DCU

Implementation of Hierarchical Design for Manufacture Rules in Manufacturing Processes

By

MD. MASUD PARVEZ

B. Sc. Engg. (ME). M ASME

Supervisor

DR. TAMAS SZECSI

School of Mechanical and Manufacturing Engineering
Dublin City University

M.Eng. **2008**

DECLARATION

I hereby declare that this material, which I now submitted for assessment on the programme of study leading to the award of Master of Engineering is entirely my own work and has not been taken from the work of others and to the extent that such work has been cited and acknowledged within the text of my work.

Signed_________ 21.10.08

ID No: 54179408

DEDICATION

This thesis is dedicated to my late father Joynal Abedin and my mother Rahima Khatun whose inspiration and constant support allowed me to reach to this success and who are to me the symbol of love, trust and greatest personality.

PUBLICATIONS ARISING FROM THIS RESEARCH

- **M. M. Parvez**, and T. Szecsi, "Implementation of Design for Manufacture Rules in Slot Features", peer reviewed *Proceedings of the 9th Cairo University International Conference on Mechanical Design & Production (MDP-9)*, pp. 204-215, Cairo, Egypt, January 8-10, 2008.

- **M. M. Parvez,** A. S. M. Hoque and T. Szecsi, "A study of Different Manufacturing Feature Recognition Methods", *International Internet Scientific Conference on MECHANICS,* June 26-27, 2008.

- **M. M. Parvez,** A. S. M. Hoque and T. Szecsi, "Implementing Design for Manufacture Rules in Pocket Features", Journal of *The Proceedings of International Conference on Manufacturing System ICMaS (ISSN 1842-3183),* University POLITEHNICA Bucharest, Machine and Production Systems Department, in November 13-14, 2008. (Submitted).

ACKNOWLEDGEMENTS

The author would like to wish his heartfelt gratitude and profound indebtness to his supervisor Dr. Tamas Szecsi, Senior lecturer School of Mechanical and Manufacturing Engineering, Dublin City University, Dublin 9, Ireland, for his constant guidance, friendly enthusiasm, valuable suggestions, encouragement, beneficent help and practical hand on assistance in carrying out the research work as well as in writing this thesis.

The author would like to express special thanks to Prof. M. S. J. Hashmi, Head of the School of Mechanical and Manufacturing Engineering, Dublin City University, for his continued guidance in all aspects during this research work.

Sincere thanks are due to the school technicians for their generous assistance; especially appreciation goes to Mr. Liam Domican, Senior Technician and Mr. Keith Hickey, IT Administrator, thanks are also due to the officers and staff of the school for their help in various stages of the study. Thanks also to all of my house mates, friends and colleagues.

The author wishes to express his heartfelt gratitude and love to his wife Manira Akter for her continuous love, support and inspiration.

At last but never the least, the author would like to express his gratitude to his family, especially his brother-in-law and sisters for their continued encouragement, inspiration and support throughout his life as well as this research work.

NOMENCLATURE

EDM = Electrical-Discharge Machining

ECM = Electrochemical Machining

CM = Chemical Machining

AJM = Abrasive-Jet Machining

AFM = Abrasive-Flow Machining

USM = Ultrasonic Machining

EBM = Electron-Beam Machining

LBM = Laser- Beam Machining

C_{duCNC} = Drilling unit level cost for CNC machine in euros per part

C_{muCNC} = Milling unit level cost for CNC machine in euros per part

C_{pt} = Power cost in euros per day

C_{tk} = Tooling cost in euros per part for category k

d_k = Depth of hole in category k in mm

d_m = Diameter of milled hole in mm

D_m = Diameter of the end-milling cutter in mm

E = Efficiency factor (ratio between productive and non productive time)

f = Feed rate (mm/rev for drilling) or (mm/tooth for milling)

H_k = Holes (or features) per hit in category k

k = Hole (or feature) category number

t_{tc} = Tool change time

T_k = Tool life for category k

M = Hourly machine (or labour) rate in euros per hour

IMPLEMENTATION OF HIERARCHICAL DESIGN FOR MANUFACTURE RULES IN MANUFACTURING PROCESSES

By

Md. Masud Parvez, B. Sc. Engg. (ME)

ABSTRACT

In order to shorten the product development cycle time, minimise overall cost and smooth transition into production, early consideration of manufacturing processes is important. Design for Manufacture (DFM) is the practice of designing products with manufacturing issues using an intelligent system, which translates 3D solid models into manufacturable features. Many existing and potential applications, particularly in the field of manufacturing, require various aspects of features technology. In all engineering fields geometric modelling which accurately represents the shape of a whole engineering component has become accepted for a wide range of applications. To apply DFM rules or guidelines in manufacturing processes, they have to be systematised and organised into a hierarchical rule system. Rules at the higher level of the hierarchical system are applied to more generic manufacturing features, and specific rules are applied to more detailed features. This enables the number of rules and amount of repetition to be minimised. Violation of the design for manufacture rules in the features, their characteristics and manufacturing capabilities are further examined in this hierarchical system. Manufacturability analysis, such as production type, materials, tolerances, surface finish, feature characteristics and accessibility, are also taken into consideration.

Consideration of process capabilities and limitations during the design process is necessary in order to minimise production time and as a result, manufacturing cost. The correct selection of manufacturing processes is also important as it is related to the overal cost.

As a result of this research, a hierarchical design for manufacture rule system is proposed which would aid designers in avoiding designs that would lead to costly manufacturing processes.

TABLE OF CONTENTS

CHAPTER 3 FEATURE CLASSIFICATIONS

CHAPTER 4 HIERARCHICAL DESIGN FOR MANUFACTURE RULES

CHAPTER 5 MACHINING COST COMPARISON OF TWO MANUFACTURING PROCESSES

CHAPTER 6 CONCLUSIONS AND FUTURE WORK

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1 INTRODUCTION

1.1 Introduction

The traditional approach to engineering and design has been to start designing in order to fulfil a design specification, then to figure out how to manufacture it and following this waiting to see how the product performs in the field. This appears to be an incorrect approach. Design for Manufacture (DFM) is the general engineering artistic creation of designing products in such a way that they are easy to manufacture. The introductory idea exists almost in all engineering disciplines, but there has been some difference of opinion in details depending on the manufacturing technology. The DFM approach has become really interesting as it has been found that the design stage determines most of the cost of the development of a product. In order to fulfil market demands and competition, reduction of production development cycle time is a crucial issue To achieve a high quality product at low cost, it is necessary to apply manufacturing constraints from the very beginning of the design stage. This is important to avoid major modifications to the product during the development cycle which would result in higher cost.

The principles of DFM and its application are not really new. The idea was introduced by Eli Whitney who developed the interchangeable parts concept. The intensive development and progress in DFM has played an important role in producing high performance hardware and software at affordable prices in the computer industry during the last decade. However, there is still a lot to do in the field of computerisation of DFM. In DFM the interaction between designers and engineers is minimal and manufacturing issues are superficially considered from the beginning of a design. DFM is the tool that enhances a number of general rules about the manufacturability of a part.

On the simple level of manufacturability, DFM for a part involves details such as determining where a hole or slot etc. is to be located and what the specifications will be. But at a more complex level, product DFM tackles the more fundamental problem of deciding on the product structure and form.

1.2 Purpose of This Study

Currently, due to the demands of customers the amount of products being produced is becoming progressively higher and, in order to satisfy the specified demands, products are becoming more and more complex in shape. In order to satisfy the customer demands, not only high product quality but also the competitive price should be taken into consideration. The most important consideration is that poor design impacts on product cost. Hence, DFM rules play a significant role in allowing co-operation between the designer and the manufacturer.

The purpose of this research is to develop a hierarchical design for manufacture system in order to implement the DFM rules which can help the designers during the design stage with manufacturing constraints information.

1.3 Structure of Thesis

Chapter 1- **Introduction**

An outline of the approach for implementation of Design for Manufacture rules during the design stage.

Chapter 2- **Literature Review**

A wide discussion of different feature recognition methods is explained in this chapter. This discussion contains the necessity for feature recognition which is important in order to report the existence of a feature in the part including its attributes and relationships.

Chapter 3- **Feature Classification**

In this section of feature classification a new approach is applied for classification of features. Features such as hole feature, pocket feature, slot feature, boss feature and step feature have been associated with their possible characteristics and manufacturing processes.

Chapter 4- **Hierarchical Design for Manufacture Rules**

In this chapter Design for Manufacture rules are explained broadly. Rules at the higher level of the hierarchical system are applied to more generic manufacturing features, and more specific rules are applied to more detailed features.

Chapter 5- **Machining Cost Comparison of Two Manufacturing Processes**

An approximate cost estimation of 10 holes is calculated for two different manufacturing processes, milling and drilling, and the cost of each has been compared.

Chapter 6- **Conclusions and Future Work**

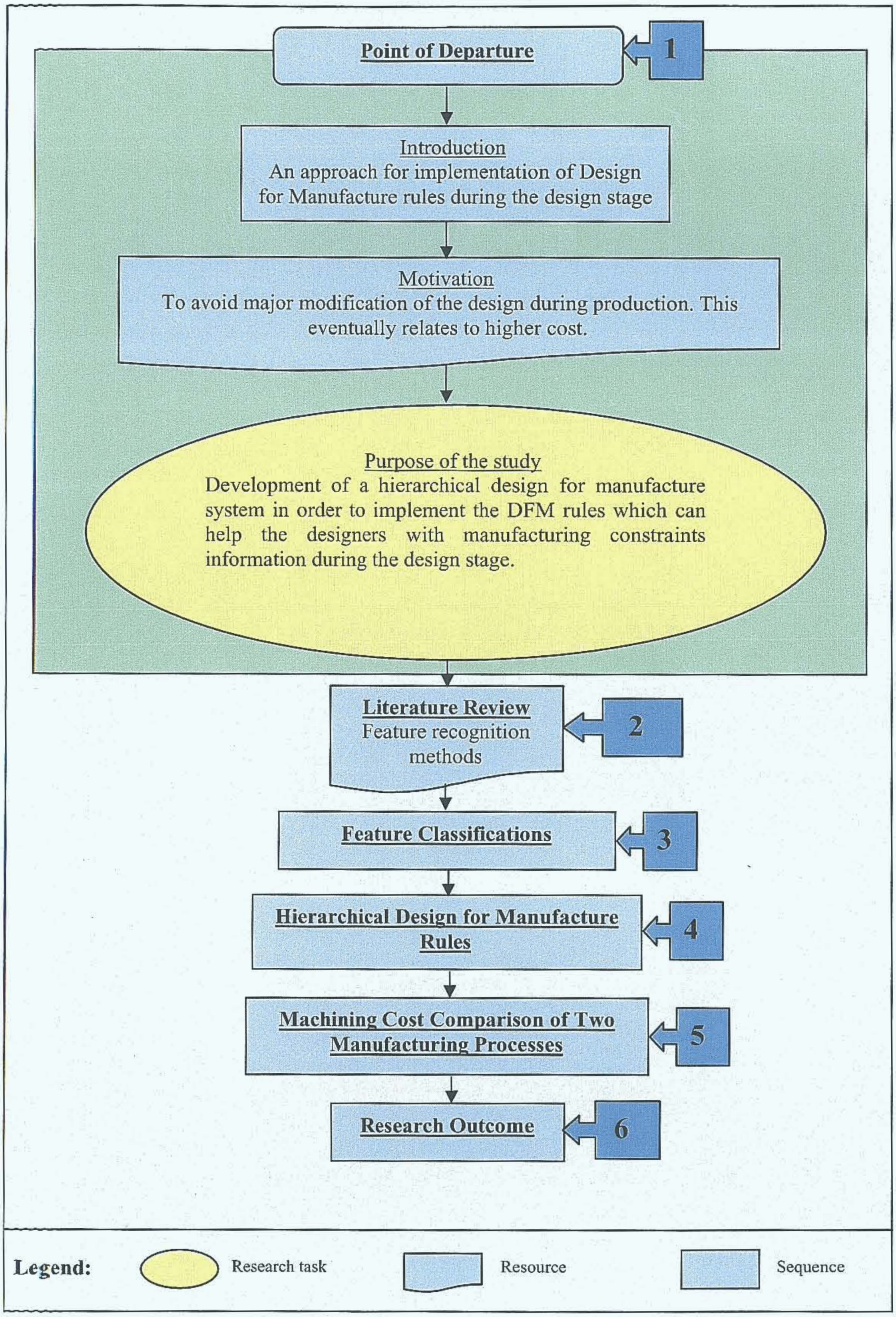


Fig. 1.1: Summary of the research process in chapter one.

CHAPTER 2 LITERATURE REVIEW

2.1 Introduction

Depending upon the manufacturing process, feature information is considered to be about volumes of material to be removed or to be added. Feature recognition is necessary in order to report the existence of a feature in the part, including its attributes and relationships. Feature recognition can be described as the finding of features within a geometric model after its creation. A geometric model accurately represents the shape of a whole engineering component which it makes easier to acknowledge where the slots, holes and their projecting slugs are.

Computer-Aided Process Planning (CAPP) is seen as a communication agent between CAD and CAM. The goal of CAPP is to generate a sequenced set of instructions used to manufacture the specified part by using the CAD data of a part. In order to do that, CAPP needs interpret the part in terms of features. Depending on the specific domain, the word "feature" signifies different meanings in different contexts. In design it refers to a web or a notch, etc., while in manufacturing it refers to slots, holes, and pockets, while in inspection it is used as a datum or reference on a part. Different ideas are presented from different backgrounds. Two of them are:
"A feature is a region of interest on the surface of a part" [1]
"Features are defined as geometric and topological patterns of interest in a part model and which represent high level entities useful in part analysis" [2]

Feature recognition is typically thought of as a process that is performed on a geometric model of a finished part but is not commonly employed in a design process.

2.2 Different Feature Recognition methods

The literature on feature recognition is large in volume. Various approaches and algorithms are proposed by different researchers. Different feature recognition methods are shown in fig. 2.1. In this section, different approaches and algorithms have been described.

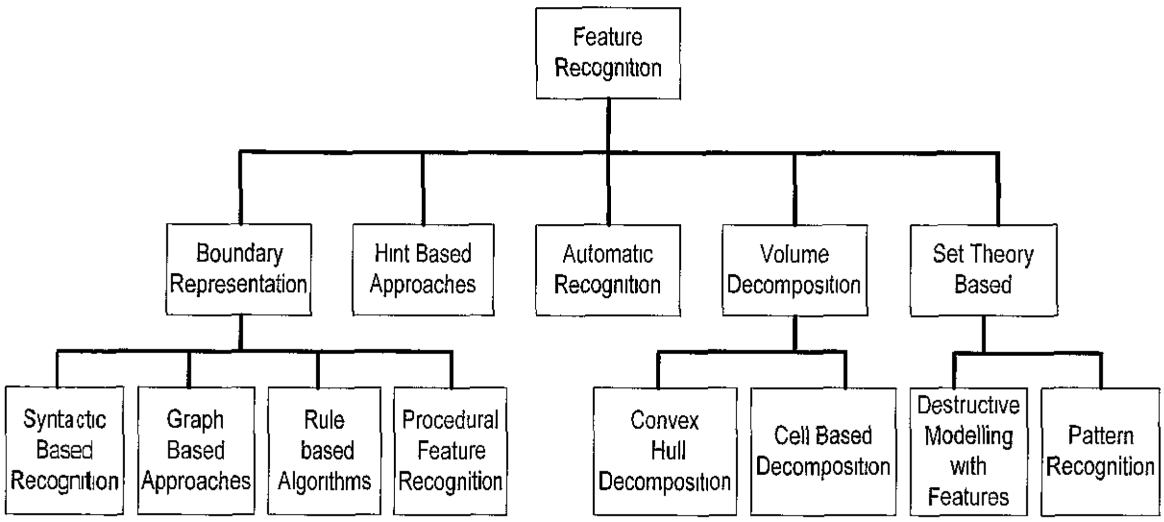


Fig. 2.1: Different feature recognition methods.

2.3 Boundary Representation Approaches

Boundary representation (B-rep) is one of the solid modelling methods that are extensively used in order to create a solid model of a physical object and also geometric data models [3]. Boundary representation describes the geometry of an object in terms of its boundaries, such as the vertices, edges, and surfaces which represent entities of zero dimension, one dimension and two dimensions respectively [4]. The orientation of each surface must be defined as the interior or exterior of the object in order to represent a solid object by its surfaces in which the inside is the material part and the outside is the void space.

A solid can be defined by a set of faces which is bounded by oriented surfaces. The topology of the solid model which presents the object as a set of faces is shown in fig. 2.2.

However, each face is bounded by edges and each edge is bounded by vertices. On the other hand, a loop such as a fastener consisting of a metal ring for lining a small hole to permit the attachment of cords or lines, or any feature with a round or oval shape, is formed by a curve that is closed and does not intersect itself. In order to separate the points that are inside or outside of the object, the direction of the surface normal is used to encode the face with information by numbering the edges in a sequence such that the right-hand rule defines the vector that points outward from the object [5].

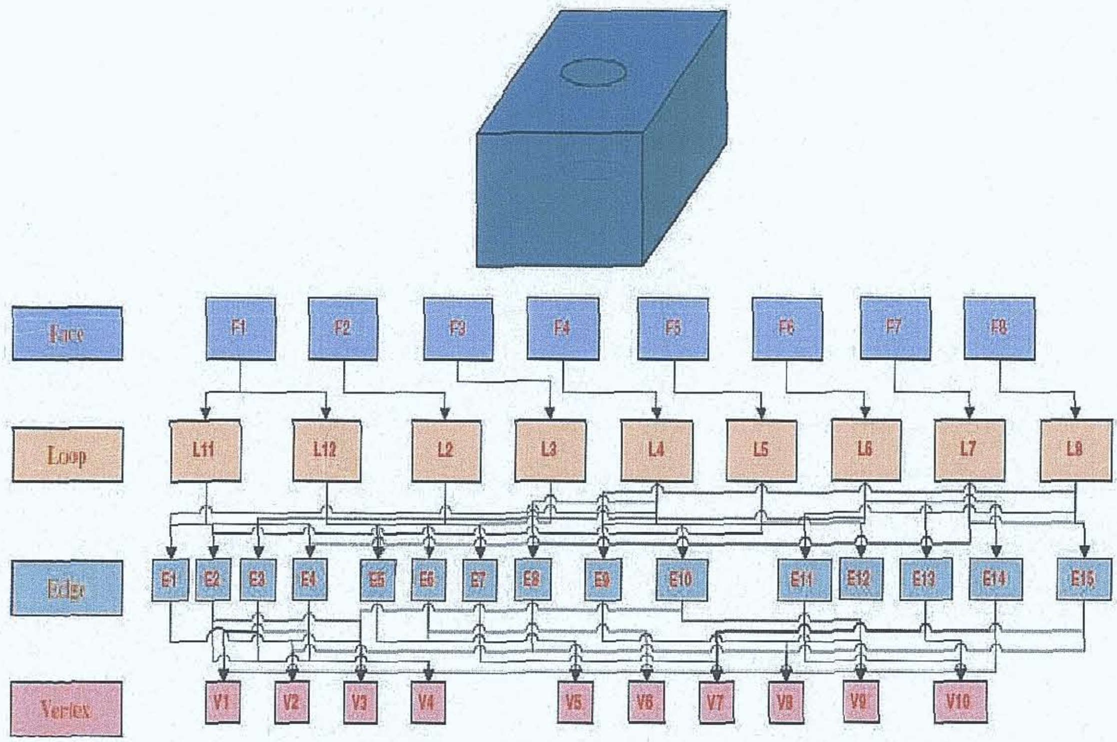


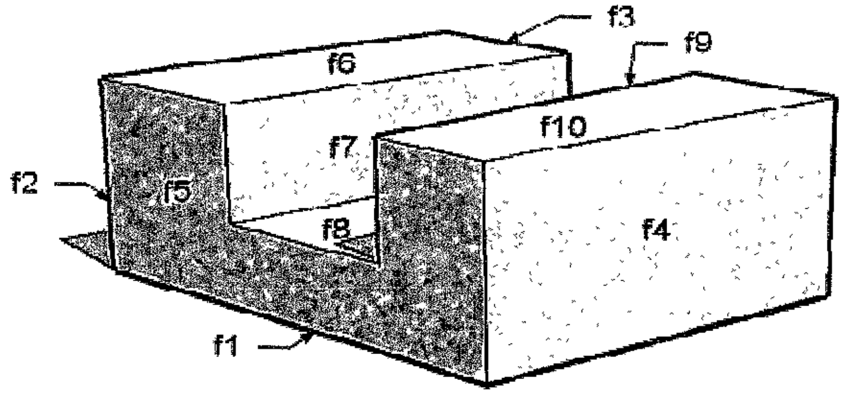
Fig. 2.2: A boundary representation (recreated from [10]).

The coordinates of the vertices and transformations (translations and rotations), metric information, such as distances, angles, area and volumes are all included in geometrical data. Many researchers used boundary representation to study machining features [6-9]. In order to identify the surfaces which are related to the shape and volume of the object, the topology relations must be stated between each set of surfaces using boundary representation.

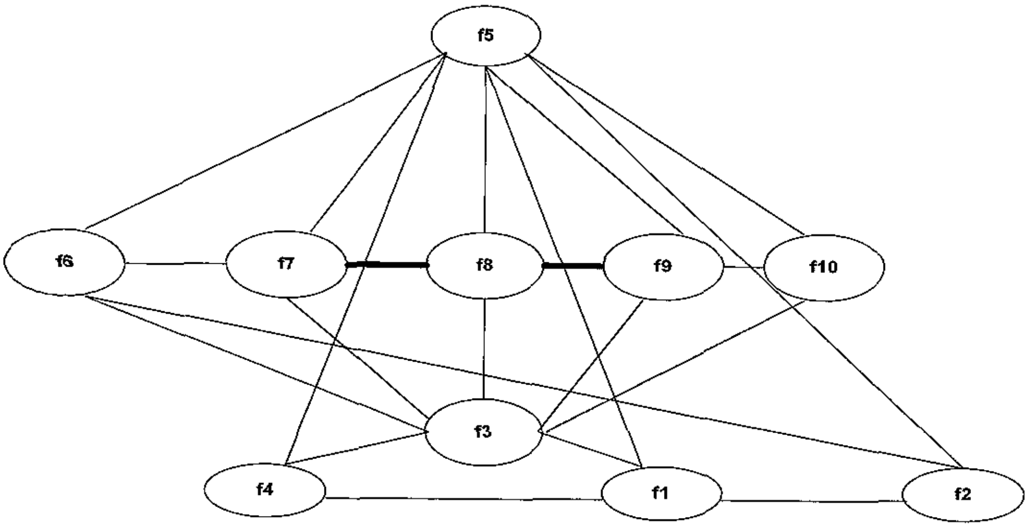
Parts can be classified as either polyhedral or curved objects. A polyhedral object (plane-based polyhedral) is presented by planar faces connected with straight edges, which in turn are connected at vertices [10]. However, a boundary representation model is not limited to a planar surface [11, 12]. Different types of surface geometries can be described by different Boundary representation models which approximate curved surfaces as a combination of planar surfaces.

2.3.1 Graph-Based Approaches

The graph pattern matching approach was first formalized by Joshi and Chang [13]. A graph pattern easily represents the boundary representation of a part where faces are considered to be nodes of the graph and face-face relationships are the arcs of the graph.



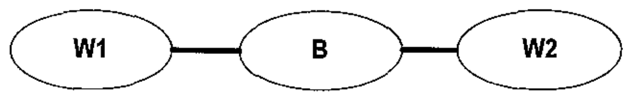
A solid part



Concave edge

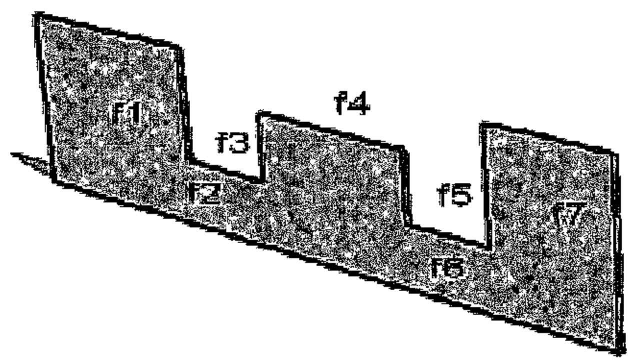
Convex edge

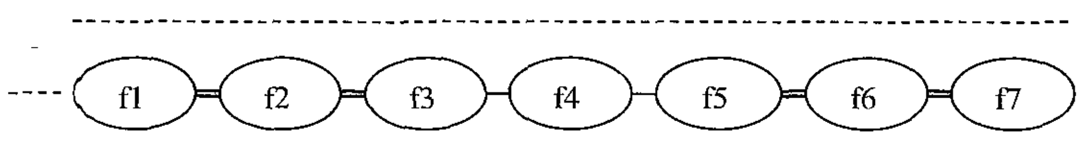
(a): A part and its graph representation.



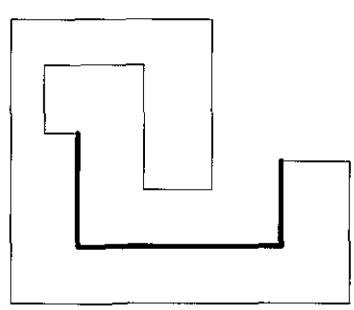
W1, W2: wall face
B: bottom face

(b): Slot template.





(c): Intersecting features



(d): Invalid slot

Fig. 2.3: Graph pattern analysis [14].

In fig. 2.3(a), it is seen that additional information, such as edge-convexity is incorporated into the graph. However, the part graph is then decomposed into subgraphs using heuristics and the face whose incident edges are all convex does not form part of a feature, which is deleted from the part graph. As a result in fig. 2.3 (a), nodes {f7, f8, f9} indicate a slot and all the nodes are deleted which is indicated in fig. 2.3 (b) as a template.

To understand the graph notation, the example of slot given in fig. 2.3 (a) can be studied and additional information may be incorporated into the graph, e.g., edge-convexity, face-orientation, etc. The neighbourhood relationships of the faces can be modelled by means of a face adjacency graph (FAG) in fig. 2.3 (a). Nodes of the graph represent the faces; the arcs represent the neighbourhood relationships between the faces.

Due to feature intersections the graph pattern analysis approach was quite successful in recognising isolated features. In fig. 2.3 (c) two subgraphs are produced, such as {f1, f2, f3} and {f5, f6, f7}. From them two slots can be recognised which are not enough to completely decompose the part. The heuristic does not always work when the features intersect and some advanced systems can recognise another slot {f1, f4, f7} which intersects with {f1, f2, f3} and {f5, f6, f7}.

Trika and Kashyp [15] in their work established an important contribution which is related to the issue of completeness. However, the input for feature recognisers is typically a solid model of the desired part, plus a solid model of the stock (raw material) from which material to be removed by machining, called the delta volume, is computed by subtracting the part from the stock as shown in fig. 2.4.

A feature recognition becomes complete for every part when the delta volume is contained in the union of all volumetric features generated by the feature recogniser. In fig. 2.4 (d) it is seen that a feature recogniser generates two features.

Unrecognised regions of the delta volume may exist if feature recognition is not completed and therefore the specified part may not be obtained though all feature removal operations are done, which is proved in Trika and Kashyp's work [15].

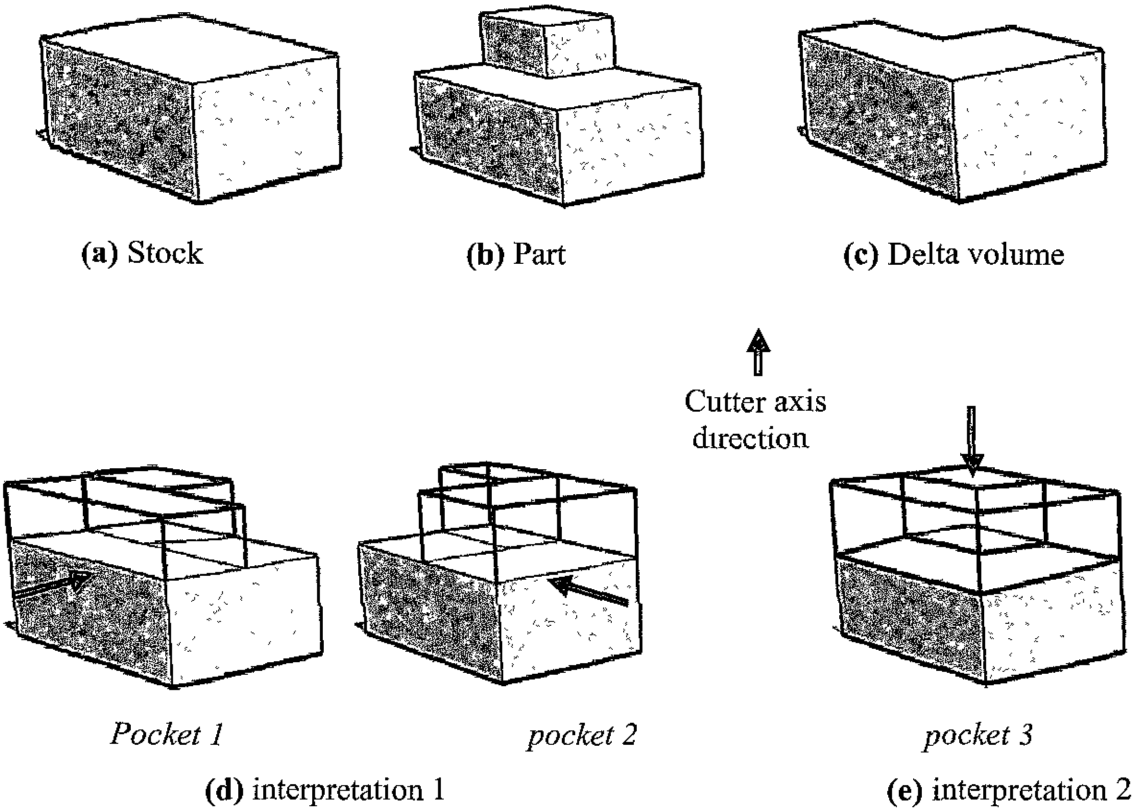


Fig. 2.4: Delta volume and multiple interpretations.

