of right-taper-bore : (2-digits)

34

2: Enter the MINOR-RADIUS of right-taper-bore : (2-digits)

04

3: Enter the MAJOR-RADIUS of right-taper-bore : (2-digits)

06

4: Enter the POSITION of right-taper-bore : (3-digits)

050

The parameters for RIGHT-TAPER-BORE feature are as follows :

- 1. Length = 34
- 2. Minor-Radius = 04
- 3. Major-Radius = 06
- 4. Position = 050

(continued)

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****
%      =====      %%
%      INTERNAL FEATURES TYPE      %%
%      =====      %%
*****

```

Internal features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER-BORE.
- => [02]. TAPER-BORE.
- => [03]. INTERNAL-THREAD.

- B. SURFACE. => [04]. INTERNAL-KEYWAY.
- => [05]. RECESS.

- C. RADIAL. => [06]. CENTRE-DRILL.
- => [07]. INTERNAL-CHAMFER.
- => [08]. EXTERNAL FEATURES.

?: SELECT internal feature type : [01] - [07],
or ENTER [08] for external features,
or ENTER [00] to end the operation.

01

```

*****
*****
% ----- %%
% Data input for CYLINDER-BORE feature that %%
% is required for design and process planning. %%
% ----- %%
*****

```

1: Enter the LENGTH of cylinder-bore :
(2-digits)

16

2: Enter the RADIUS of cylinder-bore :
(2-digits)

04

3: Enter the POSITION of cylinder-bore :
(3-digits)

000

The parameters for CYLINDER-BORE feature are as follows :

- 1. Length = 16
- 2. Radius = 04
- 3. Position = 000

?: Any data to correct ? (y/n/cancel)

n

```

*****
*****

```

(continued)

```

%      =====      %%
%      INTERNAL FEATURES TYPE      %%
%      =====      %%
%      %%%%%%%%%%%%%%%%%%%%%%%%%%%

```

Internal features are subclassified into three (3) types :

- A. AXIAL. => [01]. CYLINDER-BORE.
- => [02]. TAPER-BORE.
- => [03]. INTERNAL-THREAD.

- B. SURFACE. => [04]. INTERNAL-KEYWAY.
- => [05]. RECESS.

- C. RADIAL. => [06]. CENTRE-DRILL.
- => [07]. INTERNAL-CHAMFER.

- => [08]. EXTERNAL FEATURES.

?: SELECT internal feature type : [01] - [07], or ENTER [08] for external features, or ENTER [00] to end the operation.

00

?: Do you want to SAVE the files ? (y/n)

y

```

%      %%%%%%%%%%%%%%%%%%%%%%%%%%%
%      %%%%%%%%%%%%%%%%%%%%%%%%%%%
%      *****
%      THE MAIN MENU SELECTIONS
%      OF THE CADOPP SYSTEM
%      *****
%      %%%%%%%%%%%%%%%%%%%%%%%%%%%
%      %%%%%%%%%%%%%%%%%%%%%%%%%%%
%      %%%%%%%%%%%%%%%%%%%%%%%%%%%
%      [C]REATE NEW FILES
%      [M]ODIFY FILES
%      [R]EVIEW FILES
%      [D]ELETE FILES
%      [E]XIT THE SYSTEM
%      %%%%%%%%%%%%%%%%%%%%%%%%%%%
%      %%%%%%%%%%%%%%%%%%%%%%%%%%%

```

?: Select desired option : or enter [E] to exit the system.

e

?: Are you sure to exit the system ? (y/n)

y

CADOPP TERMINATED and Fortran STOP.