2.3.2 Syntactic-Based Recognition

Syntactic pattern recognition is a classical method for recognising shapes from raster images. Choi et al. [16] developed a syntax-based recognition system which works using a linguistic pattern-matching approach. Three surfaces, such as start surfaces, some element surfaces, and bottom surfaces are required for a valid feature. Fig. 2.5 establishes an example of a valid feature using this system.

In order to analyse the element surfaces at the bottom, the surfaces should be cylindrical. A hole can have a number of bottom surfaces; for example a flat bottom, a cone bottom or a through hole, each of these are distinguished by slight variations in the syntactic patterns [17].

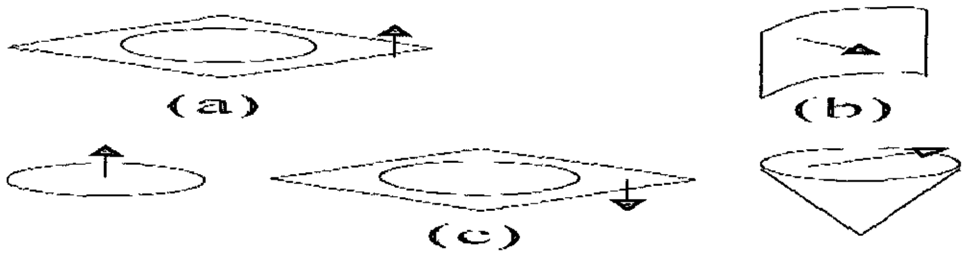


Fig. 2.5: (a) Hole Start Surface (HSS), (b) Hole Element Surface (HES), (c) Hole Bottom Surfaces (HBS).

Henderson [18] used a syntax-based approach in his system, rather than using the boundary representation as it stands, for the structure which is converted into PROLOG predicates. To compare the parts with feature patterns, features can then be located by running through the structure using predicate calculus.

2.3.3 Rule-Based Algorithms

In this approach, features are formalised by templates, defined for both general features (like holes) and specific features (e.g., flat bottomed, constant diameter hole) that consist of pattern rules. The hole begins with an entrance face in which all subsequent faces of the hole share a common axis. All faces of the hole are sequentially adjacent and the hole terminates with a valid hole bottom [19]. General features (depression, protrusion, passage), and classification of general features into specific features (T-slot, round hole, rectangular pocket, etc.) is recognised in this procedure by creating and subtracting the volume corresponding to each feature from the cavity and repeating the procedure until there are no residual entities.

2.3.4 Procedural Feature Recognition

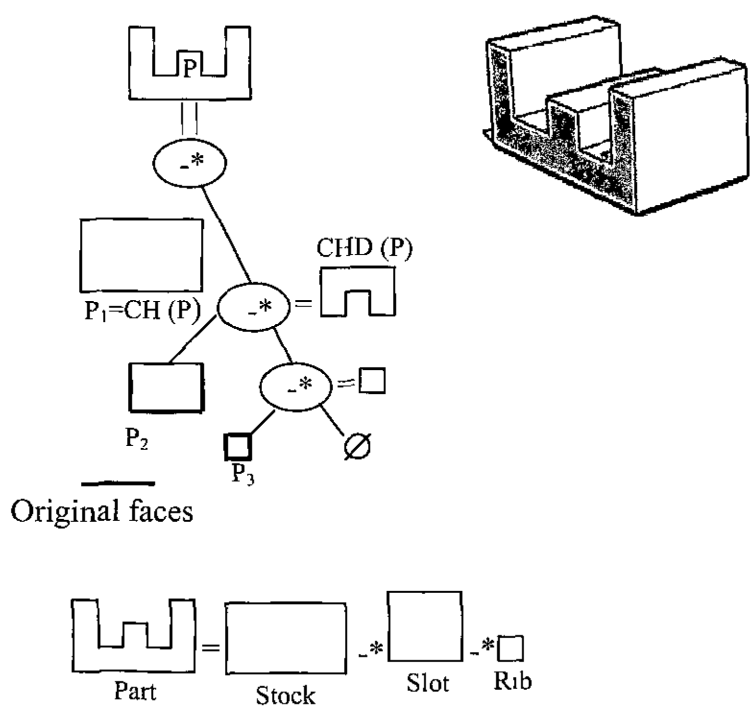
Pattern of feature definition is not the only method that can be used for feature recognition; purely procedural representations can be used. In this approach, the recognition is performed by a specialised procedure that can recognise features of a particular type. In the boundary model, references to relevant model entities can be scanned during the traversal and after all entities form a feature, the attributes of the feature are computed.

2.4 Volume Decomposition Approaches

In the previous section several important issues in feature recognition have been discussed. The most critical issue is how to recognise intersecting features. Two algorithms of intersecting features, which show similar characteristics, are discussed below In these algorithms input objects are decomposed into a set of intermediate volumes and then influence the volumes to produce features.

2.4.1 Convex Hull Decomposition

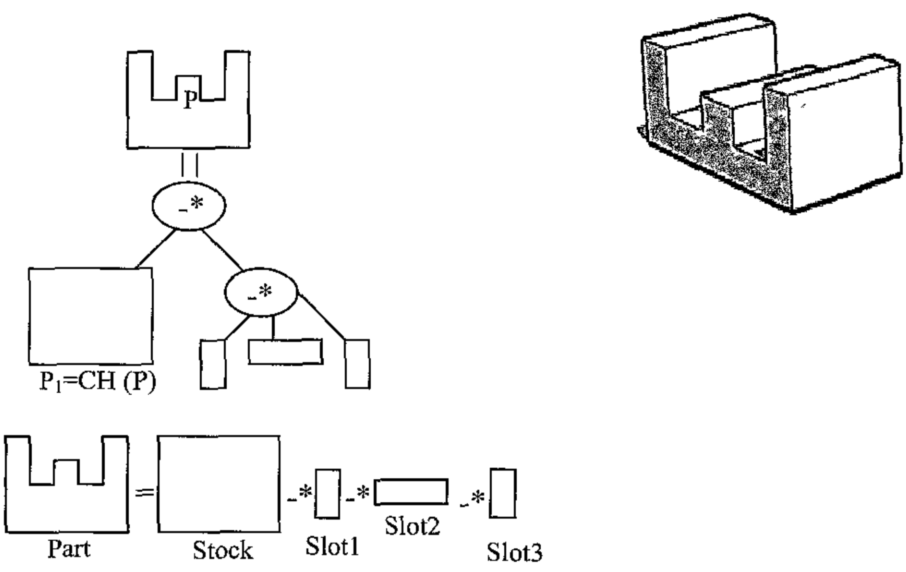
In 1980, Kyprianou in his work on seminal feature recognition originally developed the convex hull decomposition method [20]. The example in fig. 2.6 below considers machining of a solid rectangular work piece. The interpretation and mapping of the design features into machining features is done by using volume decomposition methods to identify the removal volumes from the initial work piece and attribute them to manufacturing features.



(a): Form feature model.

The faces, edges, and vertices of a geometric model, (i.e. 2-, 1- and 0 dimensional entities of the boundary representation) are used for feature recognition techniques. The convex hull CH (P) of polyhedron P is the smallest convex point set containing P. The convex hull difference CHD (P) is the regularised set difference (-*) between CH (P) and P. Conversely, P can be expressed as CH (P) -* CHD (P) [21]. The decomposition terminates if P is convex and CHD (P) is empty. Fig. 2.6 (a) shows the convex hull decomposition of an example part. After observing the pattern of alternating volume contributions, Woo [22] decided to call this an alternating sum of volumes (ASV) decomposition. However, ASV decomposition may not necessarily be adjacent. Then ASVP decomposition is proposed by Kim [23] which proved its adjacency.

Kim [24] proposed to use the ASVP decomposition to generate form features. However, in his approach, a form feature refers to a shape macro constructed for convenience, with limited connection to function or manufacturing. The faces of the given part are marked as original in the ASVP components. The ASVP component P_2 has three original faces which are transitively connected and is recognised as a form feature, classified as a slot. Similarly, P_3 is recognized as a rib. ASVP decomposition may have unrecognised components, specifically those with at most one original face or with separated original faces and for this reason two combination methods have been provided by Kim [24] in order to combine them with other components.



(b): Machining feature model

Fig. 2.6: Convex hull decomposition.

Although the combination method is applied, some volumetric components may not be recognized as form features; component P_2, shown in fig. 2.7, is an example.

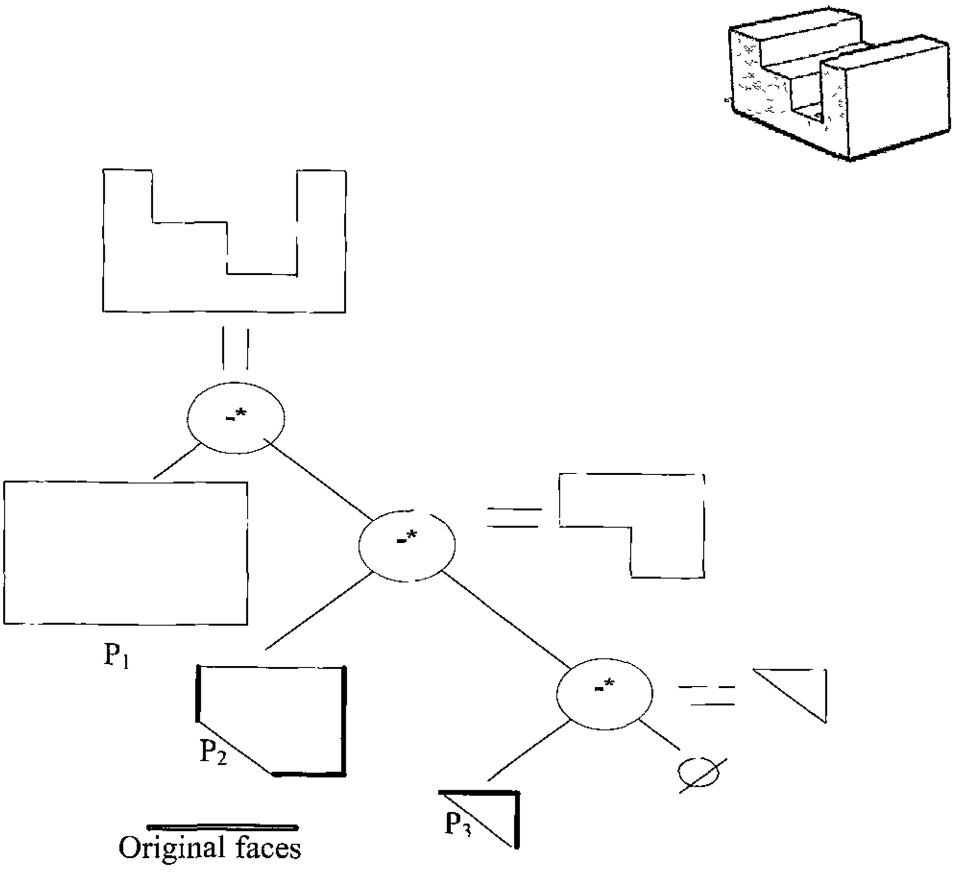


Fig. 2.7: Unrecognised ASVP component.

Waco and Kim [25-27] proposed that machine product could be generated by rewriting the Boolean expression of every positive form feature using the halfspaces determined by its original faces. Consider that all machining features are negative as they are subtracted from the workpiece. In fig. 2.6 (a), P_3 is an example of a positive form feature and similarly in fig. 2.6 (b), three negative features are seen which are considered to be three slots.

The new negative components are necessarily convex, and the algorithm often terminates with a set of clumsy shaped negative features. The aggregation of the primitive negative features and some conditions for aggregating primitive components is proposed by Waco and Kim [24].

From the computational geometry viewpoint, the convex hull decomposition approach is interesting. As discussed above in detail, the main problem with this method was that the operations in each step do not guarantee success and it may end up with an undesirable machining feature model. The feature activity generation based on the ASVP decomposition is presented by the recent work of Kim et al. [28].

The approach is inherently based around a polyhedral representation of the part which was another problem with this approach. To work in practical domains of curved parts involves the removal of curves, blends, fillets, etc., reducing the part to a polyhedral approximation; when finished, the results have to be converted back. The feature recognition algorithms for parts with cylindrical surfaces are proposed by Martino and Kim [29] which handle limited cases of feature intersection.

In fig. 2.8, a test result for Kim's algorithms on a variety of benchmark parts is shown [30]. The figure also demonstrates the recognition capability of Corney's graph-based system but the graph-based algorithms and the volumetric decomposition approach produced different sets of features and the main reason of this is that they have different definition sets of features.

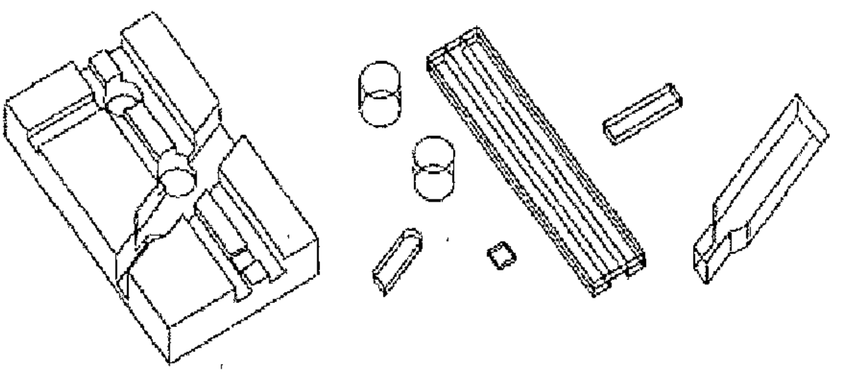


Fig. 2.8: Convex hull decomposition on a benchmark part (recreated from [30])

2.4.2 Cell Based Decomposition

In 1983, a research group from Allied Signal Aerospace in Kansas City explored the cell decomposition approach [31]. In 1994, Sakurai and Chin described the cell decomposition method, aimed at generating all possible machinable features accompanying a given part and stock. Demonstrated in fig. 2.9, the top left part was manufactured from a rectangular block of material. In the first step, a cross-shape delta volume is recognised as a feature. The top right figure shows that the delta volume has been partitioned into five convex cells. The partitioning is performed by splitting the delta volume with the extended surfaces contained in it; in this case the volume is split with the "side" surfaces of the cells. In the cell-based decomposition approach, the differences of the proposed algorithms mostly consist in the methods for combining cells into a feature. In the bottom part of fig. 2.9, it is shown that the various features have been generated by combining the cells; these include an open pocket, two long slots, and four smaller slots.

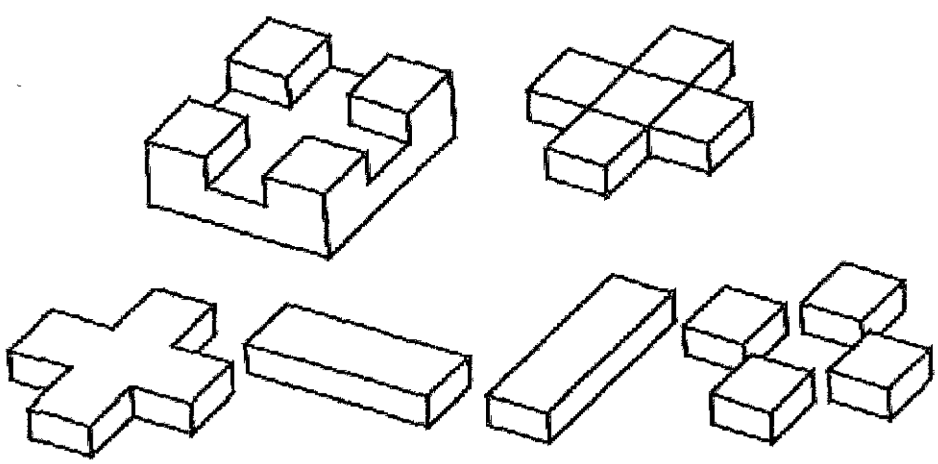
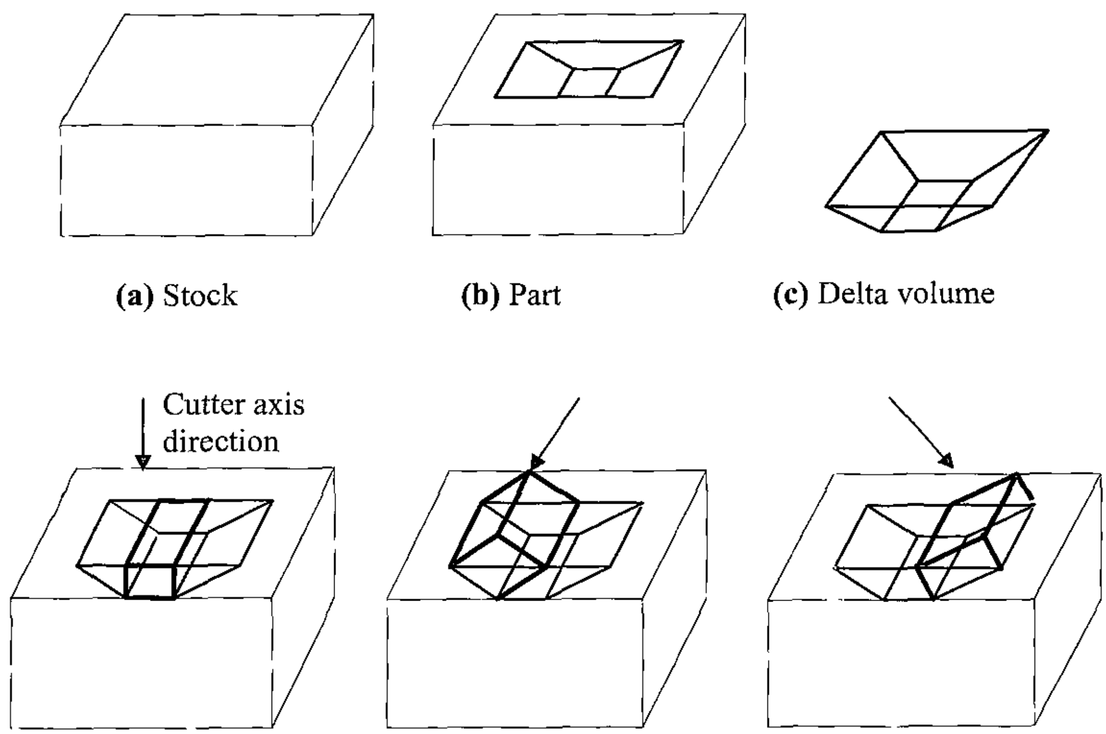


Fig. 2.9: Feature recognition by cell decomposition.

As a machining feature usually leaves its traces in a localised area of the part there is a problem of global effect in the local geometry. However, the cell decomposition step extends globally beyond the surfaces or halfspaces associated with the faces of the delta volume and quite often generates a huge number of cells.

Sakurai and Chin [32] proposed to generate all possible features even though some heuristics are used to crop unpromising compositions which the algorithm cannot avoid due to exponential time complexity. In order to compose the cells into convex volumes Coles et al. [33] proposed an approach but their approach is also subjected to combinatorial explosion.



(a) Stock **(b)** Part **(c)** Delta volume

(d) Desirable feature model

Fig. 2.10: Unclassified feature model.

A tractable composition algorithm which does not allow two features to share any cell is proposed by Shah et al. [34]. Starting from a cell, neighbouring cells are combined one at a time such that the intermediate volume remains convex. After that when no more combination is possible, the volume is deleted from the set of cells and in this way a new cell is selected and the same procedure is followed again.

The cell-based feature recognition is based on multiple-step reasoning such as cell decomposition, cell composition and feature classification which is similar to convex hull decomposition. A composed volume may not match with any predefined feature type and this is proved in Sakurai and Dave's algorithms [7]. Fig. 2.10(a), (b) and (c) show a stock, a part and the delta volume, respectively. In this example, the delta volume happens to be a single cell and therefore the cell classified as a feature which is shown in fig. 2.10 (d).

On the other hand, cell-based techniques in other feature applications, such as feature-based design and feature model conversion, have recently been reported. Cellular representations for feature models that can be used for a variety of feature applications has been presented by a research team led by W. Bronsvoort at Delft University of Technology (Netherlands) [36, 37].

2.5 Set Theory Based Approaches

A set- theoretic modeller needs little modification to encompass design-by-features technology, as the feature primitives along with their associated operators are stored explicitly within the modeller's data structure. By using the full characteristics of a set-theoretic geometric modeller Requicha [38] created a design-by-features interface.

2.5.1 Destructive Modelling with Features (Destructive Solid Geometry-DSG)

In 1982, Arbab [39] first developed the Destructive solid geometry technique following the manufacturing process of a 2 ½ D component closely. Although the name of this is the same as destructive solid geometry in design-by-features, the actual process is quite different and Li et al. developed the process [40, 41].

In design-by-features techniques the model is created by removing feature primitives by hand from a blank with the set difference operator but in the case of destructive modelling with features three set operators (Union, Intersection & Difference) and feature primitives are used to build the model. When the model is complete by entirely subtracted volumes the CSG is automatically traversed and modified to produce the DSG.

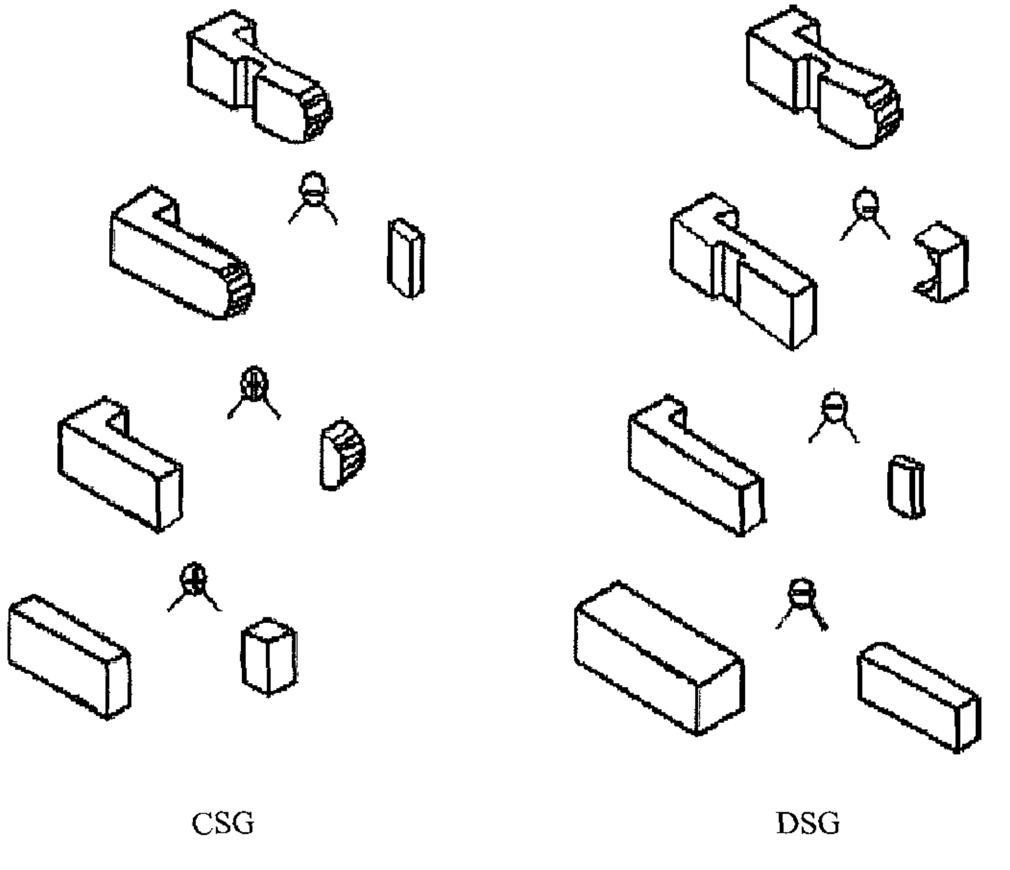


Fig. 2.11: Destructive solid geometry (CSG to DSG) [42].

The system developed by Li is limited to simple faceted primitives in which set-theoretic primitives yielded by the intersection operation are also excluded from their system. The feature primitives that the recogniser contains are limited as only 18 orbiter parts are represented. However, orbiter parts are the primitives that the feature recogniser uses to union with the model to create simple primitives [42]. In fig. 2.11, an orbiter part is used to transform the cylindrically ended block into a rectangular primitive.

2.5.2 Pattern Recognition

With the advent of powerful Computer-Aided Design tools, many manufacturing enterprises use computer software to design and model mechanical parts before production. The modern design phase starts with Computer-Aided Design (CAD) packages by producing a prototype design of solid mechanical parts. It is then used to evaluate whether the part under consideration is an existing design [43]. It works by using physical shape as a direct index to existing designs and manufactured components. This system eliminates time-consuming and error-prone searches of the taxonomy.

In order to produce a surface triangular mesh which represents the boundary of the object, this system uses a standard digital representation of the solid object. This system has significant applications in industries which seek to reuse existing designs and inventory, thereby reducing manufacturing costs.

2.6 Hint-Based Approaches

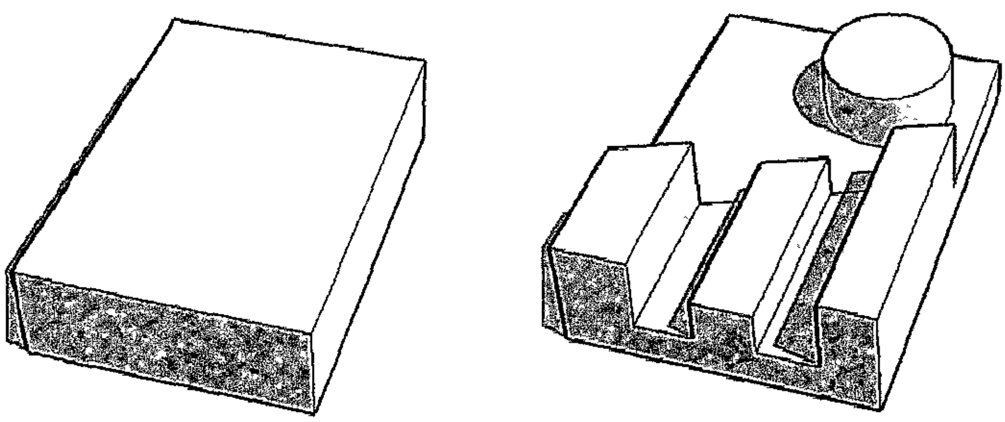
To avoid the intersecting features problem in faces, edges and vertices, Vandenbrande and Requicha [44, 45] proposed hint based reasoning, first implemented in OOFF (object-oriented feature finders) at USC. Similarly, this design is also implemented in F-Rex [46] at the University of Maryland, IF^2 (integrated incremental feature finder) at USC [47] and Feature-Based Machining Husk (FBMach) System at Allied Signal Aerospace, Federal Systems Division [48]. This section discusses the hint-based reasoning algorithms using the IF^2 example.

Design attributes such as normal geometries, design features and tolerances which are associated with the CAD model may be comprised of hints, for example a hole could be treated as a hint. However, other nongeometric varieties of manufacturing information such as design features, tolerances and design attributes are included by the extension of hint-based algorithms. The basic components of a hint-based feature recogniser have been described by Regli et al. [46, 49] as follows.

1. A set of feature types, $\mathcal{M}$
2. Each feature type M in $\mathcal{M}$ has associated with it a finite set of hint types h_{M1}, h_{M2}, …………, h_{Mk}
3. For each feature type M, there is a geometric completion procedure $\mathcal{P}_M$ which starts from the hint instances, performs extensive geometric reasoning, and finally constructs feature instances of type M.

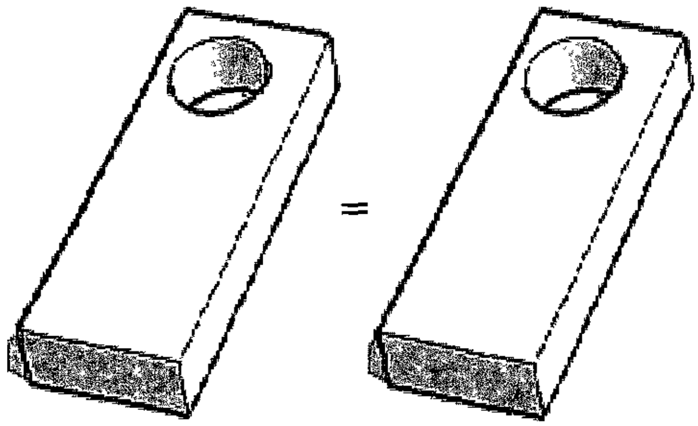
Holes, slots and pocket are recognised by the IF^2 method. In this example slot features have been discussed. A slot hint is generated from a nominal geometry when parallel opposing planar faces are encountered and is defined to be the wall faces of a slot. Fig. 2.12 (a) shows a slot feature represent by shaded faces. More traces are found in hint-based approaches which creates a problem when recognising good features. A trace or hint is simply an implication for the possible existence of a feature, and therefore a significant number of traces may not lead to valid features.

A generate-test-repair prototype is followed by the geometric completion procedures of IF^2 [50], the first step of which is to find the slot floor. The part faces that are planar and perpendicular to the wall faces are taken as floor candidates which is conceived from the space between the wall faces. An example of several floor candidates and the heavily shaded face is shown in fig. 2.12 (a).

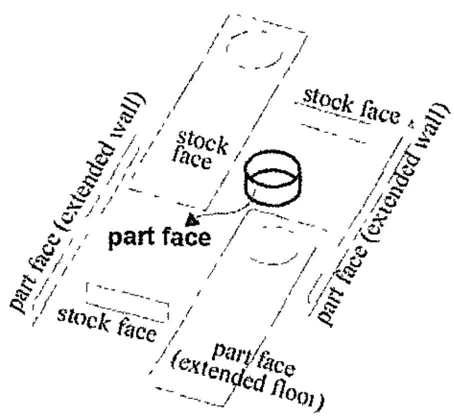


Stock   Part

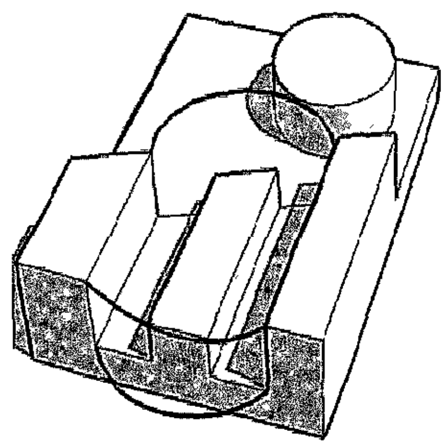
(a) Input stock and Part



(b) Proposed removal volume



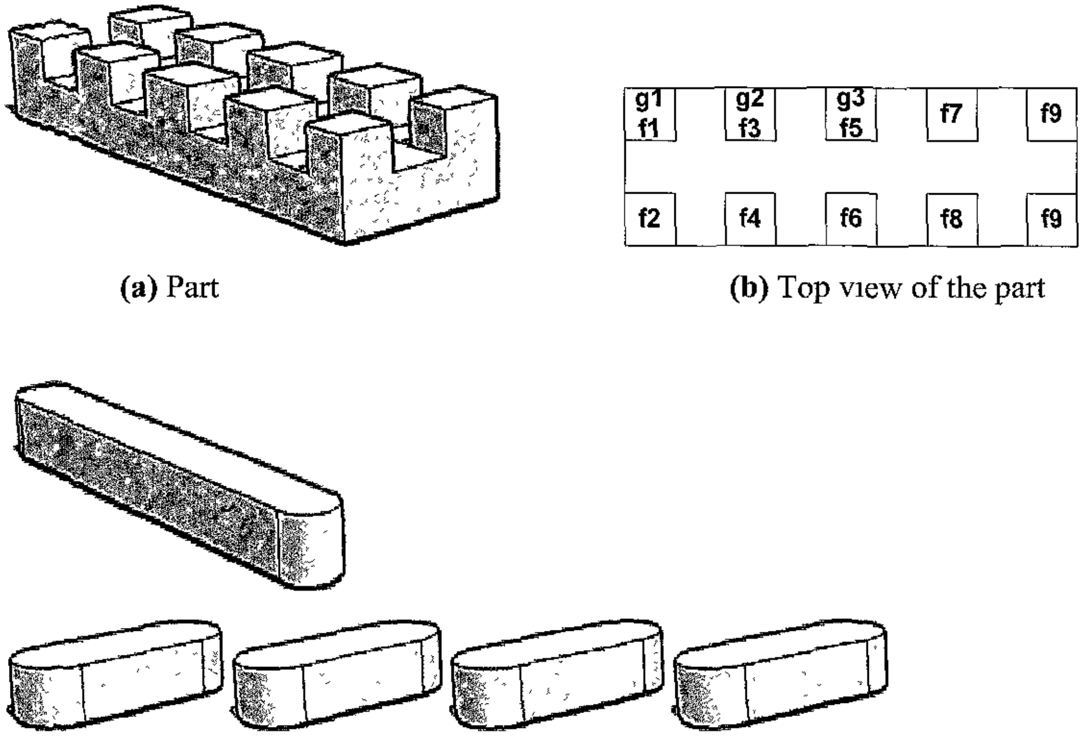
(c) Boundary analysis



(d) Recognized slot

Fig. 2.12: Slot recognition in IF^2

Fig. 2.12 (b), shows the portion of the delta volume between the walls and above the floor which is proposed as a volume to be removed by a slot machining operation. Stock faces are those to be removed by feature machining operations and part faces are those to be created by feature machining operations. If a slot boundary contains any part faces besides the walls and floor the proposed removal volume is not machinable as a whole. In fig. 2.12 (c), the cylindrical face portrayed in bold lines is such a part face. However, if the test step determines that the volume proposed by the generate step is not machinable as a whole, the repair step tries to instantiate a feature volume which is maximally extended but removes a subset of the proposed removal volume, such that the machining operation does not intrude into the part face. This is a geometric fitting problem, and in fig. 2.12 (d), it is shown that the example IF^2 finally produces a parameterised slot volume.



(a) Part **(b)** Top view of the part

(c) Desirable features: 5 slots

Fig. 2.13: An example of part with many extra traces.

It is inefficient to perform expensive geometric reasoning on every trace even though the number of traces is bounded by a polynomial. In fig. 2.13, five slot traces, such as (f1, f2), (f3, f4), (f5, f6), (f7, f8) and (f9, f10), lead to the same slot, the long slot, which is shown in fig. 2.13 (c).

In order to reflect the new features, which will influence other traces, the priority queue is updated. For example in fig. 2.13, once a slot is recognised from (f1, f2), the strength of (f3, f4), (f5, f6), (f7, f8), and (f9, f10) are reduced such that they attract less attention, as they would lead to redundant slots.

IF^2 updates the material to be removed by subtracting the new feature volume from it and checks for a null solid after updating the priority queue; this is called termination test. Initially, the material to be removed equals the delta volume. When the result is null the delta volume is fully decomposed and the process stops. On the other hand, IF^2 takes the new top-ranked trace and repeats the same process.

By focusing on promising traces IF^2 avoids unnecessary reasoning as much as possible and also tries to produce a desirable interpretation (machining feature model). The current implementation of IF^2 generates the interpretation of a single pocket, but success is not guaranteed. However, IF^2 shows an effort for handling the problems of completeness and multiple interpretations.

For a robust library of machining features and feature recognition algorithms the Feature-Based Machining Husk (FBMach) is very useful. It uses three different approaches to define surface features: Automatic recognition, Interactive recognition and Manual identification. A procedural algorithm is used in automatic recognition to search for feature hints and then create feature instances using hints without user interaction. In generating the feature instances the interactive recognition allows the user to provide some hints for FBMach to use in generating the feature instances. However, the manual identification allows the user to create a feature instance by adding each face to the feature individually and defining each face's role in the feature. A human-supervised reasoning approach implemented by FBMach has also been explored by Van Houten [51].

The University of Maryland's IMACS system (interactive manufacturability analysis and critiquing system) uses the F-Rex for their feature recognition component [49, 52-56]. Many important issues in feature recognition such as manufacturing process specific features, recognition of alternative features, multiprocessor techniques, incorporation of manufacturing resource constraints etc. are formally addressed by IMacs/ F-Rex.

The main problem for hint-based approaches results from there being more traces than there are good features to recognise. In order to overcome this problem two methods are proposed by Han, one is to generate a sub-optimal interpretation and allow users to demand alternatives [57] and the other is to pursue an optimal interpretation by incorporating some manufacturing knowledge into the process of feature recognition [58].

2.7 Automatic Feature Recognition (AFR)

AFR methods apply knowledge acquisition techniques for generating feature recognition rules and feature hints automatically which is a major advantage in comparison with other rule-based and hint-based FR methods. To construct valid features from the geometrical and topological information stored in B-Rep part models, a set of rules and two geometric reasoning algorithms are employed by the feature recognition process.

AFR techniques are an important tool for achieving a true integration of design and manufacturing stages during product development. The realisation of a true integration between the product and process design stages is a challenging goal and it requires a consistent utilisation of product information at different levels of abstraction [59]. AFR techniques are applied to identify geometrical entities, features in the CAD model which are semantically significant in the context of specific downstream manufacturing activities in order to bridge the information gap between CAD and CAM.

To develop more flexible AFR systems, their knowledge bases should be easily adaptable to changes in the application area and also extendable to cover other applications. However, the objective of this system is to develop a feature recognition method that employs knowledge acquisition techniques. In order to achieve this, a new AFR method that combines the 'learning from examples' concept with the rule-based and hint-based feature recognition approaches is proposed which contains two main processing stages: learning and feature recognition.

Rules and feature hints are extracted from training data during the learning stage and then these hints and rule bases are utilised in the feature recognition stage to analyse B-Rep part models and identify their feature-based internal structure.

2.8 Hybrid Approaches

Gao and Shah, in their work [60], established a recent example of combining some characteristics of existing approaches to feature recognition which is an extended version of the part graph discussed earlier where, for example, each face node is classified into either a stock face or a part face. A set of subgraphs called minimal condition subgraphs (MCSGs) is made by the repeated decomposition of the input graph. A MCSG is a subgraph of a specific feature's template graph which remains in the part graph. Finally each MCSG is completed to produce a feature. However, face nodes are dynamically split and missing arcs are added through extensive geometric reasoning by both generating and completing MCSGs.

Therefore, Gao and Shah claimed that their approach is a combination of the conventional graph-based approach and the hint-based approach in which a hint is defined as a minimal piece of information indicating potential existence of a feature. The concept of using alternative interpretations on demand is followed in Gao and Shah's work, which proposed hint-based approaches.

The Graph-based approach, the volumetric decomposition approach and the hint-based approach are unique techniques for feature recognition and therefore it is difficult for an approach to take some algorithms from more than one approach. However, Gao and Shah [60] in their work, proved that it would be constructive to take some of the fundamental concepts from each approach.

2.9 Summary of This Chapter

A wide discussion of different feature recognition methods has been presented in this chapter. This discussion details the necessity of feature recognition; it is important to report the existence of a feature in the part including its attributes and relationships. However, in order to find the features, feature recognition methods play a vital role. Once the features have been identified it is then easy to find the appropriate manufacturing process with their specifications. Volume decomposition approaches that have been used for feature recognition (such as hole, slot, pocket, boss, and step) are shown in chapter 3.

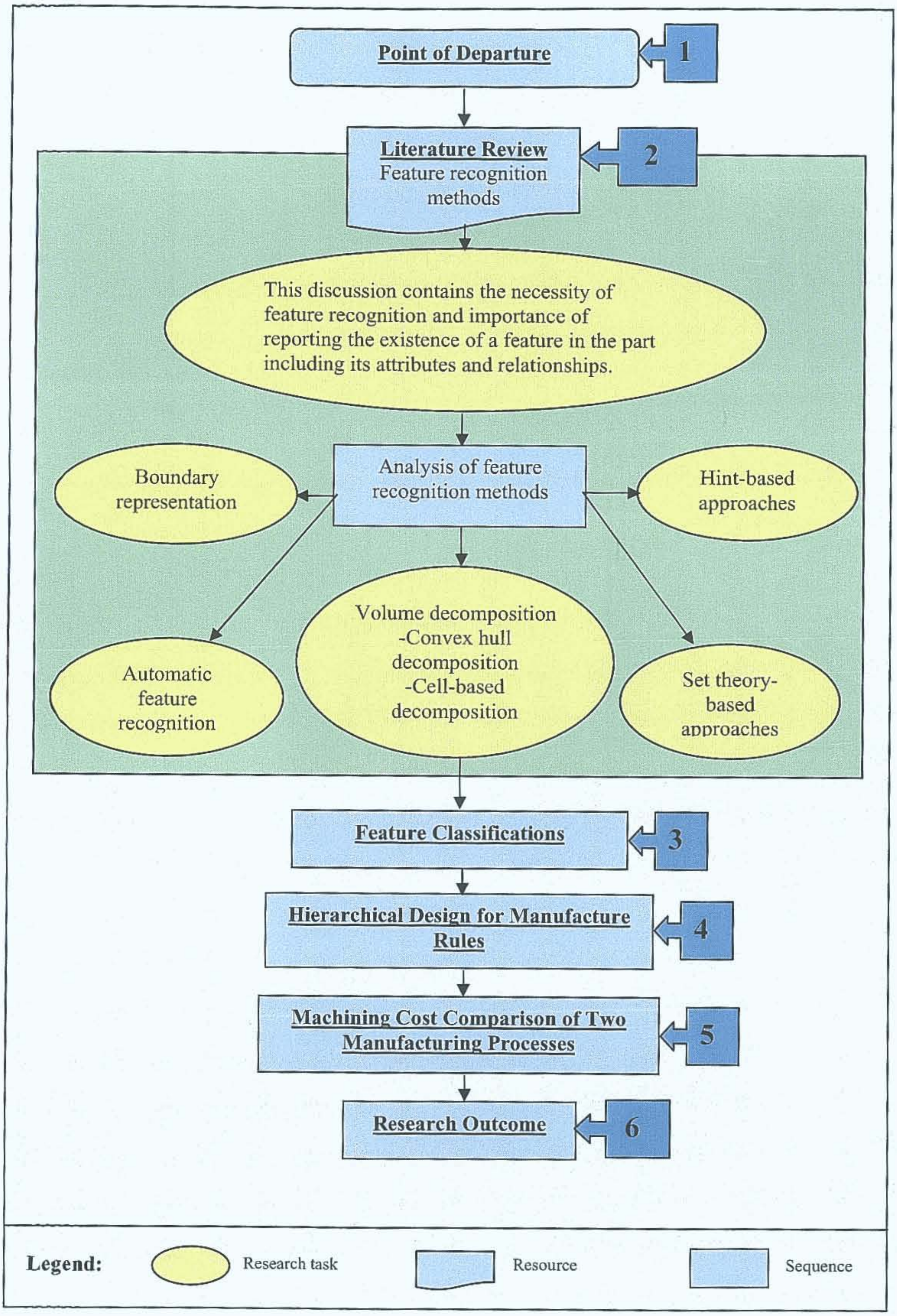


Fig. 2.14: Summary of the research process in chapter two.

CHAPTER 3 FEATURE CLASSIFICATIONS

3.1 Introduction

Feature technology is a flourishing subject, with research being carried out worldwide in many academic and industrial establishments. Many existing and potential applications, particularly in the field of manufacturing, need various aspects of features technology. In all engineering fields geometric modelling has become accepted for a wide range of applications which accurately represent the shape of a whole engineering component. For most applications information about the shape of the different parts is needed in order to know where the slots and holes in the component are. From this it is easier to know where the projecting lugs are and so on; these are called features and the mathematical and computational techniques for dealing with them make up the subject of feature technology.

However, for manufacture, feature information can be considered to be about volumes of material to be removed or to be added, depending upon the manufacturing process being considered. The features can be associated with manufacturing operations and machine cutters [13,61]. Two examples can be considered: simple planar slots which are considered as machine operations, and T-slots, can be considered as special-cutter operations.

3.2 Design features

Design features which are viewed from the designing stand point, present only topological and geometrical information. Features used at the design stage, defined by the user or from the CAD modeller library, are called design features. They do not take into consideration any manufacturing, assembly or inspection constraints. There are three types of design features: depression, protrusion, and transition. A boss feature is the depression feature as an increment of the shape. A hole feature is the protrusion feature as a decrement of the shape. Depending upon the profile whether it is convex or concave a transition feature could be either a decrement or an increment. Slot, hole, pocket, rounding, cylinder, block, protrusion, cut, chamfer, user defined features are examples of design features [62-64]. The CAD system ensures that the underlying geometry remains consistent with the functionality of the feature.

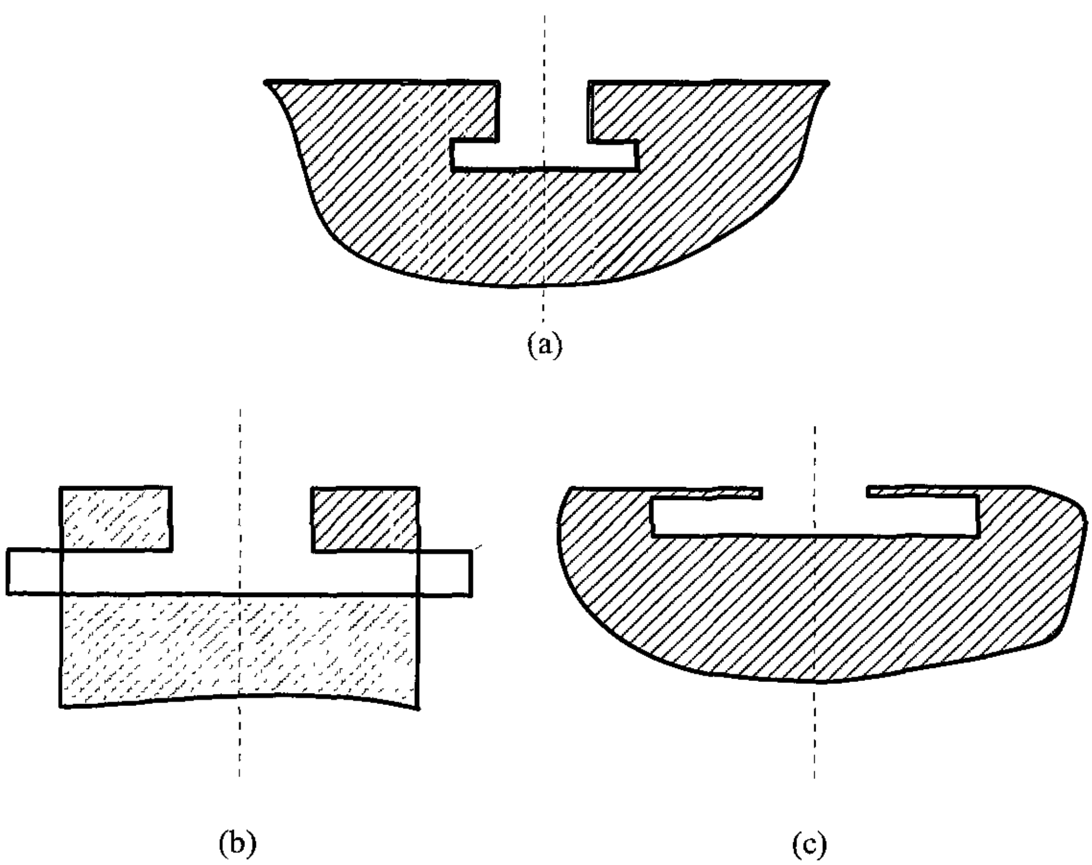


Fig. 3.1: **(a)** T-slot design feature **(b)** Impossible geometry **(c)** Element with low strength.

Besides geometric information design features can contain tolerance, roughness and other information. Design features are related to existing surfaces/ features. Fig. 3.1(a) indicates the correct T-slot design and (b) and (c) indicate incorrect T-slot designs.

As an example of a design feature, a through hole can be considered. The bottom of the hole is related to the bottom surface of the box. If the dimensions of the box are changed, the CAD system automatically adjusts the hole so that it remains through; this is shown in fig. 3.2.

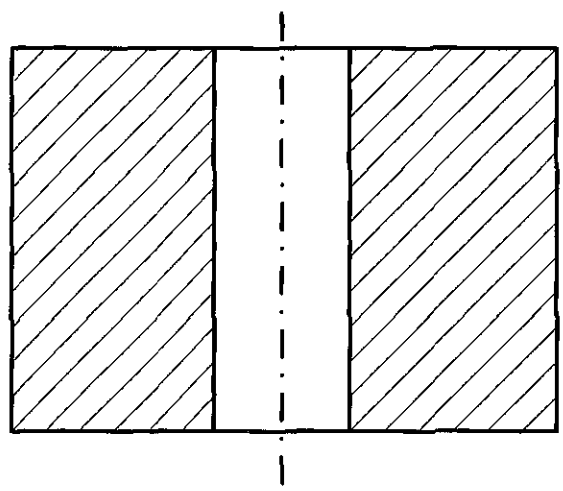


Fig. 3.2: Through hole design feature.

3.3 Form features

Form features can be classified into two categories based on the attributes of the geometric and topological entities: interior form features and exterior form features [65]. Interior form features can be classified into two types, concave features and convex features. Pocket and Hole features are considered to be concave features. Similarly, boss features are considered to be convex features. Two types of exterior form feature, a slot feature and a step feature are shown in fig. 3.3.

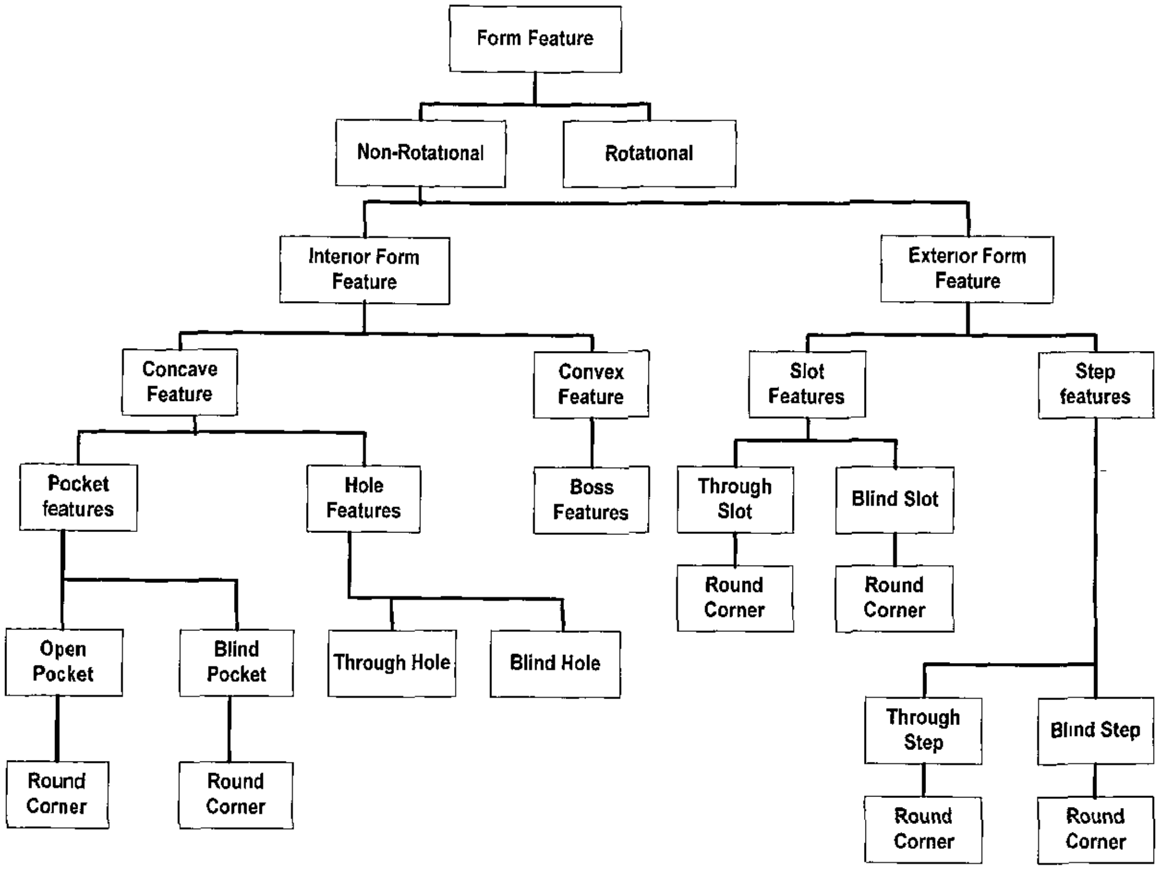


Fig. 3.3: Hierarchical classification of Non-Rotational Form features.

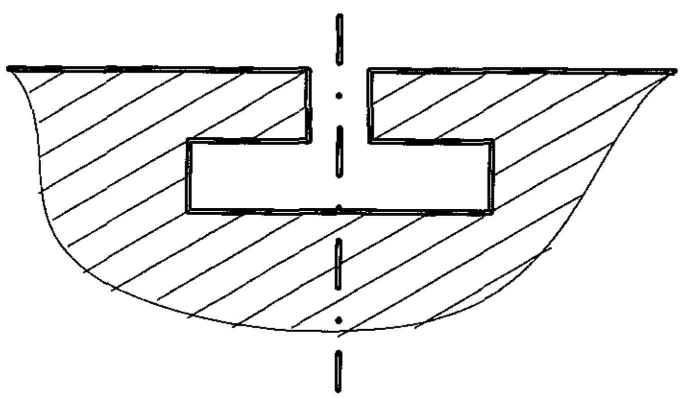
3.4 Manufacturing Features

Manufacturing features which are viewed from the manufacturing stand point present topological, geometrical and manufacturing information. A manufacturing feature is typically defined as a collection of related geometric elements which as a whole correspond to a particular manufacturing method or process or can be used to determine the suitable manufacturing methods or processes for creating the geometry [66]. A manufacturing feature is a feature which is interpreted as a continuous volume that can be removed by a single machining operation in a single set-up [67]. It depends on both the shape and size of the geometric feature and manufacturing processes to be used to produce this feature [68]. It can be concluded that a manufacturing feature is the function of machine tools, set-up, tools and parts. Hole, pocket, open pocket, face, boss, step, open step, slot, notch, grove, knurl, thread, fillet, chamfer, etc are the examples of manufacturing features that can be found [63, 69, 70].

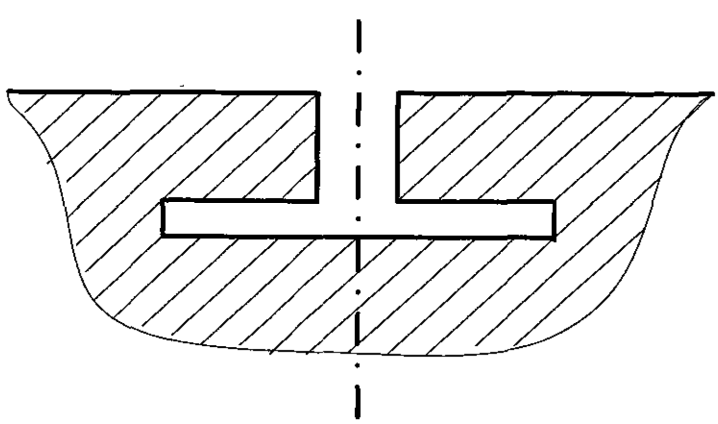
3.5 Machining Features

Machining features include the characteristics of the design features (geometry, tolerance, roughness) and contain additional manufacturing related information (machine tool, cutting tool, cutting conditions, fixturing, relative machining price information and others). A feature-based CAD system should ensure that the manufacturing feature remains consistent with the underlying design features.

An example of a machining feature is a T-slot connected with the machining process end milling (with T-mill) which is shown in fig. 3.4 (a) and (b).



(a)



(b)

Fig. 3.4: Rejected T-slot machining feature. (a) Too small a tool shank diameter, (b) Too narrow a cutter.

If the designer reduces the opening width of the slot below the manufacturability limit (narrow opening results in a small mill shank diameter, the cutting tool can not withstand the cutting force), the CAD system will refuse that type of design. Similarly a narrow slot will be rejected as it requires a thin cutter with large diameter which is not recommended for machining.

Another example of a drilled hole machining feature can be considered. Since interrupted holes as shown in fig. 3.5 are not recommended from a machining point of view, the system may reject it as machining feature even though it is acceptable as a design feature. These features do not contradict the design but rather machining principles.

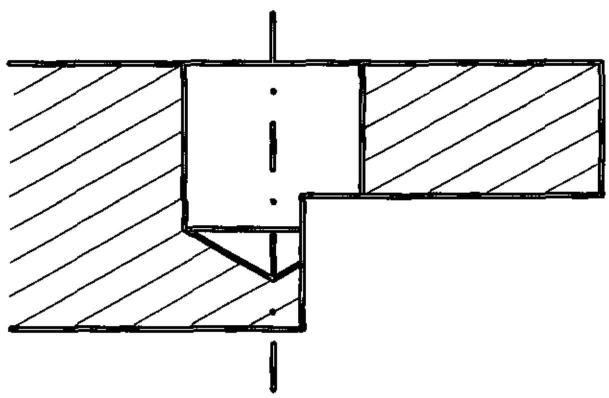


Fig. 3.5: Rejected drilled hole machining feature.

3.6 Hole Features

Depending upon the manufacturing processes, a hole feature can be classified into two types such as Material Removal and Material Transformation. Material Removal relates to the amount of material that can be removed by machining processes and Material Transformation relates to the processes through which material can be transformed to produce the desired shape. Hole features can be classified into three main categories, Machined holes, Cast holes and Formed holes. After that every main category can be classified into two sub-categories, through holes and blind holes. Every sub-category contains different hole features with their manufacturing processes. Fig. 3.6 to fig. 3.9 show the classification of different hole features based on the examples as indicated in fig. 3.10 and fig. 3.11.

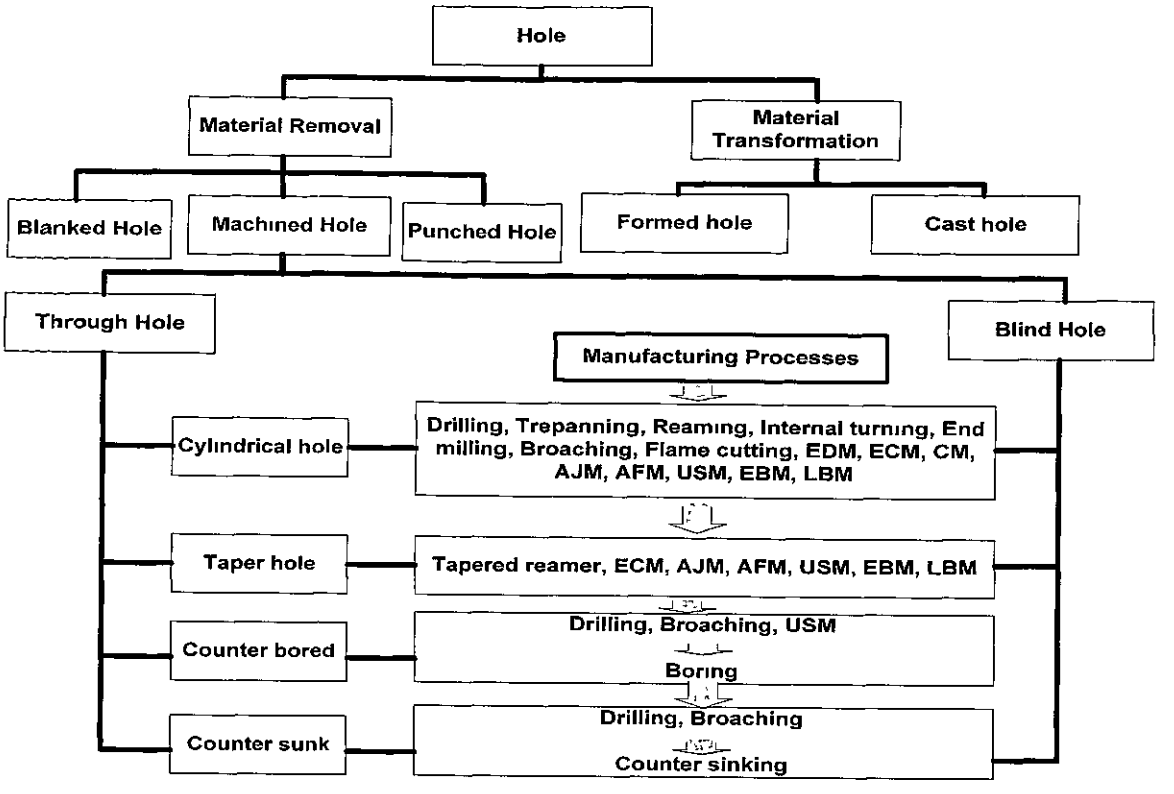


Fig. 3.6: Through and Blind hole classification of Machined holes.

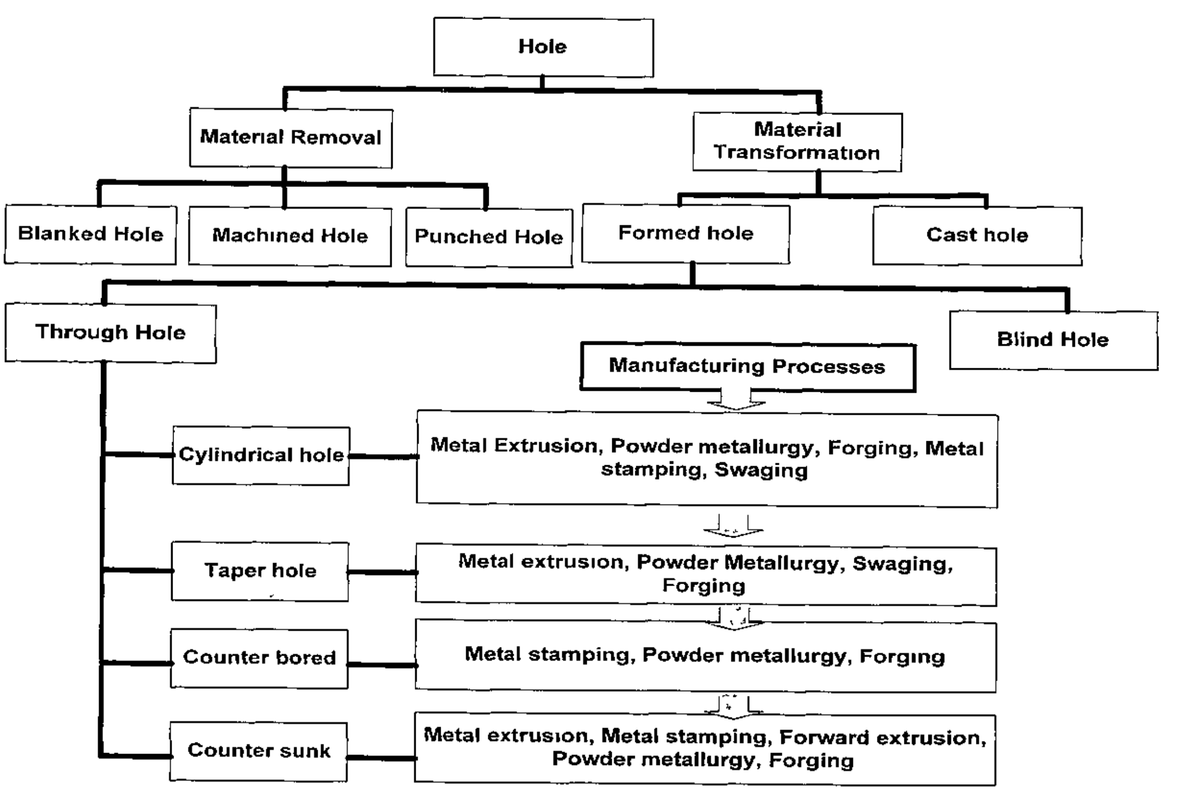


Fig. 3.7: Through hole classification of Formed holes.

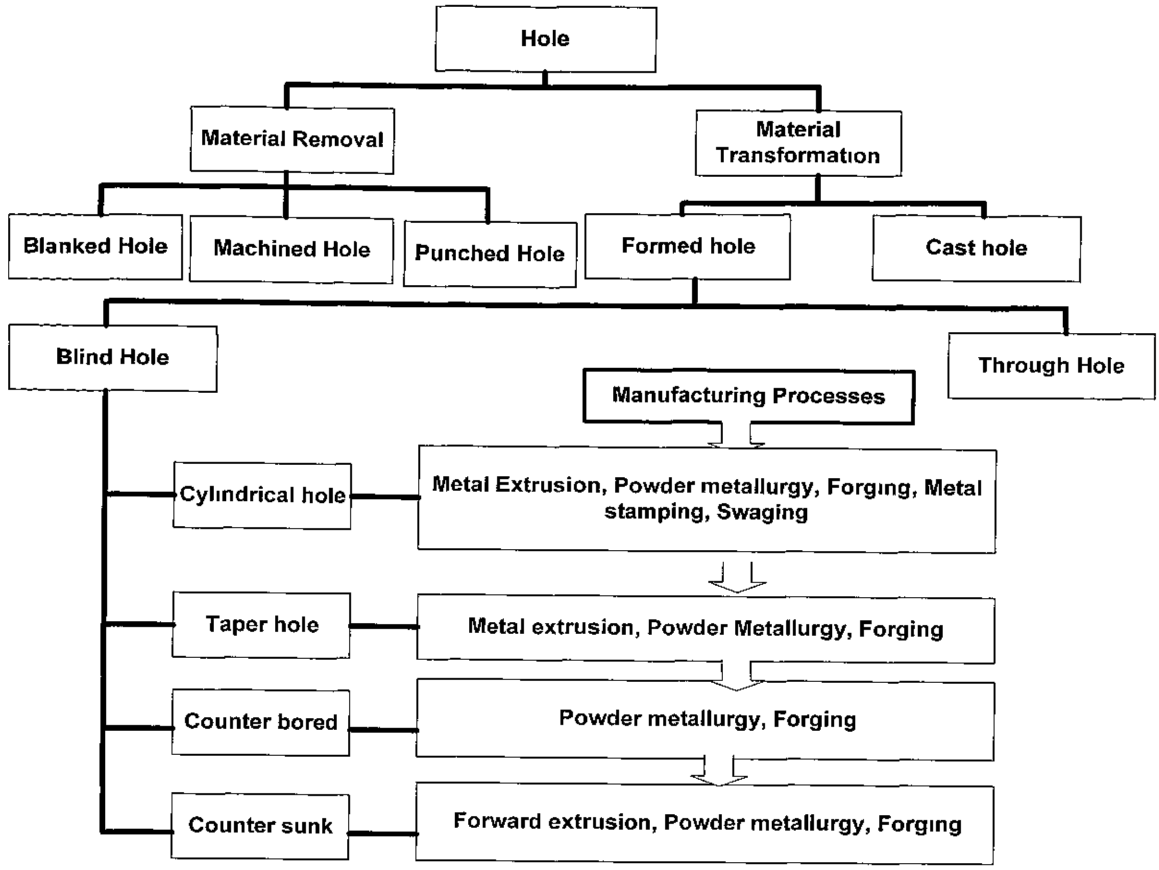


Fig: 3.8: Blind hole classification of Formed holes.

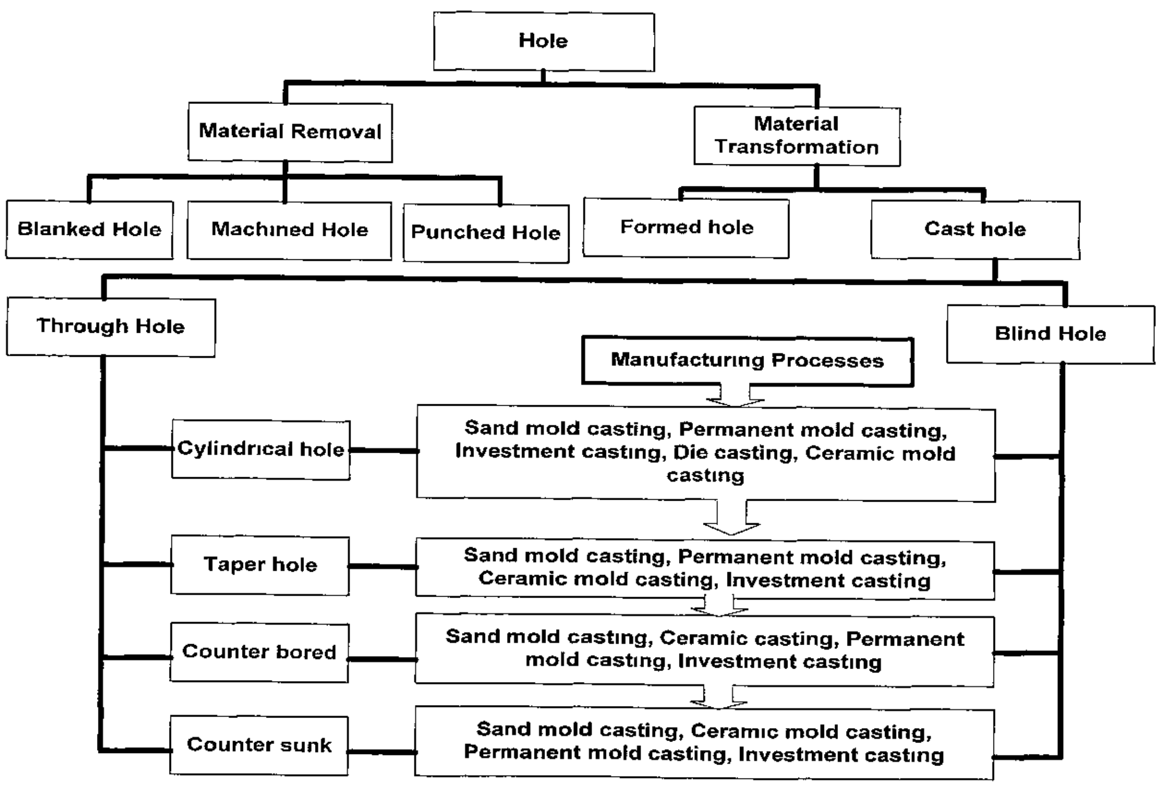
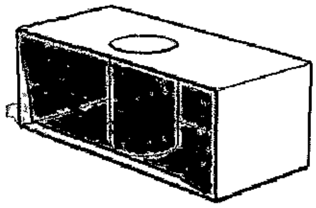
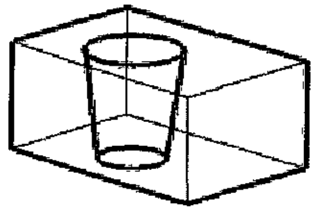
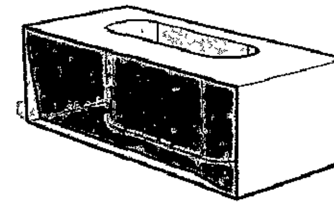
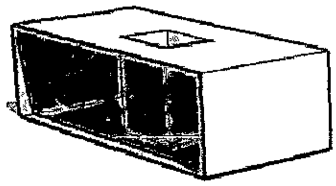
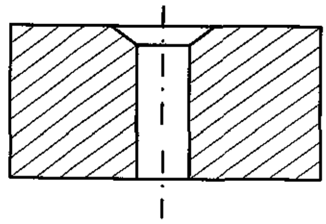
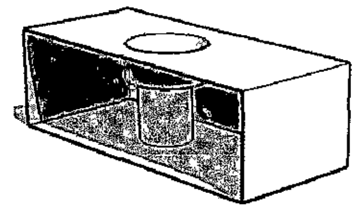


Fig. 3.9: Through and Blind hole classification of Cast holes.

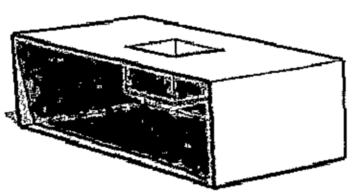
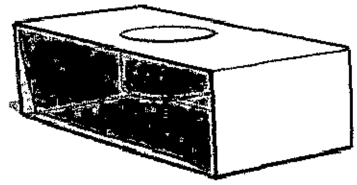
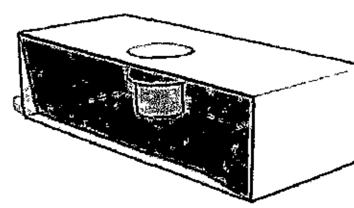




(a) Cylindrical hole **(b)**Taper hole **(c)** Free form non-rotational hole

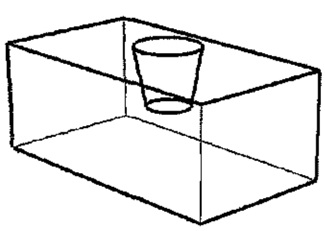
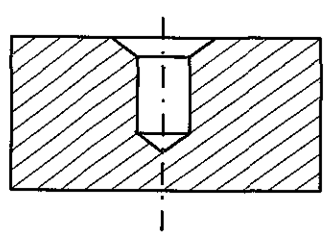
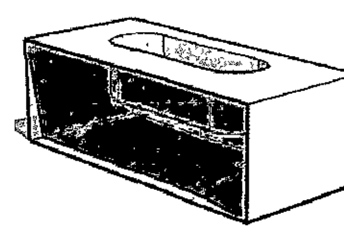




(d) Multi side non-rotational hole **(e)** Countersunk hole **(f)** Counterbored hole

Fig. 3.10: Through hole features.

(a) Multi side non-rotational hole **(b)** Cylindrical hole **(c)** Counterbored hole

(e)Taper hole **(f)** Countersunk hole **(g)** Free form non-rotational hole

Fig. 3.11: Blind hole features.

3.7 Slot features

Slot features can be categorised into three types, Machined slots, Formed slots and Cast slots. Machined slots are part of a Material removal operation. On the other hand, Cast slots and Formed slots are the result of Material transformation operations. Each type of slot can be classified into two main categories, Through slots and Blind slots. Each category of slot features can be further classified into different slot features with their possible manufacturing processes. Fig. 3.12 to fig. 3.17 explain the slot feature classification based on the example as indicated in fig. 3.18 and fig. 3.19.

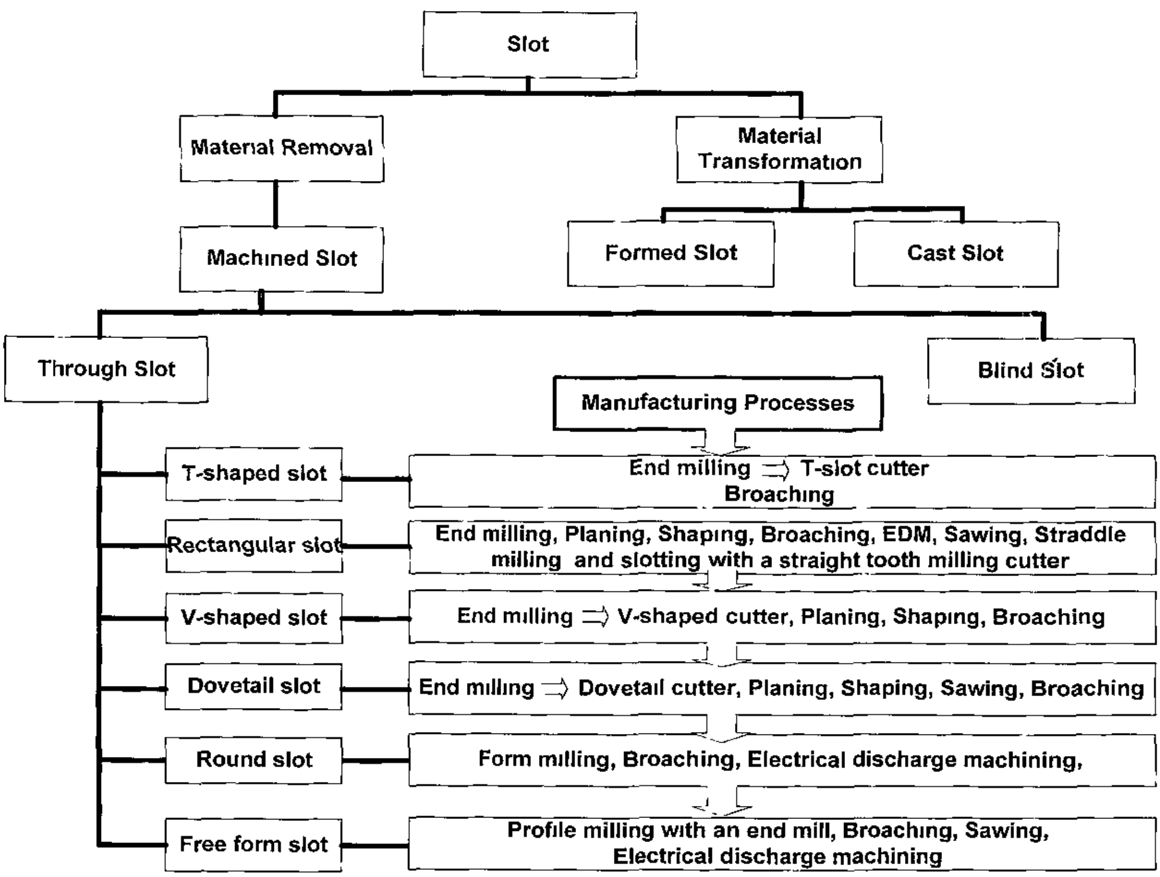


Fig. 3.12: Through slot classification of Machined slots.

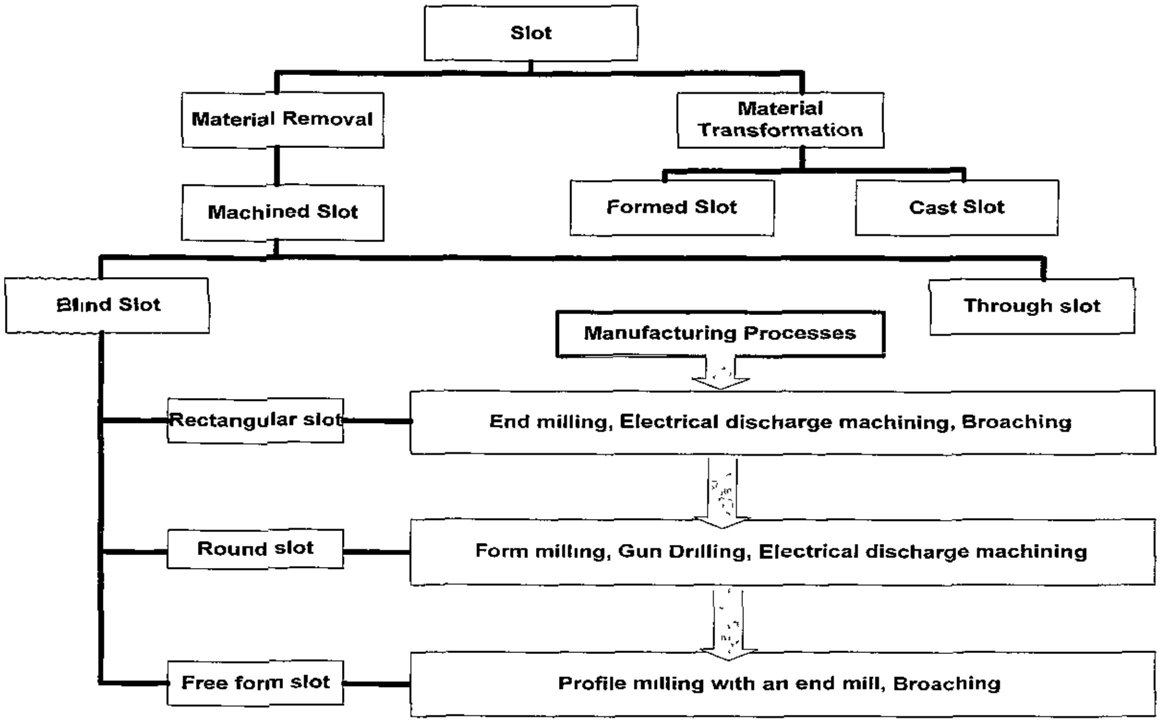


Fig.3.13: Blind slot classification of Machined slots.

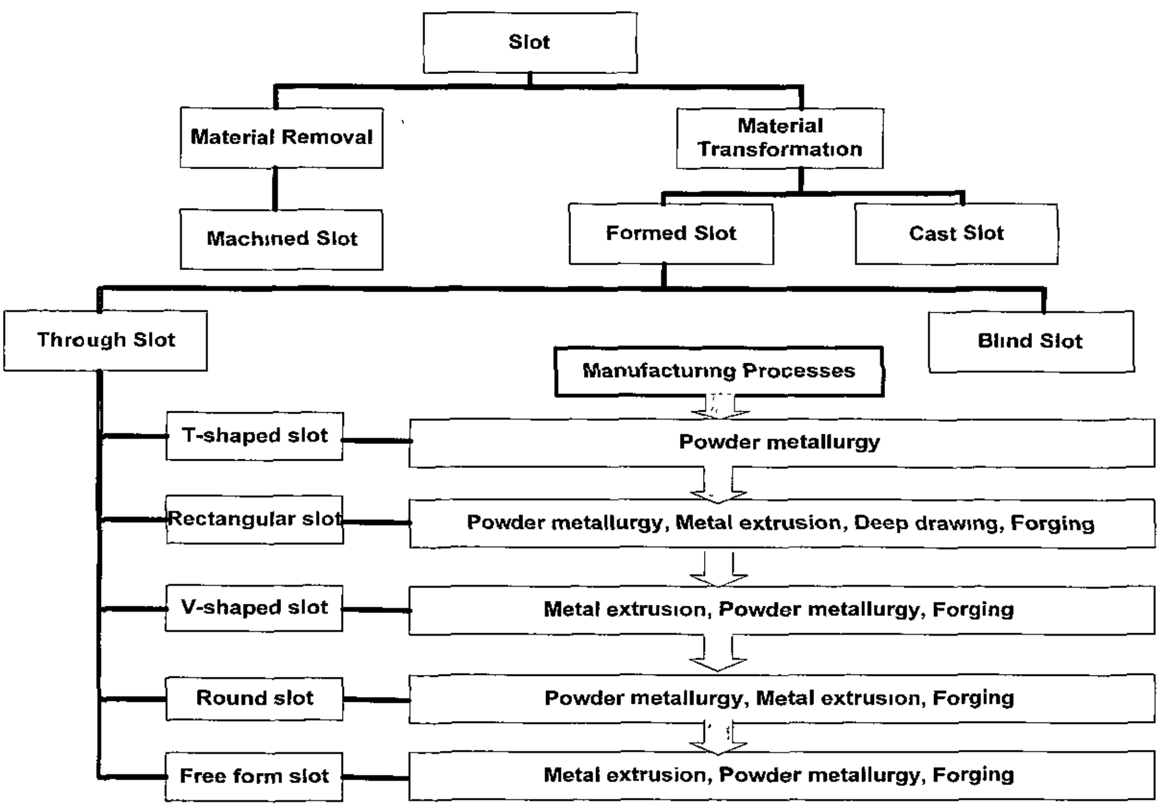


Fig. 3.14: Through slot classification Formed slots.

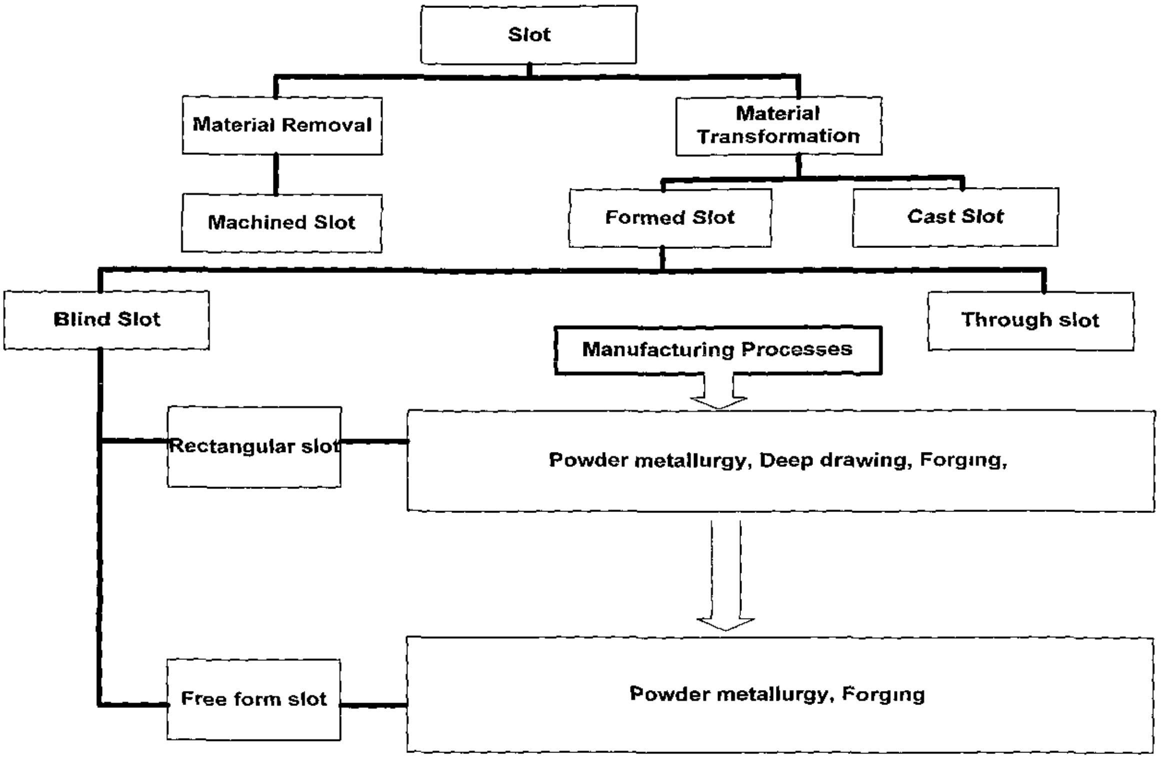


Fig.3.15: Blind slot classification of Formed slots.

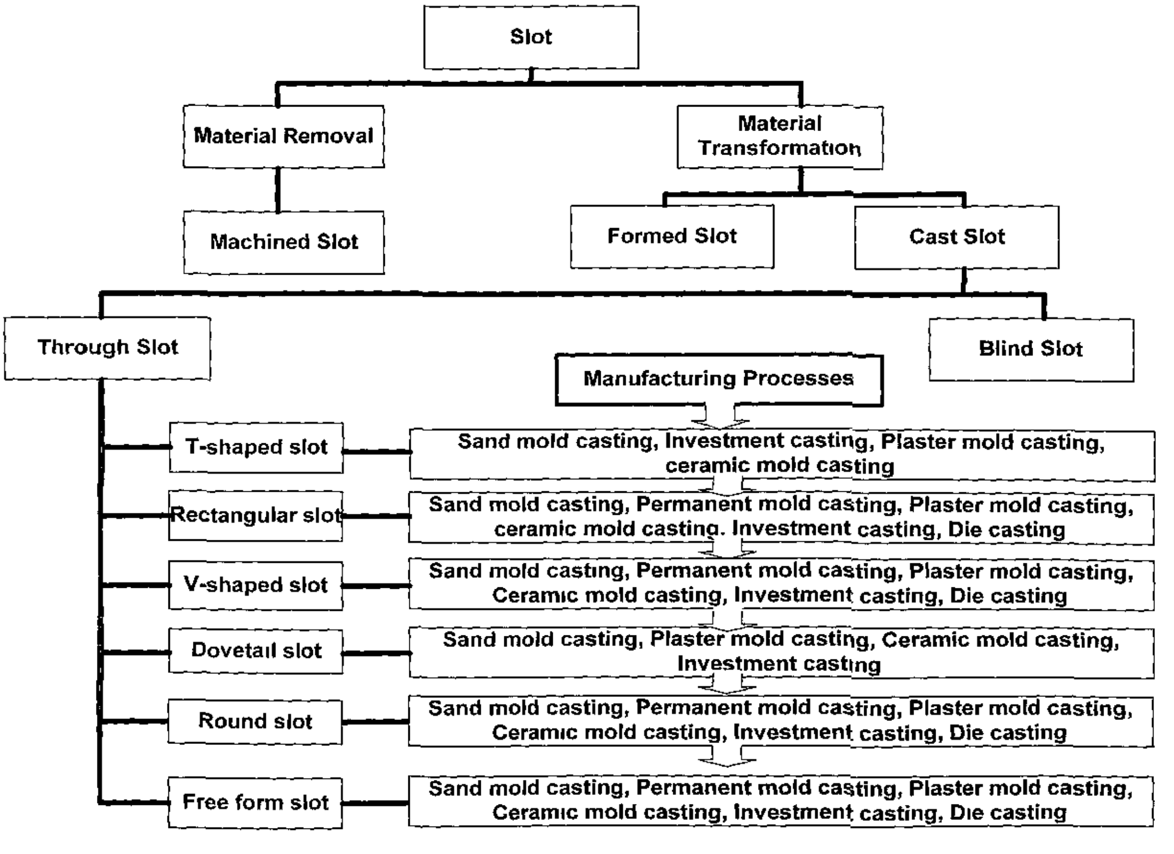


Fig. 3.16: Through slot classification Cast slots.

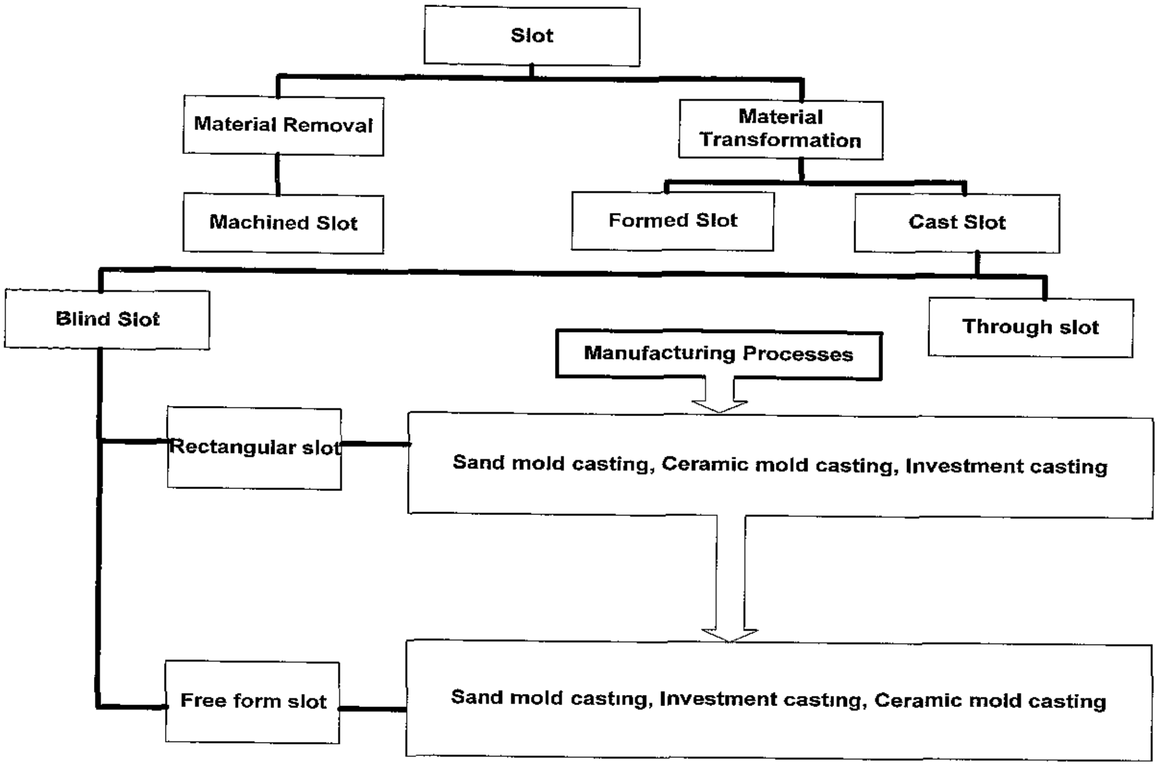
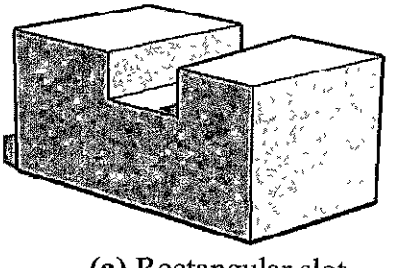
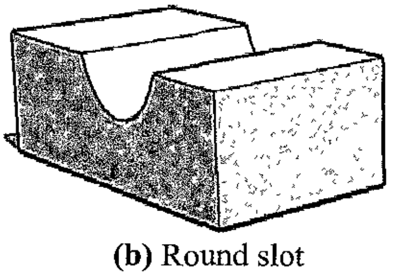


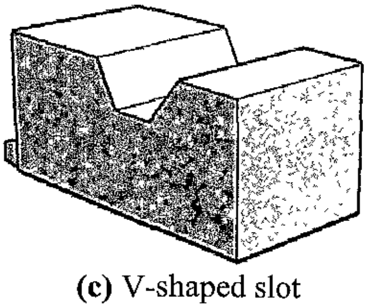
Fig. 3.17: Blind slot classification of Cast slots.



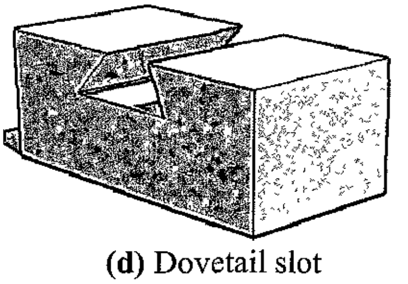
(a) Rectangular slot



(b) Round slot



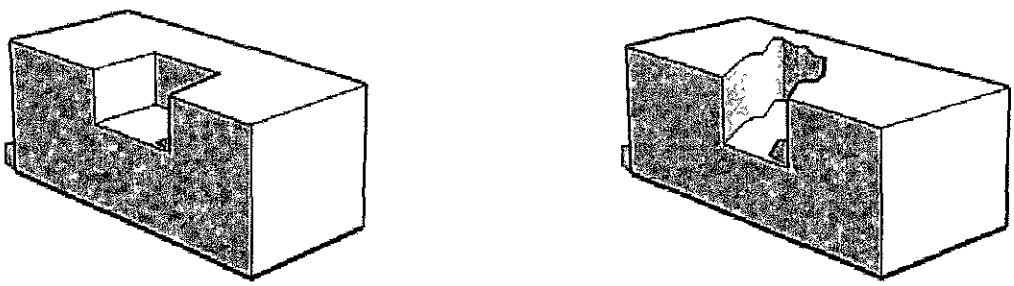
(c) V-shaped slot



(d) Dovetail slot

(e) T-slot **(f)** Free form slot

Fig. 3.18: Through slot features.



(a) Rectangular slot **(b)** Free form slot

Fig. 3.19: Blind slot features.

3.8 Pocket Features

Another approach for feature classification is in terms of Pocket features. Pocket features are the features which are classified into Machined pockets, Formed pockets and Cast pockets. A Machined pocket is a pocket which is manufactured by a machining process, a Formed pocket is manufactured by formation of metal, and a Cast pocket is manufactured by a casting process. Pocket features can be firstly classified into two categories, Open pockets and Blind pockets. Following this, Open pockets can be classified into three categories; Rectangular pockets with rounded end, Square pockets with rounded end and free-form pockets. Blind pockets can be classified in the same way. Fig. 3.20 to fig. 3.25 describes the classification of pocket features based on the example as indicated in fig. 3.26 and fig. 3.27.

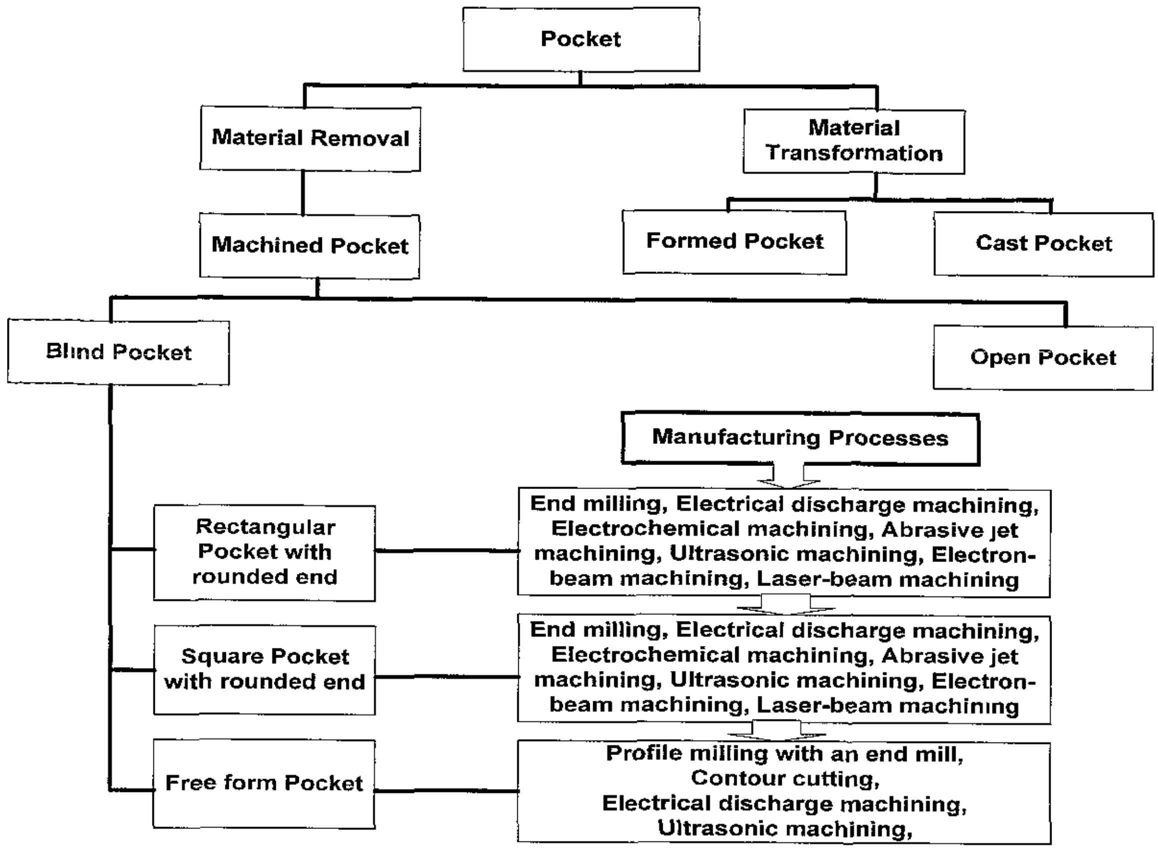


Fig. 3.20: Blind pocket classification of Machined pockets.

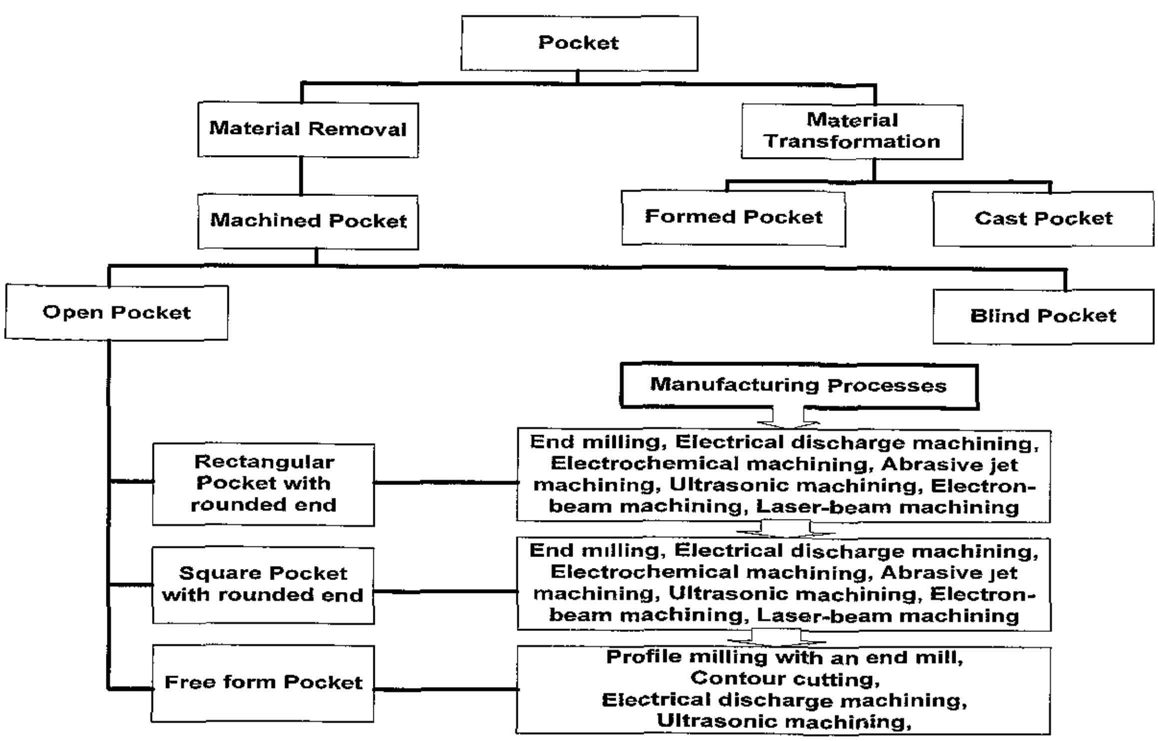


Fig. 3.21: Open pocket classification of Machined pockets.

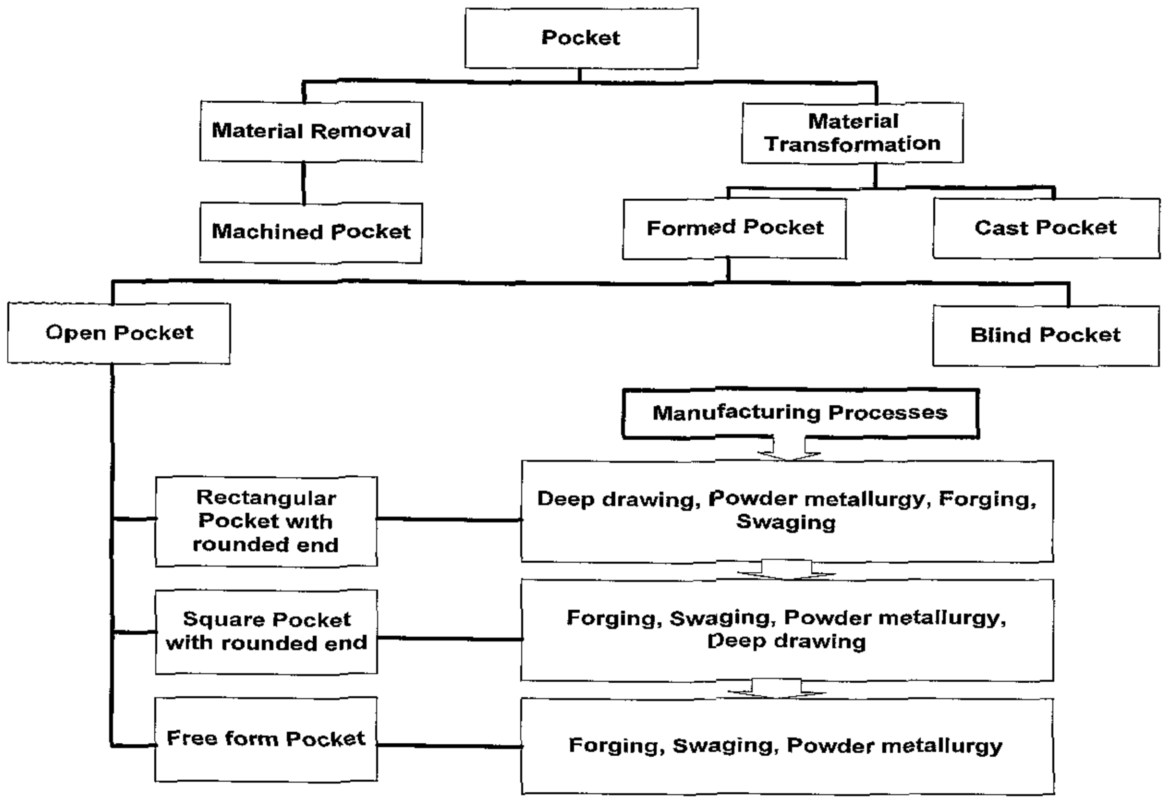


Fig. 3.22: Open pocket classification of Formed pockets.

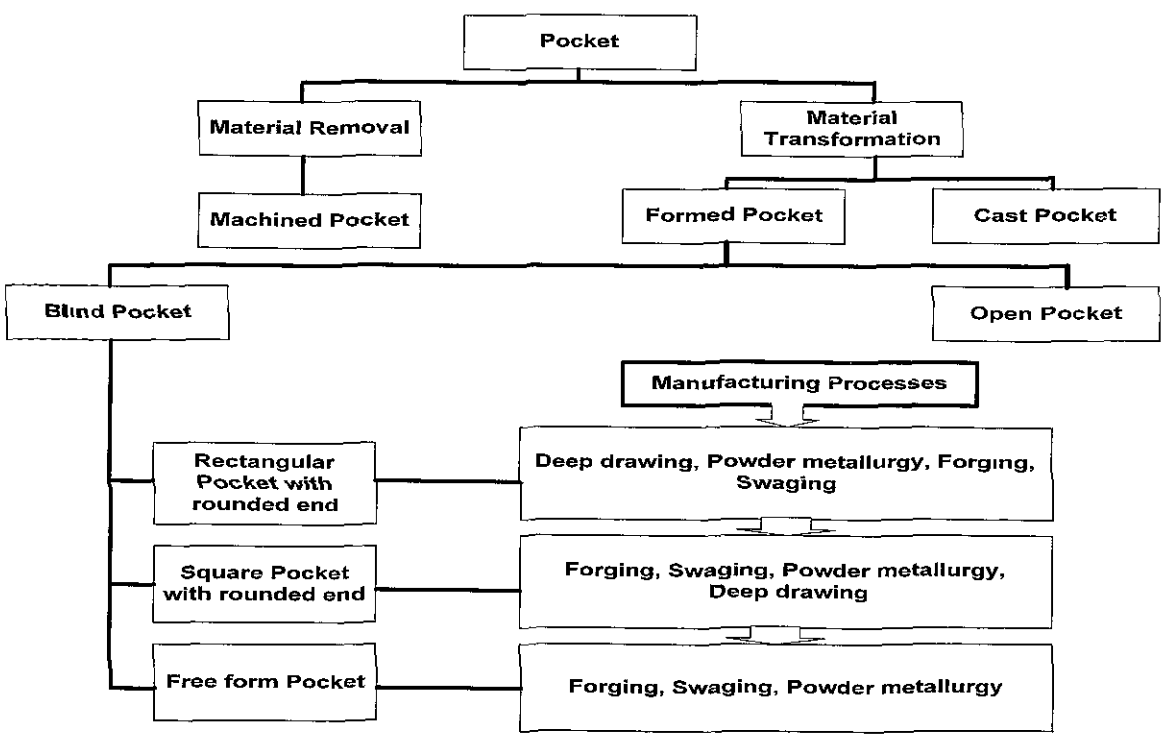


Fig. 3.23: Blind pocket classification of Formed pockets.

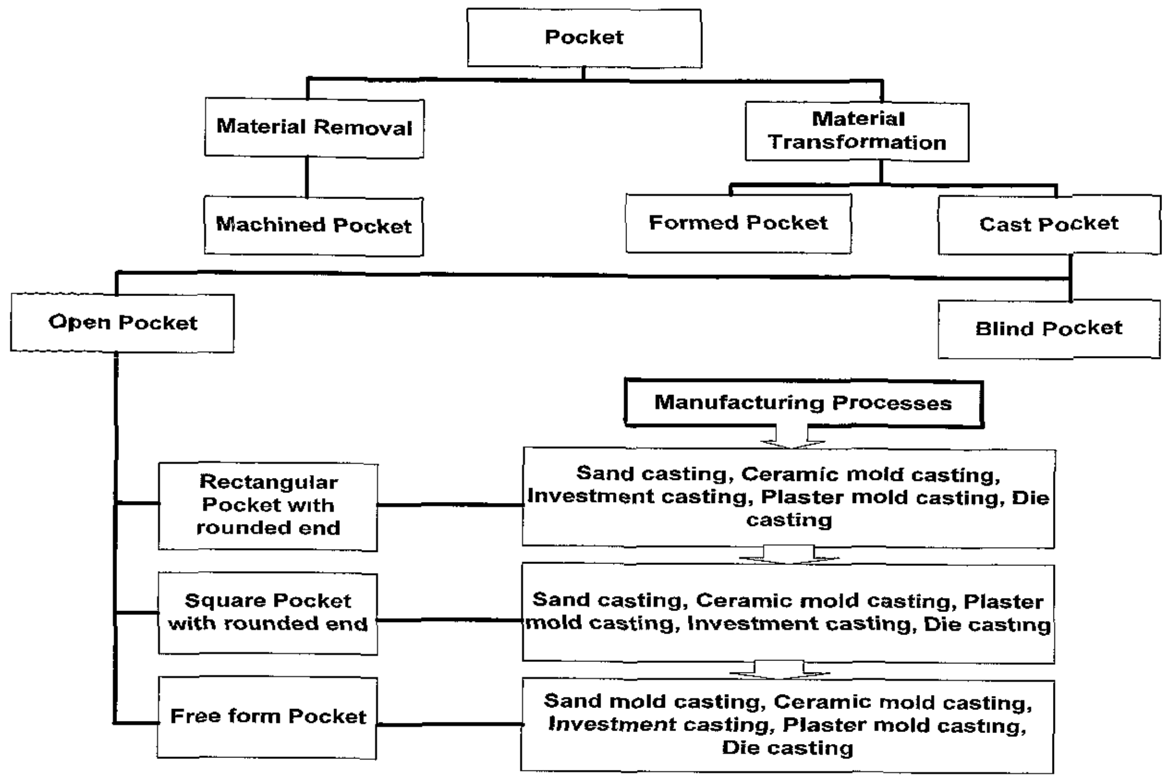


Fig. 3.24: Open pocket classification of Cast pockets.

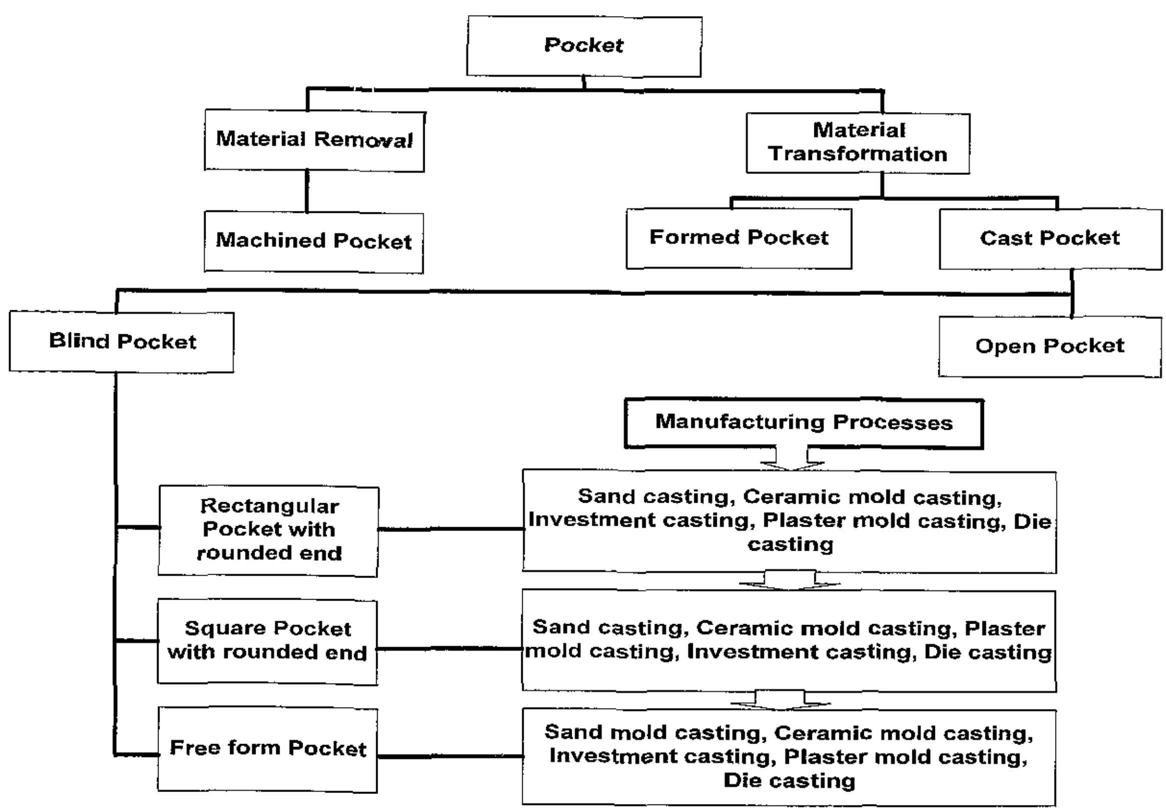
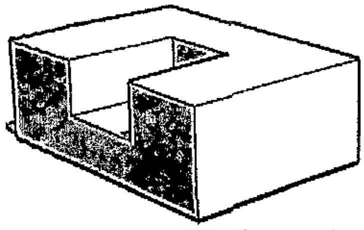
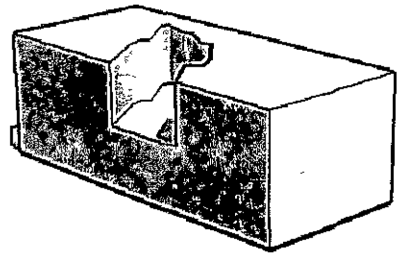


Fig. 3.25: Blind pocket classification of Cast pockets.

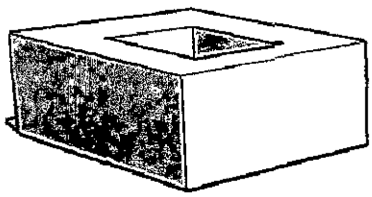


(a) Rectangular pocket

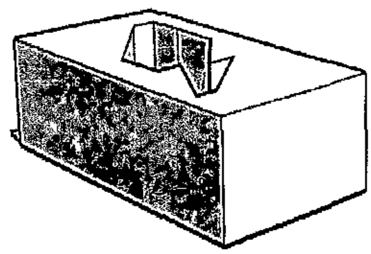


(b) Free form pocket

Fig. 3.26: Open pocket features.



(a) Rectangular pocket



(b) Free form pocket

Fig. 3.27: Blind pocket features.

3.9 Boss Features

Boss features can be first classified into three types depending upon their manufacturing process; Machined boss, Formed boss and Cast boss. Like some other features, Boss features have only Through features as they are convex form features. Therefore, the three main categories can be classified into one type which is the Through boss type. Through boss can be classified into four types, Circular boss, Rectangular boss, Dovetail boss and Free Form boss. Fig. 3.28 to fig. 3.30 show the classification of Boss features based on the example as indicated in fig. 3.31.

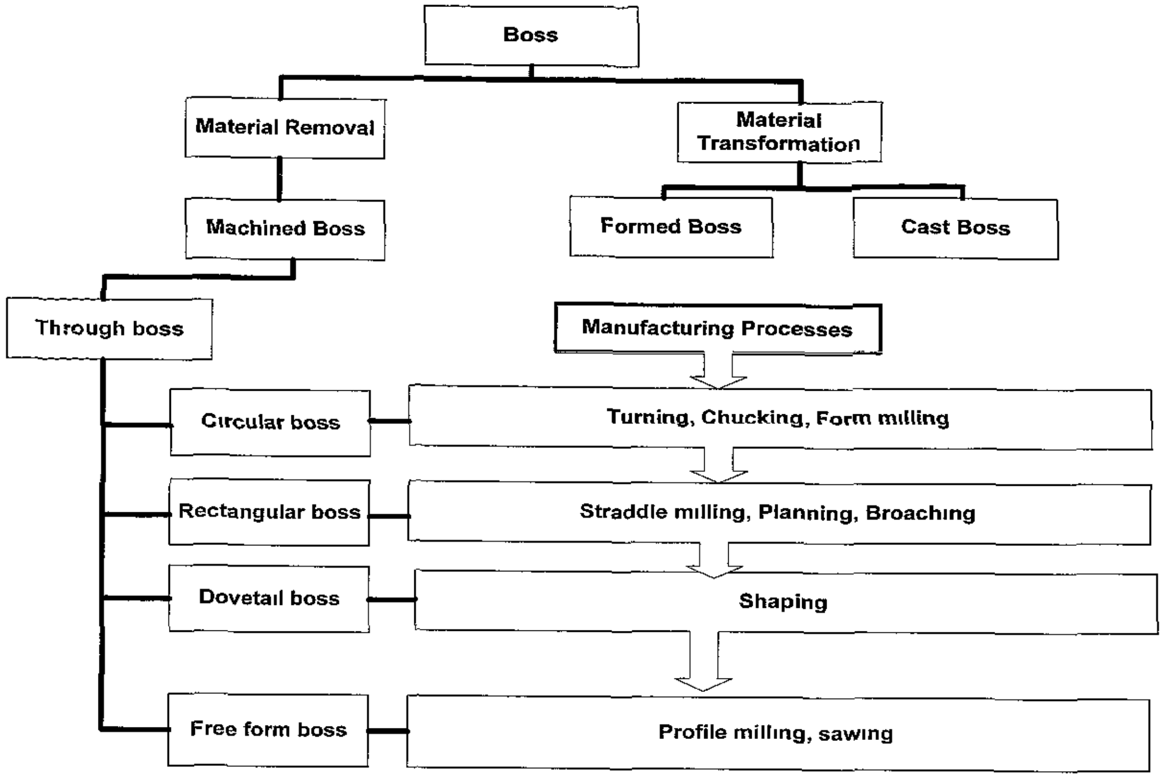


Fig. 3.28: Through boss classification of Machined bosses.

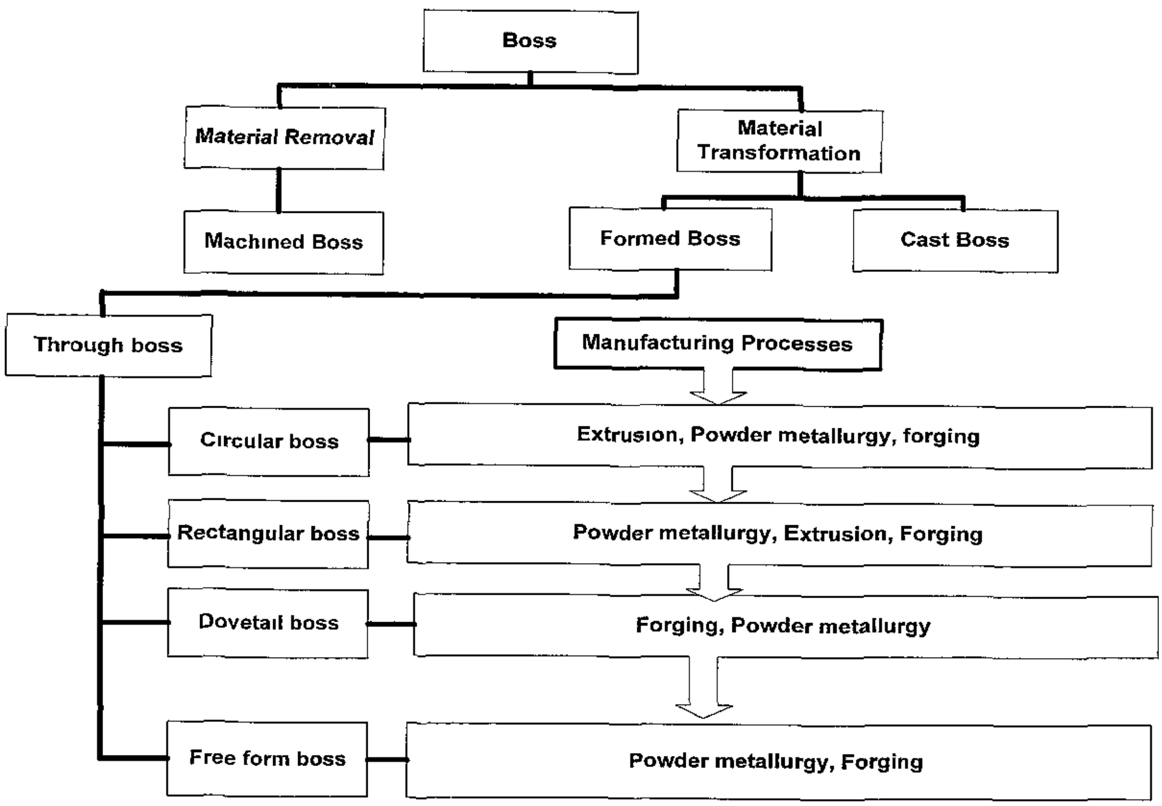


Fig. 3.29: Through boss classification of Formed bosses.

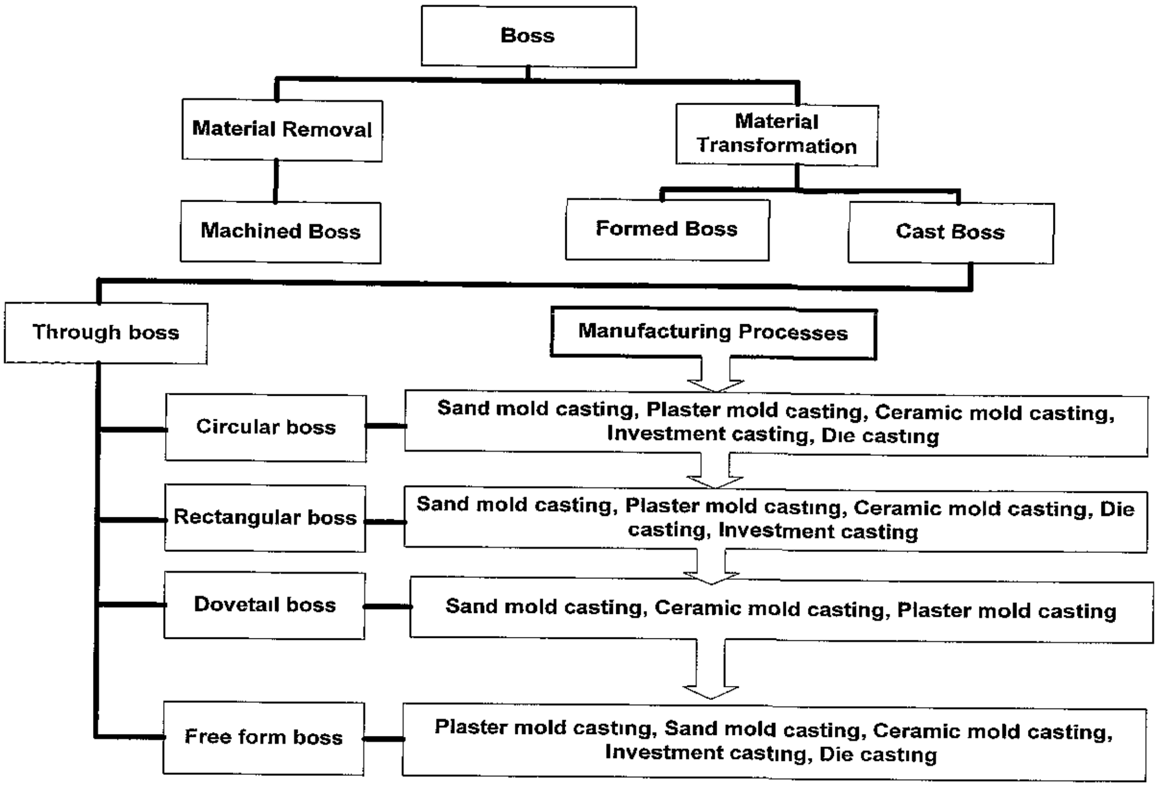
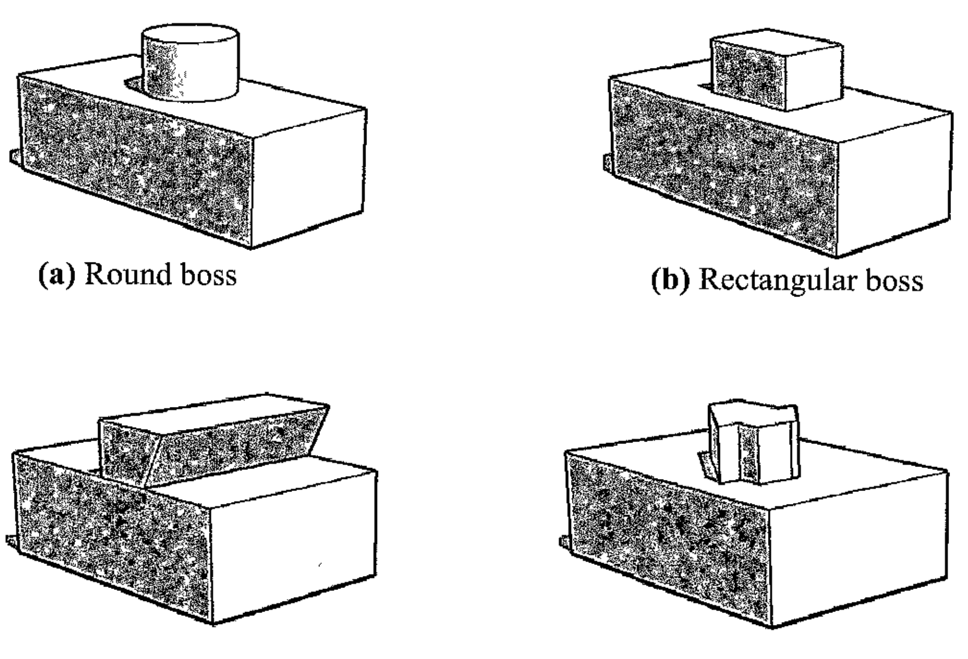


Fig. 3.30: Through Boss classification of Cast bosses.



(a) Round boss

(b) Rectangular boss

(c) Dovetail boss

(d) Free form boss

Fig. 3.31: Different Boss features.

3.10 Step Features

Another new approach for feature classification is the step feature classification method. Step features can be classified into Machined step, Formed step and Cast step. Machined step is the step which is manufactured by a machining process, Formed step is manufactured by formation of metal and Cast step is manufactured by a casting process. Step features can first be classified into two categories, Open step and Blind step. Open step can be classified into three categories; Rectangular, Wedge and Round steps. Blind step can be classified in the same way. Fig. 3.32 to fig. 3.37 describes the classification of pocket feature based on the example as indicated in fig. 3.38 and fig. 3.39.

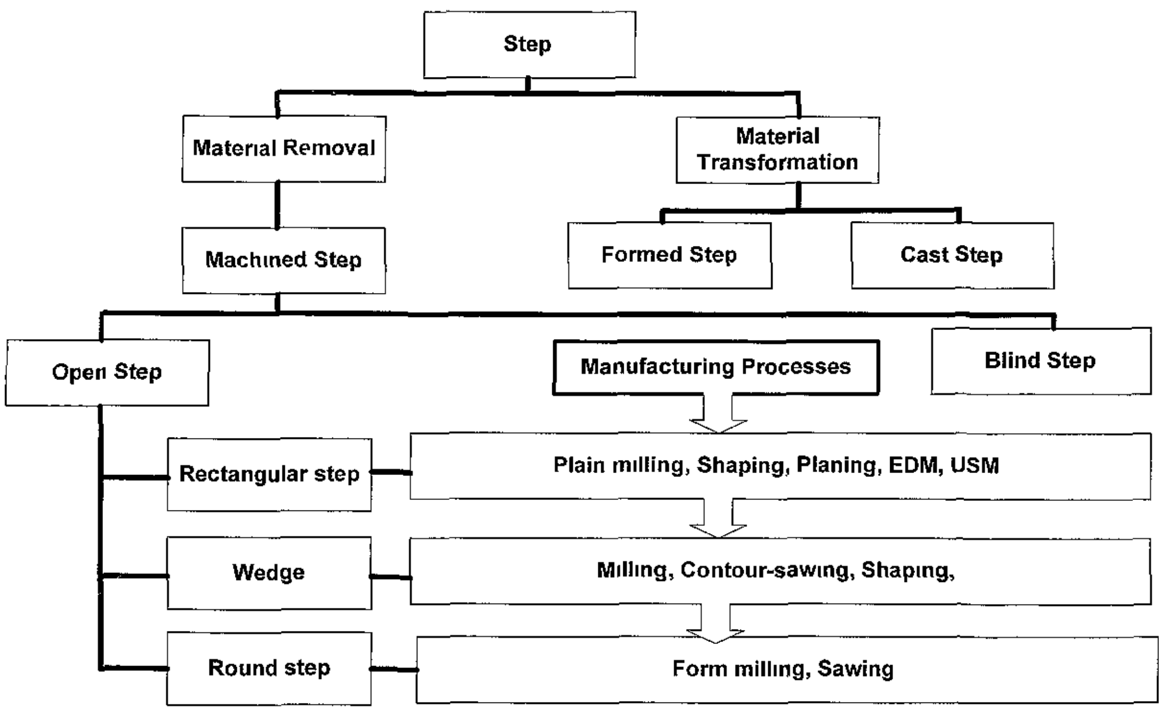


Fig. 3.32: Open step classification of Machined steps.

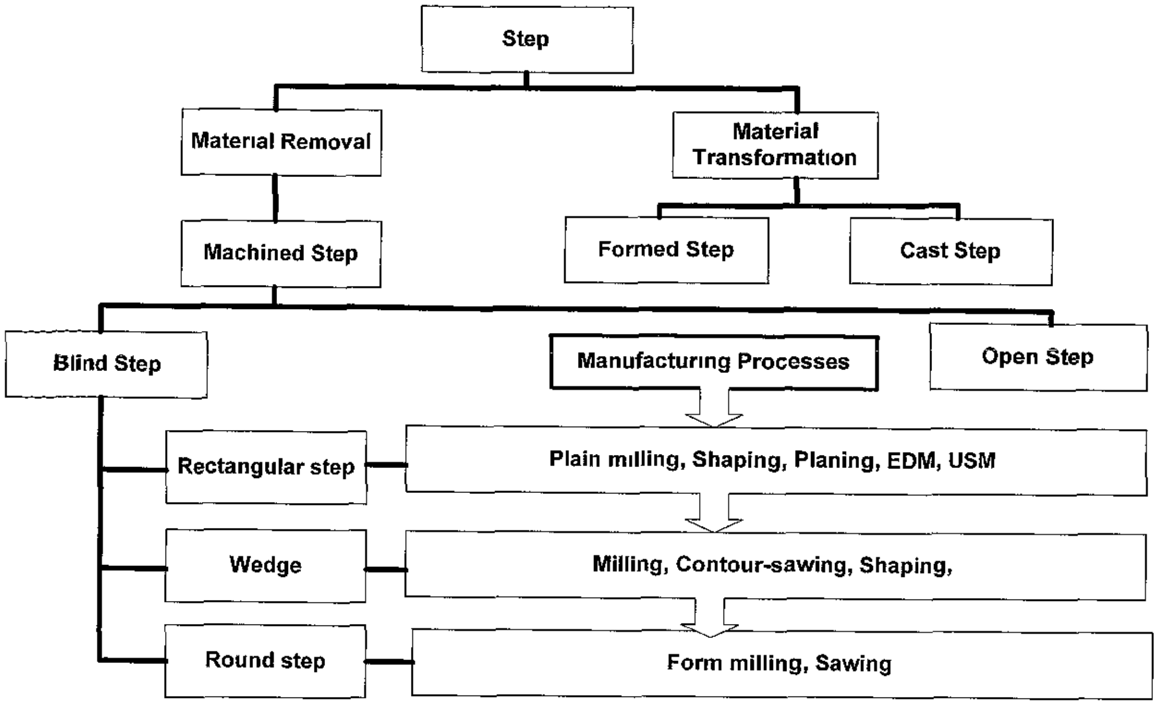


Fig. 3.33: Blind step classification of Machined steps.

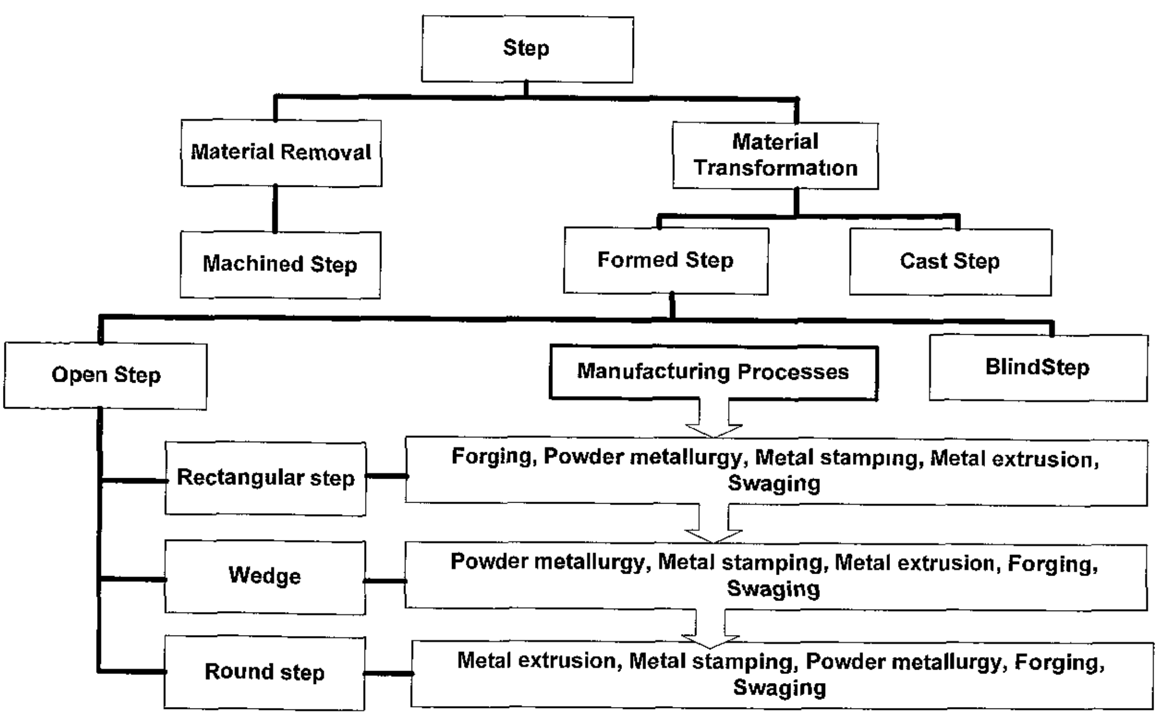


Fig. 3.34: Open step classification of Formed steps.

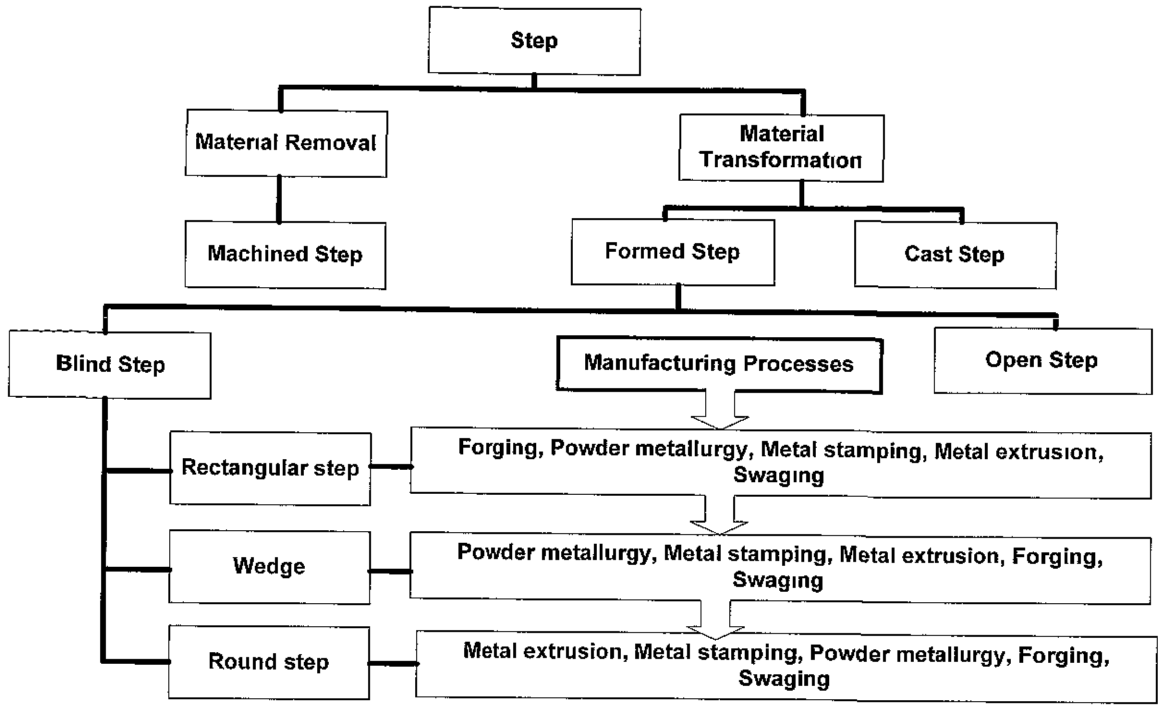


Fig. 3.35: Blind step classification of Formed steps.

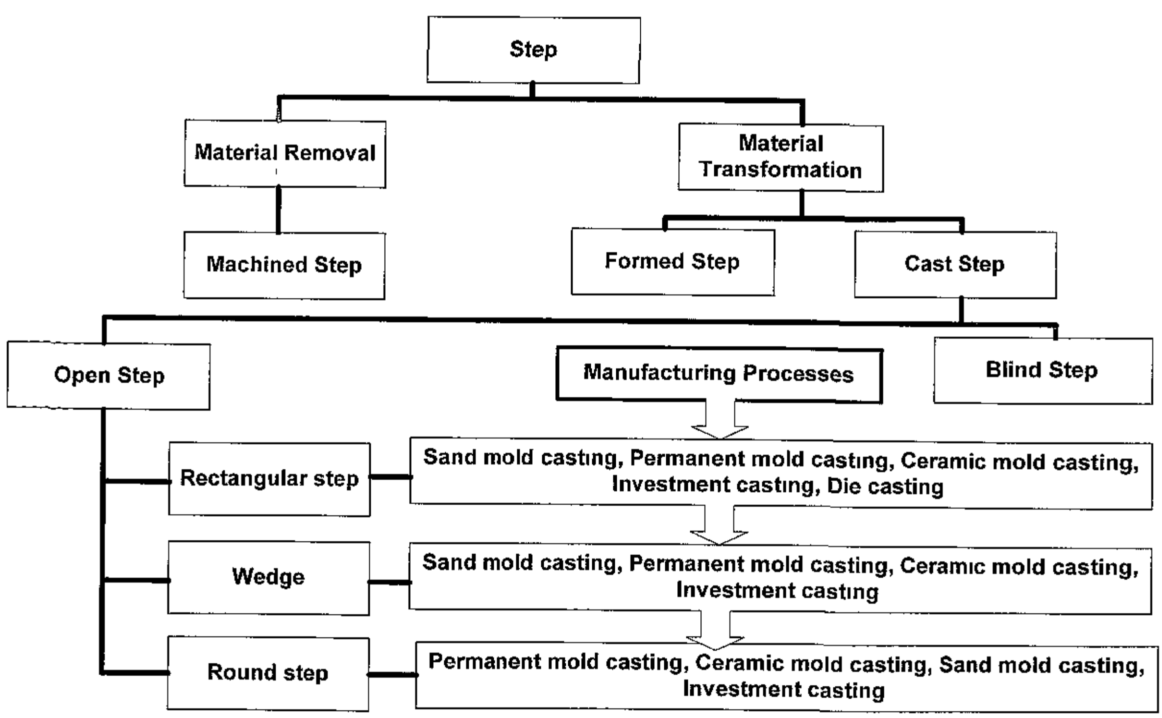


Fig. 3.36: Open step classification of Cast steps.

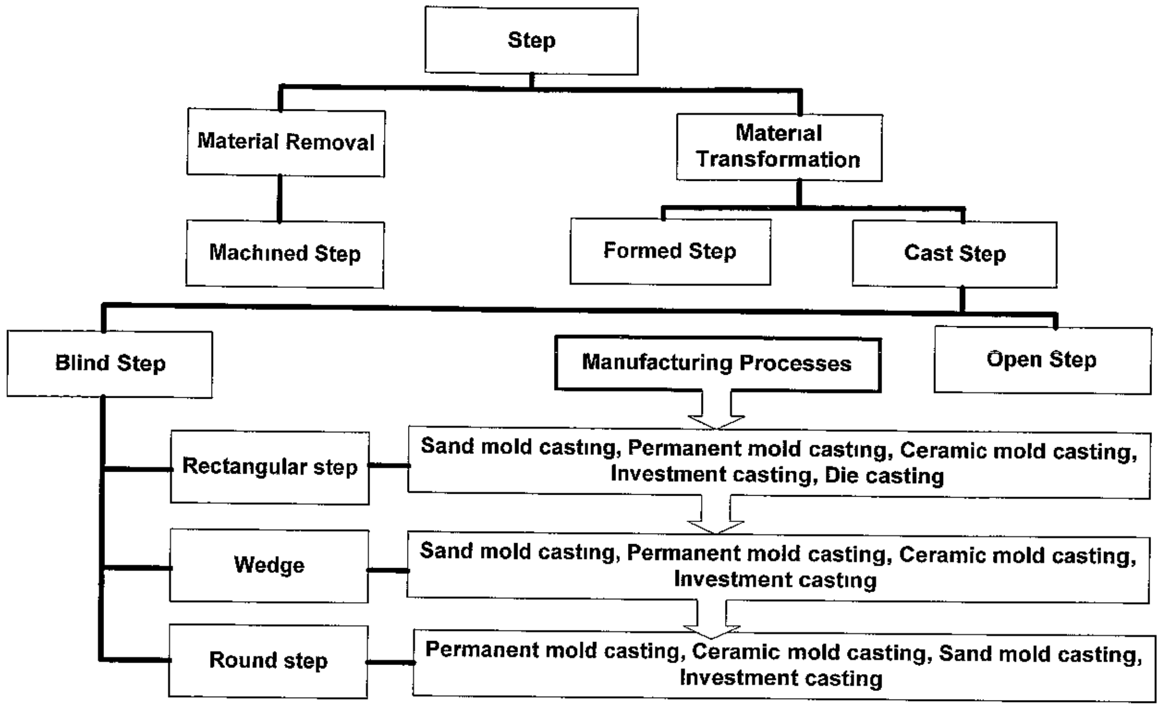
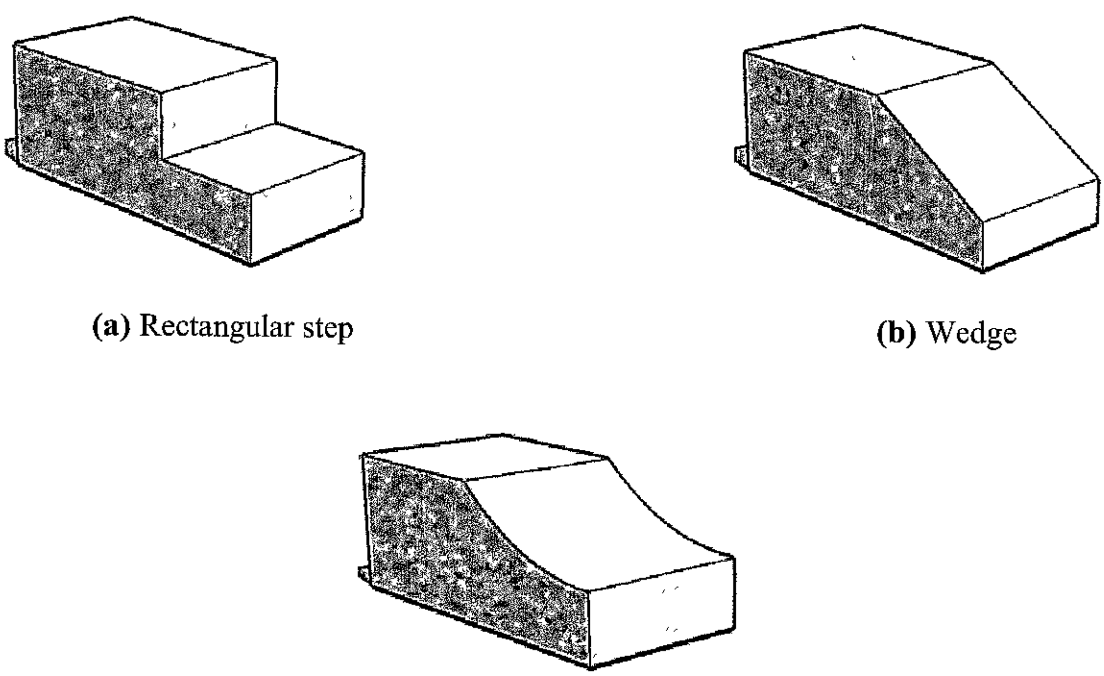


Fig. 3.37: Blind step classification of Cast steps.

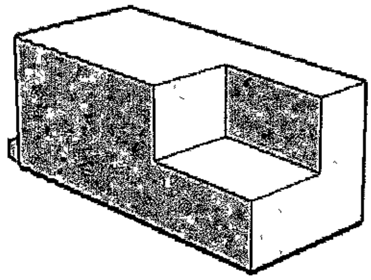


(a) Rectangular step

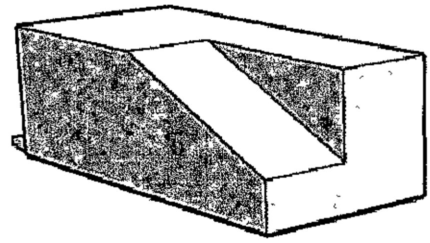
(b) Wedge

(c) Round step

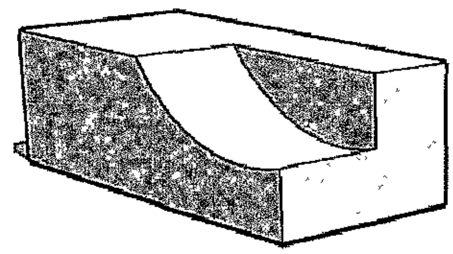
Fig. 3.38: Open step features.



(a) Rectangular step



(b) Wedge



(c) Round step

Fig. 3.39: Blind step features.

3.11 Manufacturability Analysis

As can be seen in Table 3.1, a Rectangular Through Slot is shown to have characteristics which are Cutter diameters, Slot depth, Slot width and the Depth-to-diameter ratio of the slot. The geometrical and topological characteristics are known from the design stage. The DFM system provides the information about the production type, Material, Tolerances and the surface finish of the part that can be used by the designer. In our example the End Milling process is selected with the manufacturing constraints of this process applied to the Slot feature and thus it warns the designer about the limitation of the process.

Table 3.1: Manufacturability Analysis of Slot Feature

Feature	Production Type	Material	Surface Finish [μm]	Depth to diameter Ratio	Slot Width [mm]	Tolerances [mm]	Manufacturing Processes
Rectangular Through Slot	Mass Production	Steel	1.5-3.8 μm	d/D≤ 1	6 3 to 25 mm	± 0.05 to ± 0 06 mm	End Milling

Another example of Manufacturability analysis of a hole feature is shown in Table. 3.2. The limitation of the drilling process from the economical point of view is the maximum value of the depth-to-diameter ratio, which should not exceed 3:1. If these limits are exceeded the product's cost will be significantly increased. The aim of our Design for Manufacturing approach is to eliminate the extra cost.

Table 3.2: Manufacturability Analysis of Hole Features

Feature	Production type	Material	Surface Finish (μm)	Depth-to-diameter ratio	Tolerances (mm)	Manufacturing processes
Cylindrical hole	Mass Production	Steel	1.6-3.2	$\leq 3:1$	$\pm(0.05\text{-}0.25)$	Drilling

3.12 Summary of This Chapter

In this section of feature classification a new approach has been applied for classification of features. Features such as hole feature, pocket feature, slot feature, boss feature and step feature have been identified together with their possible characteristics and manufacturing processes. This system helps the manufacturer to select the correct manufacturing process which can be beneficial in terms of the production cycle time.

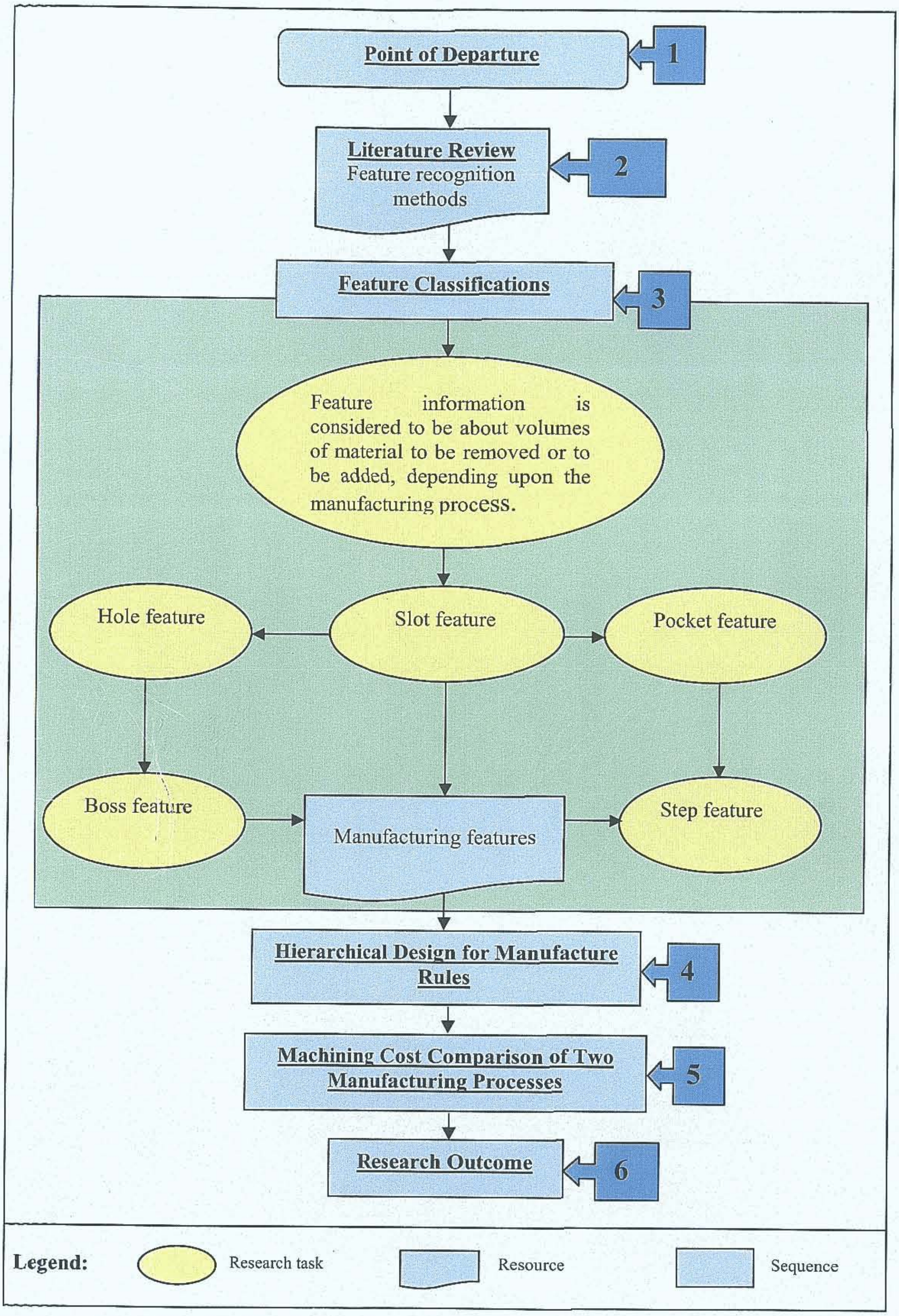


Fig. 3.40: Summary of the research process in chapter three.

CHAPTER 4 HIERARCHICAL DESIGN FOR MANUFACTURE RULES

4.1 Introduction

Design for manufacture is the consideration of process capabilities and limitations during the design process in order to minimise manufacturing cost. In order to shorten the product development cycle time, minimise overall cost, and smooth the transition into production, early consideration of manufacturing processes is important. This does not involve attempting to be correct in all aspects of the design. It aims to reduce costs and improve the ease with which products can be made. The concept of DFM is not really new, in 1788 LeBlanc, a Frenchman, devised the concept of interchangeable parts in the manufacture of muskets which previously were individually handmade [71].

In the world of competitive markets it is important to control the product price while maintaining quality. There is a conflict between the manufacturer and the consumer about the cost and the quality of the product. To figure out this problem early selection of manufacturing processes is important. If the correct manufacturing processes are selected the result is lower production time, reduced labour and overall production cost. If the production cost is lower then consequently the overall cost of product will automatically be less. In chapter three classifications of features have been shown with their possible manufacturing processes. In chapter four the design for manufacture rules of manufacturing processes will be discussed.

4.2 DFM Procedure

Martin O' Driscoll [72] described the principle of DFM which avoids the redesign and unexpected cost through the integration of the activities indicated in Fig. 4.1. The proposed DFM procedure contains a descriptive guide concerning the activities which should be undertaken to improve the manufacturability of a product.

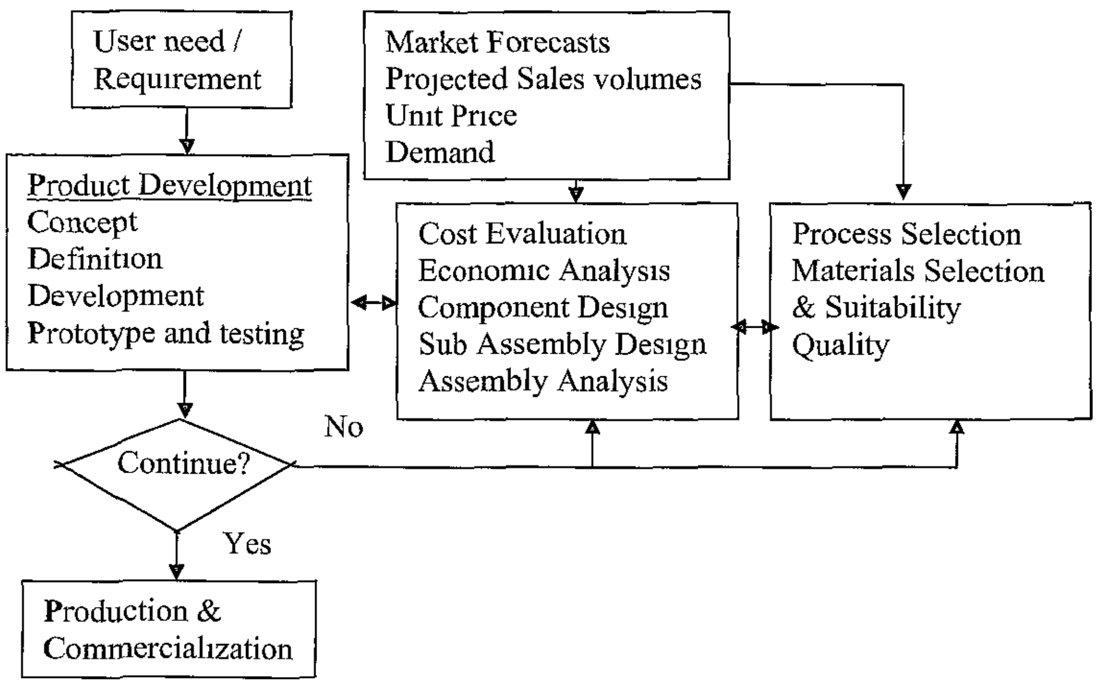


Fig. 4.1: Typical DFM flowchart.

4.3 General Design Guidelines for Manufacturability

- Create designs with lower number of parts where possible by designing one part so that it performs several functions. As the number of parts goes up, the total cost of fabricating and assembling the product goes up. Extra design documents and manufacturing processes result in a more expensive product due to NRE (Non-Recurring Engineering) and manufacturing costs [73].

- Avoid design for high labour-cost operations whenever possible. For example a punch-press-pierced hole can be made more quickly than a drilled hole. Drilling in turn is quicker than boring. Tumble deburring requires less labour than hand deburring [71].

- Designs should consider the hole spacing in machined, cast, moulded, or stamped parts so that they can be made in one operation without tooling weakness (fig. 4.2) [71].

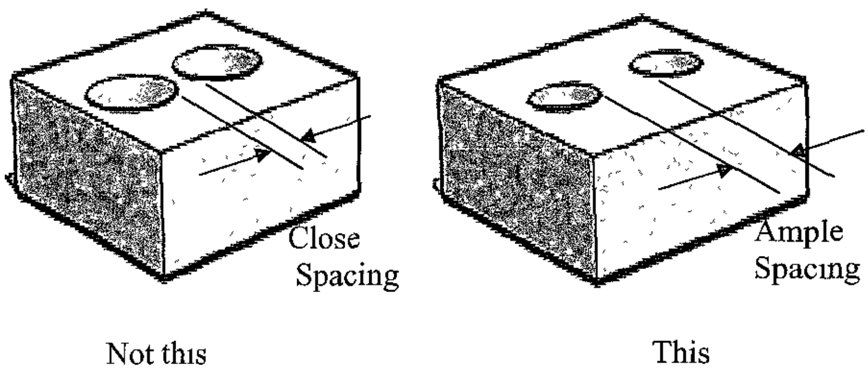


Fig. 4.2: Minimum hole spacing for manufacturing processes.

- Generally, design a part in such a way that as many operations as possible can be performed without other machining operations. This reduces the number of operations and handling time, but equally importantly promotes accuracy since the required precision can be built into the tooling and equipment [74].

- Avoid designing parts that require sharp corners and sharp points in cutting tools because these increase the probability of cutting tool breakage. Use generous fillets and radii. Generally rounded corners provide a number of advantages. There is less stress concentration on the part and on the tool. Some exceptions cannot be avoided, eg:
  "The external corners of a powder-metal part where surfaces formed by the punch face intersect surfaces formed by the die walls, will be sharp"[71].

- Avoid generalised statements on drawings which may be difficult for manufacturing personnel to interpret. Examples are; "Polish this surface", "Corners must be sharp", "Tool marks are not permitted" and "Assemblies must exhibit good workmanship".

- Avoid the design that requires special tooling (dies, form cutter, gun drilling etc) whenever possible, except for the highest levels of production, where the labour and materials saving of special tooling enable their costs to be controlled. Designers should become familiar with general purpose and standard tooling [73].

- Avoid dimensioning from space points; instead, dimension from the specific surfaces or datum points on the part itself as much as possible. This greatly facilitates fixture and gauge making and helps avoid tooling, gauge, and measurement errors (fig. 4.3) [71].

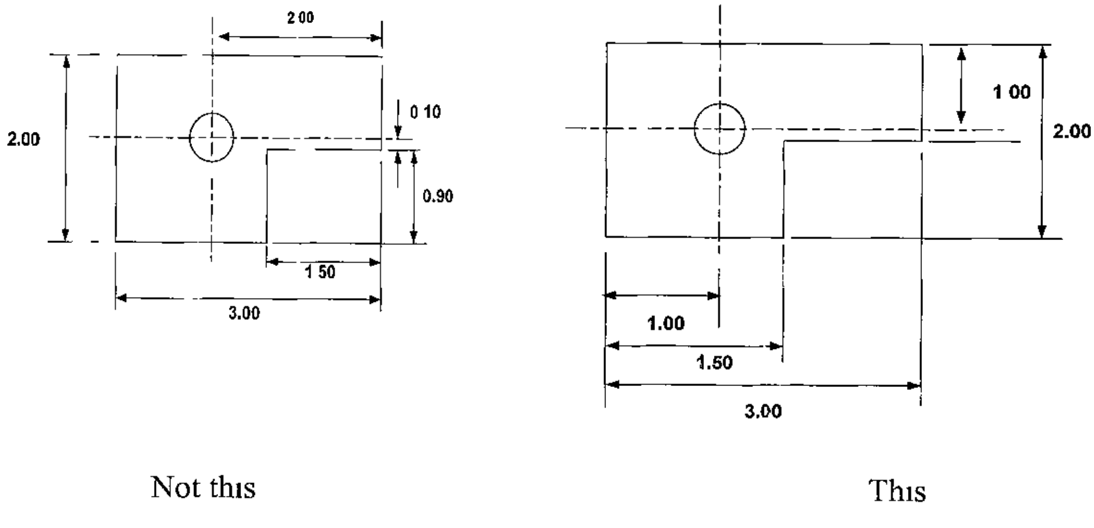


Fig. 4.3: Dimensions should be made from points on the part itself rather than from points in space.

- Avoid stepped parting lines from the design of cast, moulded, or powder-metal parts which increase mould and pattern complexity and cost.

- Design parts in such a way that for all casting and moulding processes the wall thickness should be as uniform as possible. This is more important for high-shrinkage materials (e.g., plastics and aluminium) (fig. 4.4) [74].

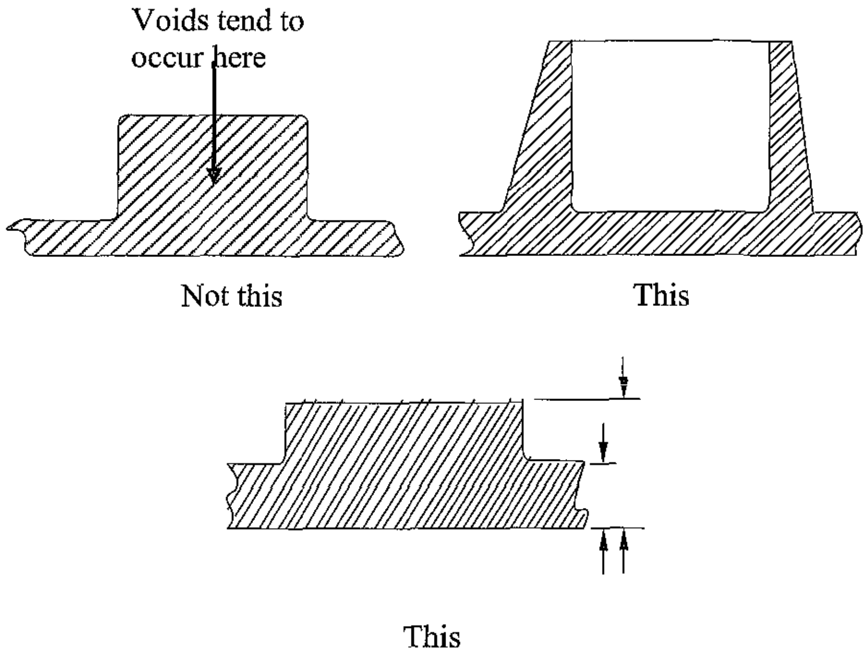


Fig. 4.4: Design suggestions for minimising material thickness at bosses.

- In the design it is necessary to consider surfaces that would allow accurate, stable and reliable fixture.

- The required accuracy and roughness of surfaces should be compliant with the functionality of the surfaces.

- When dimensioning surfaces the functional relationship between those surfaces should be considered. The application of this principle assures the shortest dimension chain which leads to maximum specifiable tolerances. The illustration in Fig. 4.3. is an example of this requirement.

4.4 General Design Guidelines for Machining Processes

- Avoid machining operations if possible. For higher volume parts, consider castings, extrusions or other volume manufacturing processes to reduce machining cost and machining time (fig. 4.5) [73].

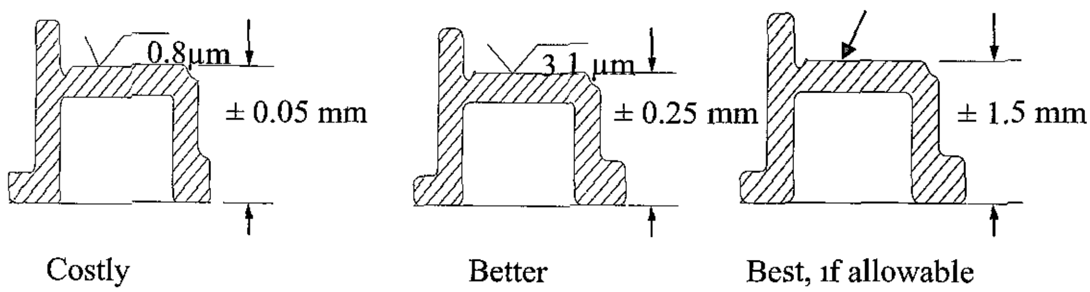


Fig. 4.5: Machining guidelines.

- To avoid costly secondary operations like grinding, reaming, lapping etc, specify the most liberal surface finish and dimensional tolerances whenever possible, consistent with the function of the surface (fig. 4.5) [74].

- Select materials with high machinability as much as possible. Hardened materials are difficult to machine and process using other operations. Harder materials also decrease cutting tool life.
- Designs should be applied in such a way that they can be easyly fixtured and held securely during machining operations. To assure a secure set-up large mounting surfaces with parallel clamping surfaces should be provided.

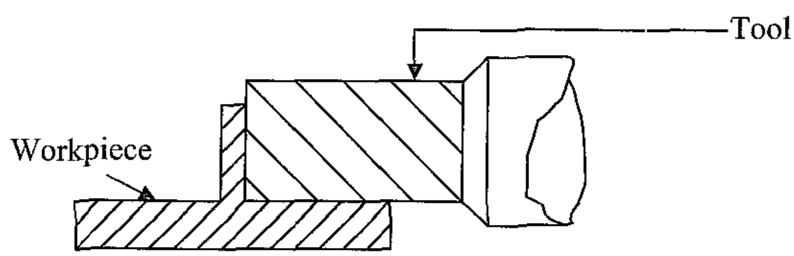


Fig. 4.6: Design parts to be rigid enough to withstand machining forces.

- Design parts to be rigid enough to withstand clamping forces without distortion. Thin slender work pieces are difficult to support properly to withstand clamping and cutting forces. The cutter tool exerts severe forces onto the workpiece which causes vibration and chatter, so the workpiece must be able to withstand the clamping forces necessary to hold the workpiece securely (fig. 4.6) [71].

- Design parts in such a way to avoid undercuts which usually involve separate operations of specially ground tools (fig. 4.7) [73].

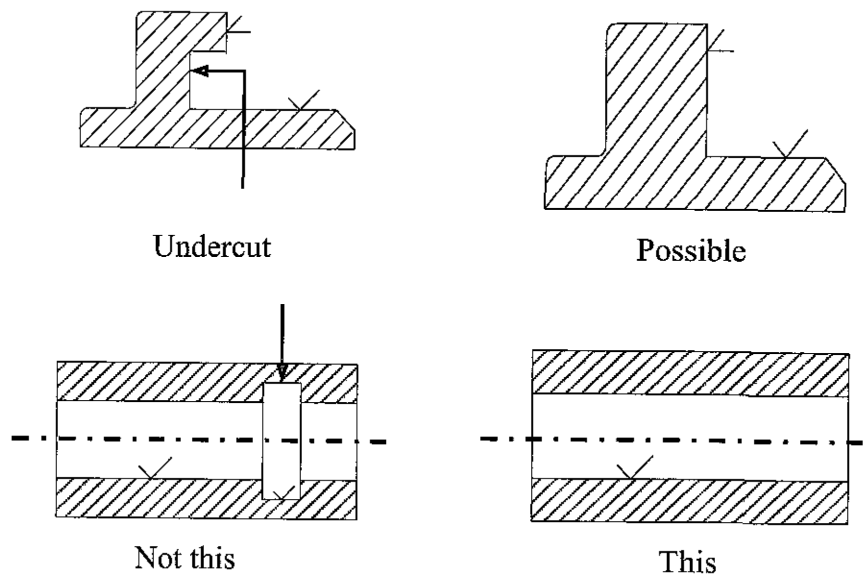


Fig. 4.7: Avoid undercut as much as possible since they require extra machining operations, which may be costly.

- Design parts in such a way that standard cutters can be used instead of special formed cutters (fig.4.8) [73].

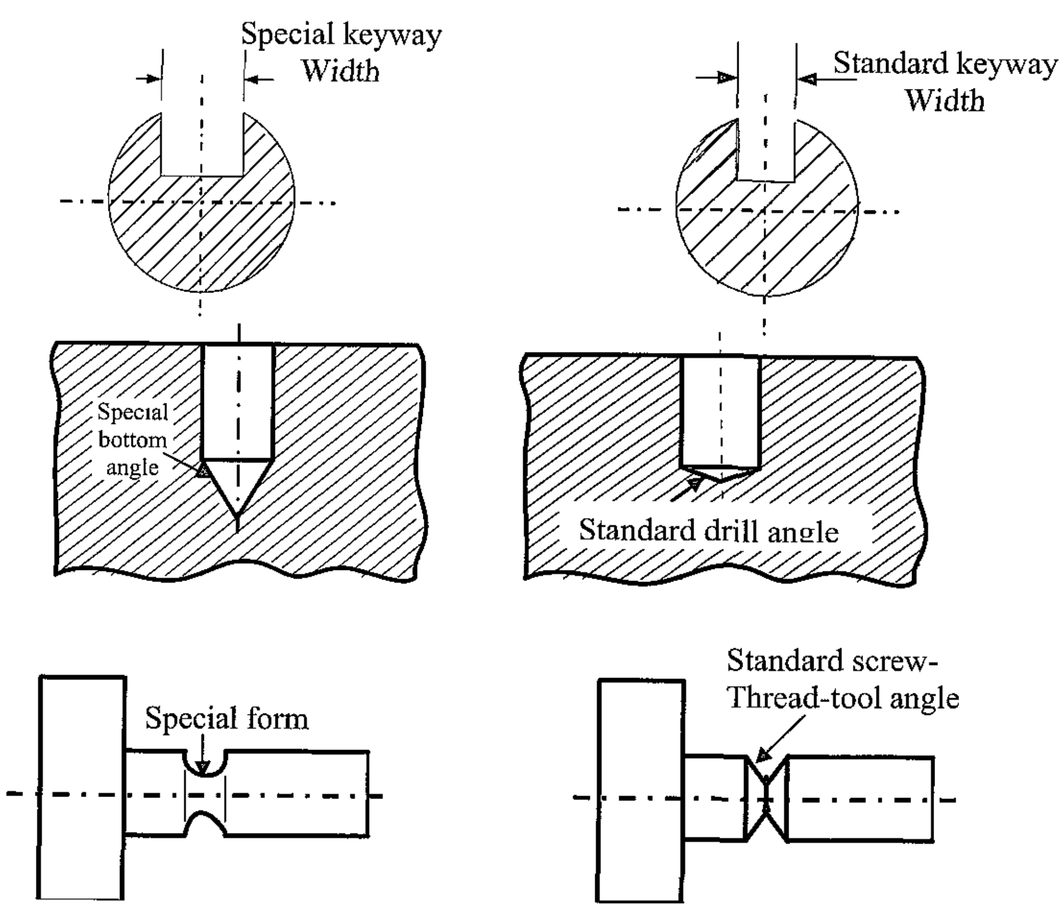


Fig. 4.8: Design parts so that standard cutting tools can be used.

- Avoid tapers and contours as much as possible in favour of rectangular shapes, which permit simple tooling and setup.

- Avoid projections, shoulders, etc., which interfere with clamping or locating surfaces. Instead, provide clearance space at the end of the cut. The space can be cast or formed to minimise machining.

- Design parts so that a rigid tool can be used and the access to the surface is still guaranteed (fig. 4.9) [71].

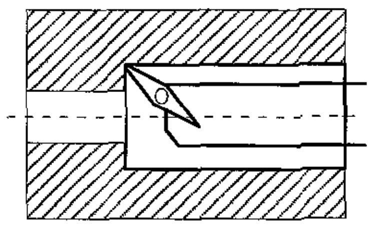


Fig. 4.9: Use of a rigid tool.

4.5 Design Guidelines for Round Shapes Machining

4.5.1 Turned Parts:

4.5.1.1 Turning (External)

- The design should be considered to incorporate standard tool geometry at diameter transitions, exterior shoulders, grooves and chamfer areas.

- The design should consider using standard, commercially available cutting tools, inserts, and tool holders.

- Design parts with radii large enough (if possible) and conform to standard tool nose radius specifications.

- The design should consider that when a knurling operation is required parts should be kept narrow and its width should not exceed its diameter.

- The design should consider that external grooving is easier than internal grooving because it is easier to incorporate with external surfaces.

- Avoid the design of longer parts (if possible) which requires extra support. Short, stubby parts are easier to machine than long, thin parts. Short and stocky parts minimise deflection (fig. 4.10) [74].

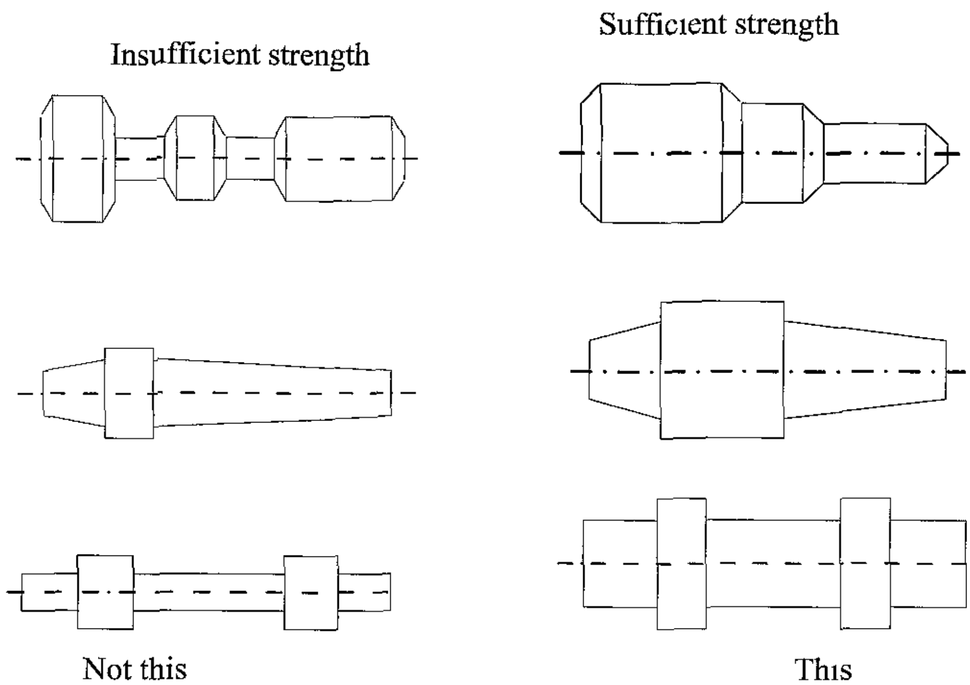


Fig. 4.10: Keep parts as short as stocky as possible to minimise deflection.

- Design parts in such a way that allows room for the threading tool to exit (fig.4.11).

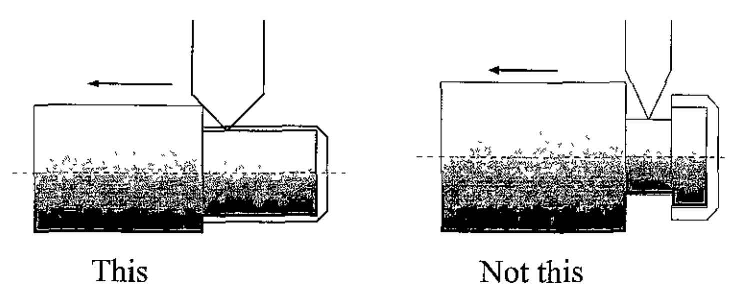


Fig. 4.11: Cutting tool operation can be performed without any obstruction.

- Whenever possible irregular and interrupted cutting action should be avoided from the product design. For example- hole intersections, curved or slant surface drilling and hole or slotting operations before turning are not preferable [73].

- Design parts so that they can be machined from one side. This eliminates chucking, no extra equipment is necessary (fig. 4.12).

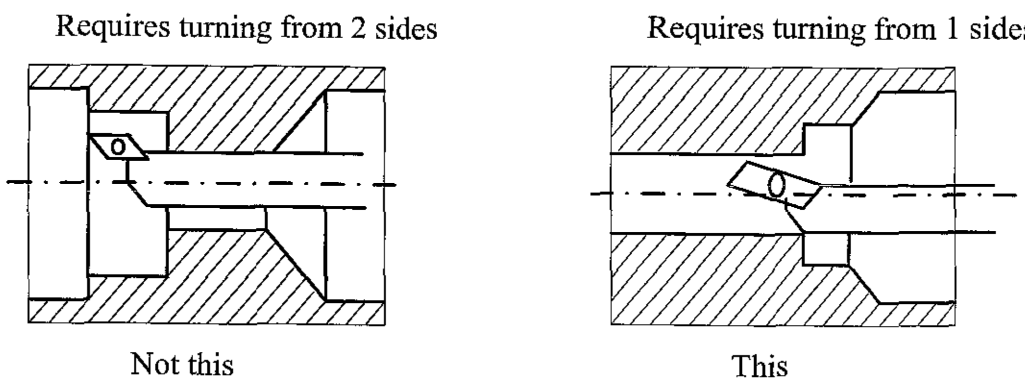


Fig 4.12 One sided machined operation eliminates chucking and extra equipment.

- Parts with long, formed areas should not exceed 2 ½ times the minimum workpiece diameter [71].

- The design should consider that for castings or forgings with large shoulders or other areas to be faced, the surface should be 2 to 3° from the plane normal to the axis of the part (fig. 4.13) [73].

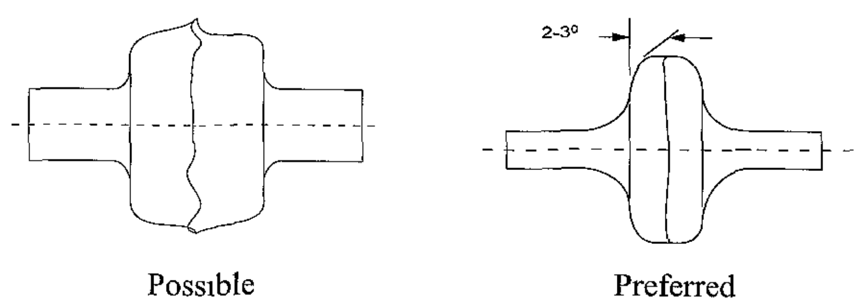


Fig. 4.13: Surface should be 2-3° from the plane normal to the axis for castings or forgings with large shoulders.

- The design should consider that for external threading, space must be provided for the thread-cutting tool.

- Design parts with the area of thread relief or undercut where the diameter of the workpiece is less than the minor thread diameter.

4.5.1.2 Turning (Internal)

- The design should consider that for blind hole threading chip clearance is important so that parts should require some unthreaded length at the bottom.

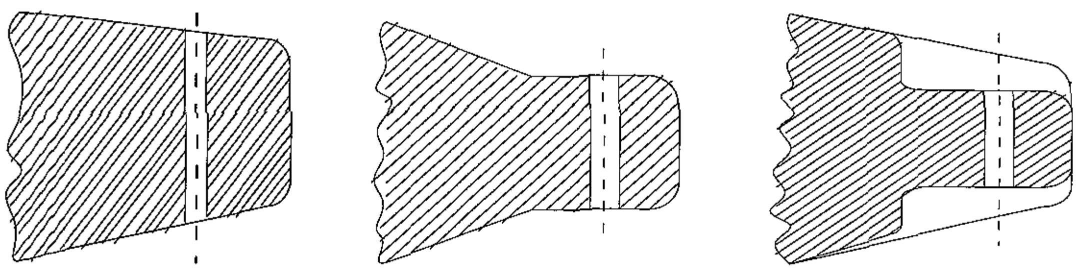
- The design should consider that for internal threads where tap breakage may be a problem, limit the depth of the threaded portion to two diameters.

- Design parts in such a way that if possible internal grooving should be avoided because the operation requires tools with both axial and transverse motion.

4.5.2 Round Hole Making:

4.5.2.1 Drilling

- Design parts in such a way as to avoid tool entry problems and proper hole geometry. The drill entry surface should be flat and perpendicular to the drill motion (fig. 4.14) [73].



Not this This or This

Fig. 4.14: The entrance and exit surface should be perpendicular to the drill bit.

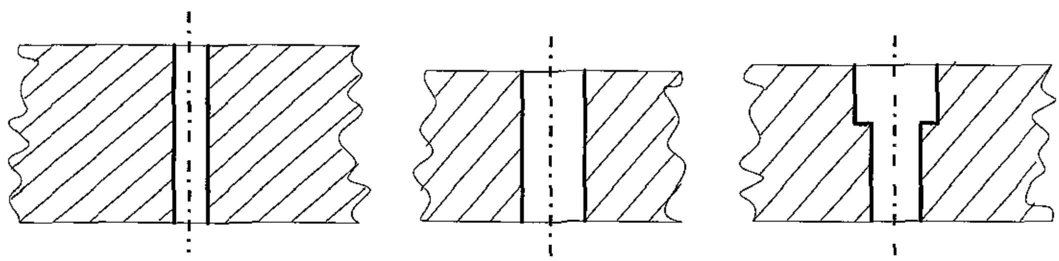
- To avoid breakage problems the exit surface should also be perpendicular to the drill axis (fig. 4.14) [73].

- The designed drilled hole depth (to the sharp point of the tool) is recommended to be at least equal to the full thread plus ½ major diameter, but never less than 1.3 mm.
- Avoid special drill operations (if possible) which are more costly and increase the product's price.

- The design should consider that through holes are preferable to blind holes, especially when secondary operations such as reaming, tapping, or honing are required for final finishing.

- Hole bottoms are most economical if they use standard drill- point angles. If flat bottoms are required, some drill-point depression in the centre should be allowed.

- To avoid chip-clearance problems and the possibility of deviations in the straightness of deep holes, holes over 3 times the diameter are not acceptable (fig. 4.15) [71].



(a) Not this **(b)** This or **(c)** This

Fig. 4.15: Avoid deep, narrow holes. For deep, narrow holes stepped diameter can be consider.

- Avoid design parts with very small holes (if not necessary). Drills with small diameters break easily. About Ó 3 mm is a desirable minimum for convenient production.

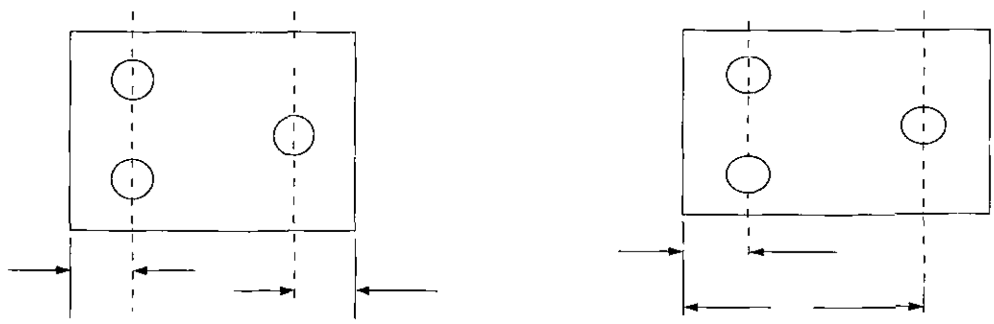
- To maintain roundness of open holes, designs that cause vibrations should be avoided (fig. 4.16) [73].

Not this This

Fig. 4.16: If holes with intersecting openings are unavoidable, it is important that the centre point of the drill remains in the work throughout the cut.

- Avoid the design of parts that require large finish holes. If necessary it is preferable to have cored (cast-in) holes in the workpiece prior to the drilling operation. This increases the tool life, allows material savings and reduces the power required for drilling.

- To simplify fixturing, dimension parts from the same surface, whenever they require several drilled holes (fig. 4.17) [73].



Not this This

Fig. 4.17: Locate all holes from one surface insofar as possible.

- In a design the location of drilled, reamed and bored holes, are better specified in a rectangular rather than in an angular co-ordinate system (fig. 4.19).

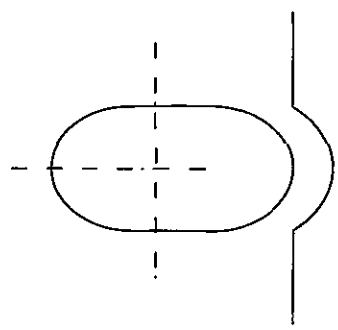


Fig. 4.18: Avoid holes with thin walls.

- Designs should consider that all drillings can be done from one side and with a minimum of fixturing or repositioning of the workpiece, which simplifies tooling and minimises handling time.

- Avoid holes with thin walls. Allow sufficient distance to withstand clamping and cutting forces, otherwise the wall deforms and the hole will not be round (fig. 4.18).

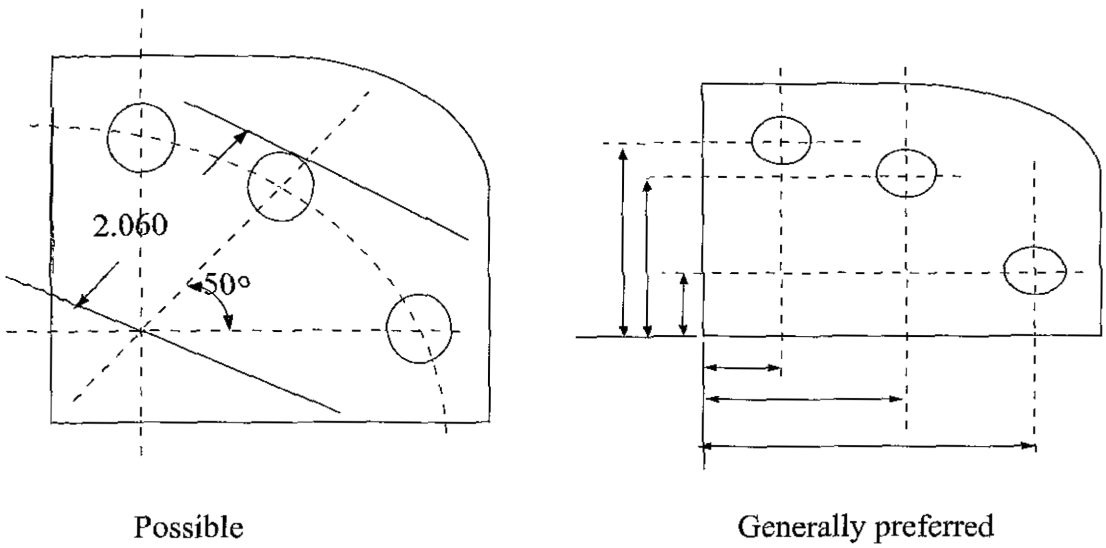


Fig. 4.19: Rectangular coordinates are preferable to angular coordinates for showing hole locations in drawings.

- Design parts so that there is room for a drill bushing near the surface where the drilled hole is started (fig. 4.20) [71].

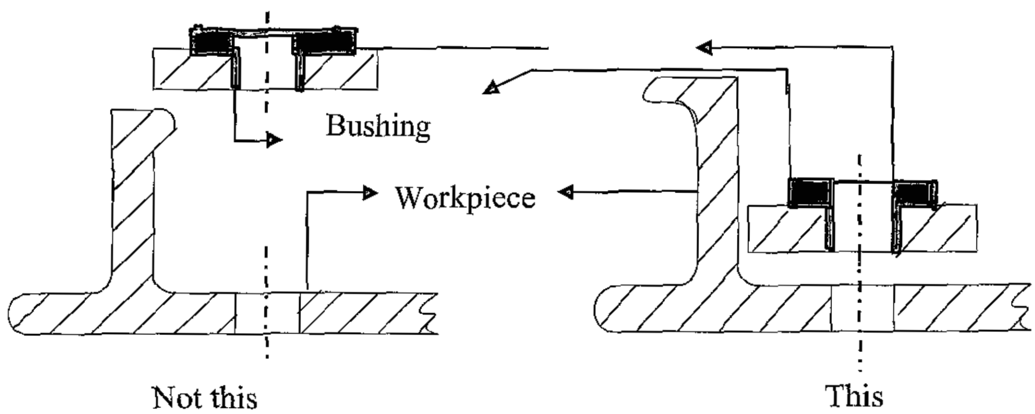
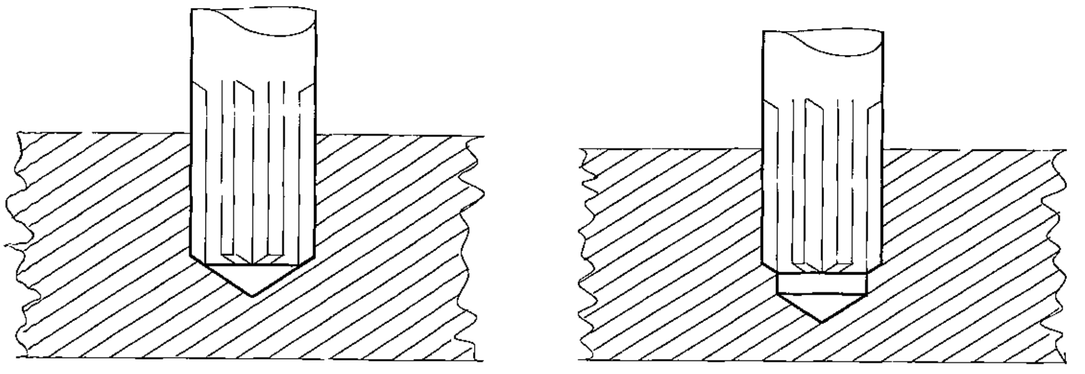


Fig. 4.20: Allow room for drill bushings close to the workpiece surface to be drilled.

4.5.2.2 Reaming

- Design parts with extra drill depth in blind holes to provide room for chips and to avoid heavy cutting conditions at the bottom of the hole (fig. 4.21) [74].

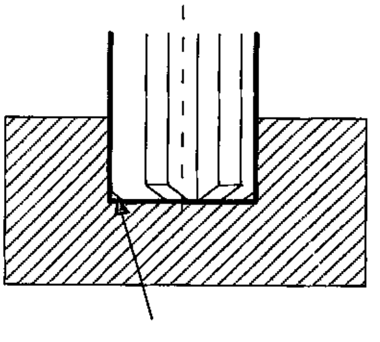
- Blind holes with flat bottoms can not be reamed close to the bottom because the reamer is tapered (fig. 4.22) [71].



Not this Preferred

Fig. 4.21: Provide extra hole depth if blind holes that are to be reamed.

- Reaming can not give correct location or alignment discrepancies unless the discrepancies are very small. It is a good practice to ream with a guide bushing when the hole location or alignment is critical.



Unmachined area

Fig. 4.22: Blind holes with flat bottoms.

- To prevent tool breakage and burr-removal problems, intersecting drilled and reamed holes should be avoided (fig. 4.23) [73].

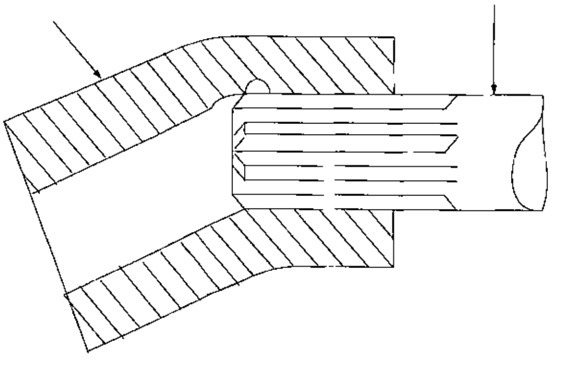


Fig. 4.23: Avoid intersecting drilled and reamed holes if at all possible.

4.5.2.3 Boring

- The design should consider that an interrupted surface tends to throw holes out of round and cause vibration and tool wear.

- To maintain accuracy avoid designing holes with a depth-to-diameter ratio over 4 or 5:1 (fig. 4.15) [71].

- Designs should consider that through holes are preferable than blind holes and for a large hole diameter a pre-existing hole is required (fig.4.24) [73].

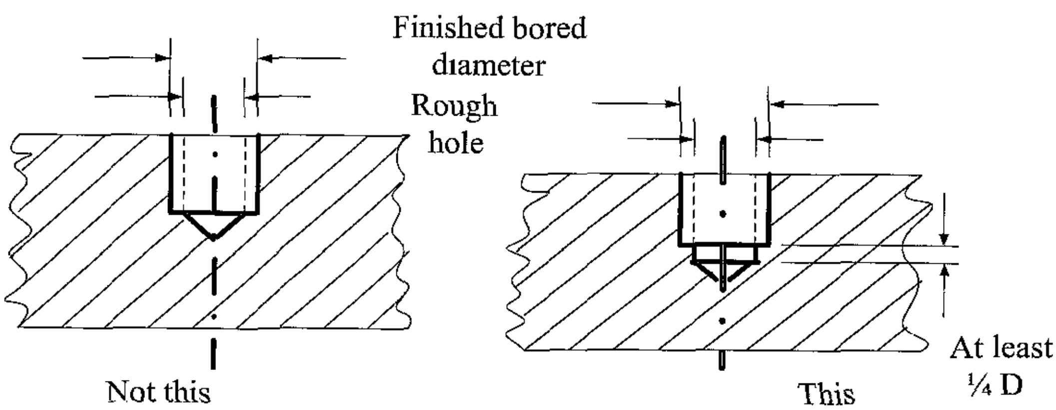


Fig. 4.24: Blind holes to be bored should be one-fourth diameter deeper than the final bored hole to allow space for chips.

- Avoid designing parts which require more costly manufacturing equipment. Boring, for example, is more expensive than drilling and reaming. Use the more costly operations only when the accuracy requirements demand it.

- The design should consider that, if the depth-to-diameter ratio is over 5:1 (8:1 for carbide bars), accuracy is limited due to the boring bar deflection (fig4.15) [73].

- Avoid designing parts with greater length-to-bore diameter ratios (if possible) as it is difficult to hold dimensions because of the deflections of the boring bar from cutting forces.

4.5.2.4 Trepanning

- The design should consider that trepanning is used in flat sheets or plates and this process can be used to make disks up to 150 mm in diameter.

- The design should consider that trepanning can make large, shallow through holes (of diameter equal to or greater than 5 times stock thickness) and machining circular grooves, such as would be used, for example, to retain O-rings [74].

4.6 Design Guidelines for Machining Various Shapes

4.6.1 Milling

- Design parts in such a way that the included corner shapes, chamfers, depth, width, radii and overall forms can be made using standard cutters. Special cutters are costly and difficult to maintain (fig. 4.25) [71].

- To avoid difficulty relating to the milling cutter, which has a finite radius, designs with internal cavities and pockets with sharp corners should be avoided.

- Avoid designs that specify a blended radius because exact blending is difficult to achieve.

- Design parts with standard keyway dimensions which permits a standard cutter to travel parallel to the centre axis of the shaft and can produce both sides and ends in one operation from its own radius (fig. 4.26) [73].

- Design parts with small steps or radii or inclined flange or shoulder surfaces for the clearance of cutter paths when milling surfaces adjacent to a shoulder or flange (fig. 4.27).

- In order to increase cutter life, the design should not include milling at parting lines, flash areas and weldments.

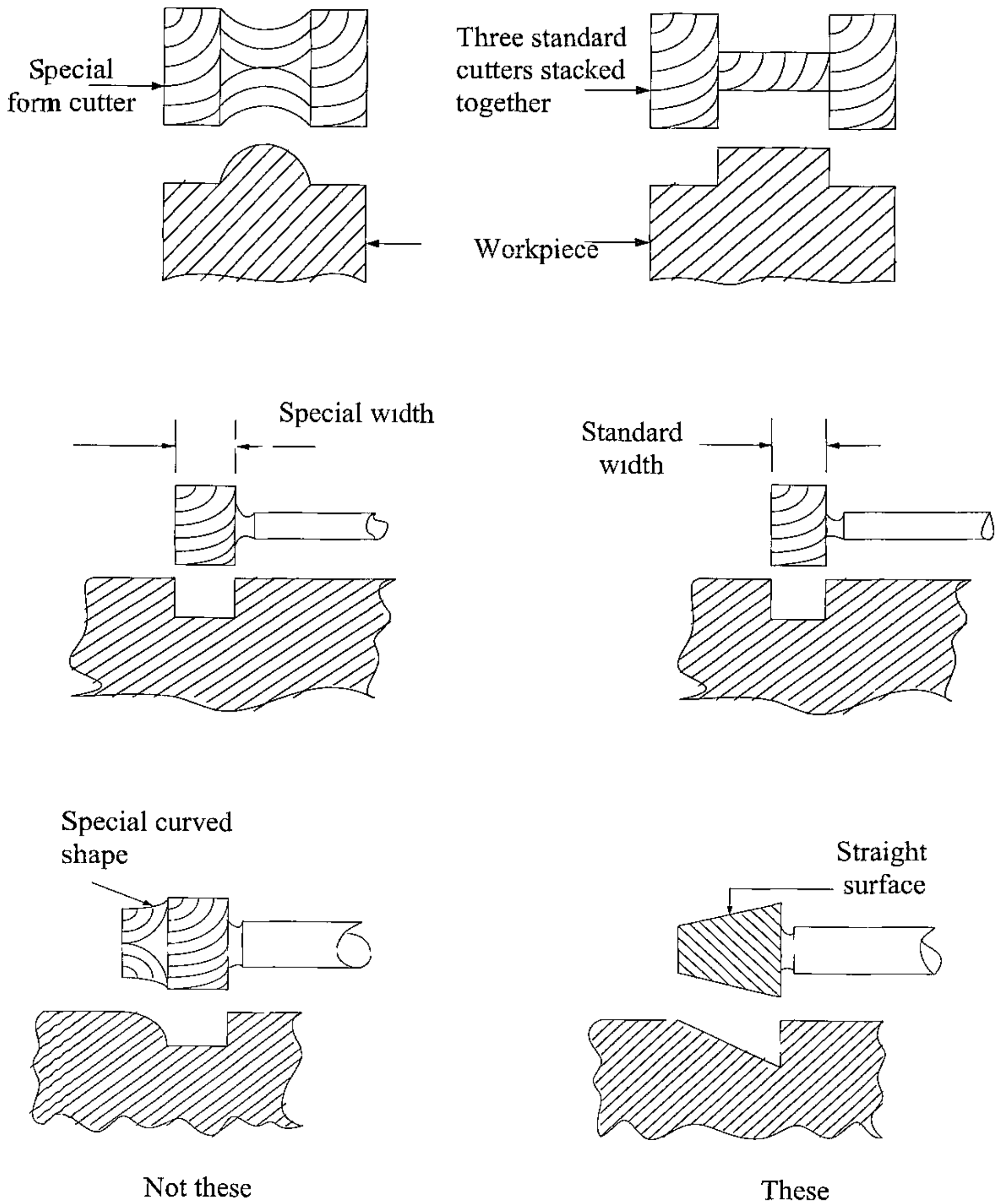


Fig. 4.25: Product design should permit the use of standard cutter shapes and sizes rather than special nonstandard cutter designs.

- Design parts that do not require large surfaces to be machined (fig. 4.28) [73].
- Design parts that include fewest separate operations which is more economical.

- Avoid blended radii on machined rails during form milling because exact blending is difficult to achieve (fig. 4.29) [71].

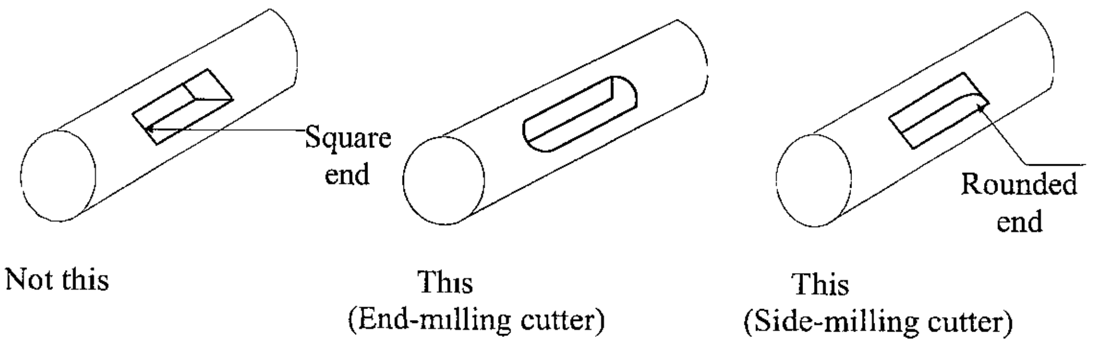


Fig. 4.26: Keyways should be designed so that a standard cutter can produce both sides and ends in one operation.

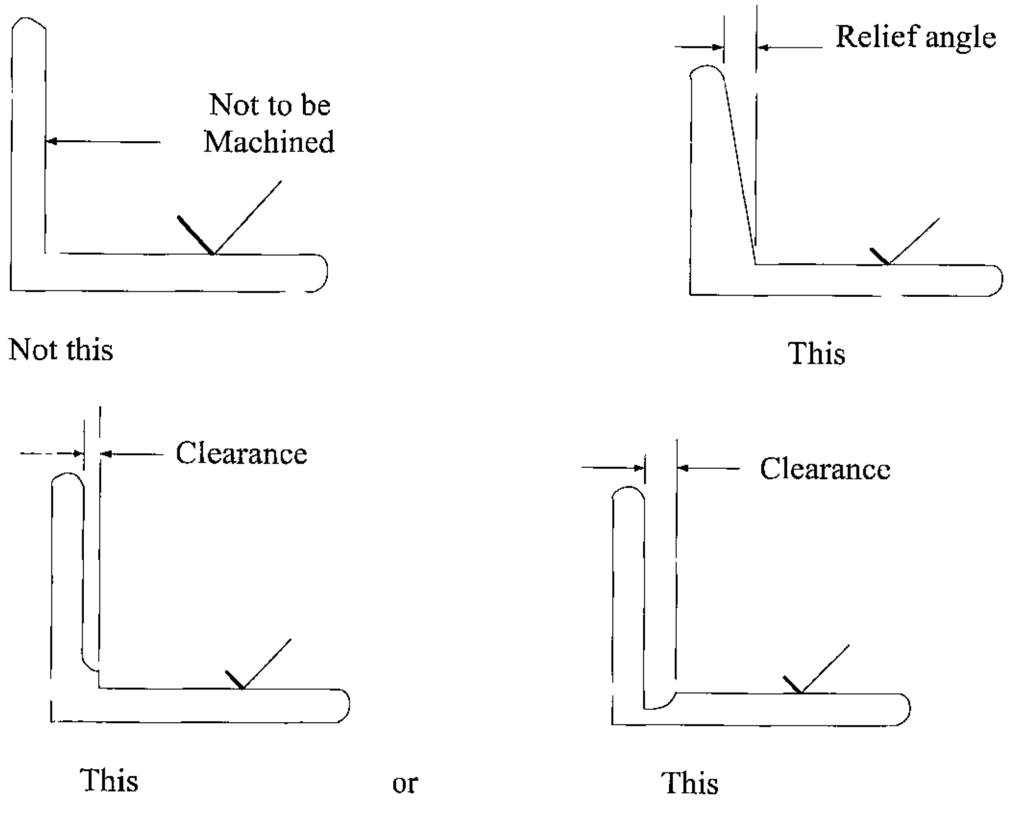


Fig. 4.27: Provide clearances for the milling cutter.

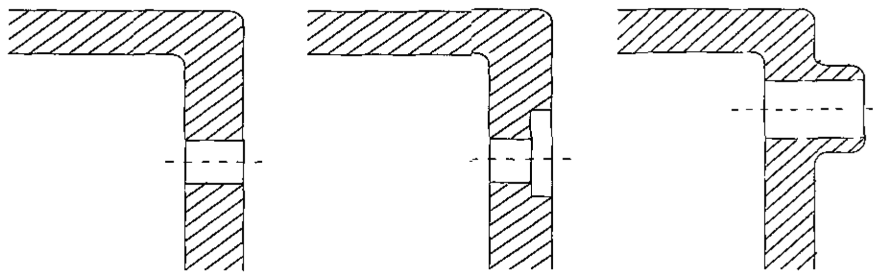


Fig. 4.28: Avoid large surfaces to be machined.

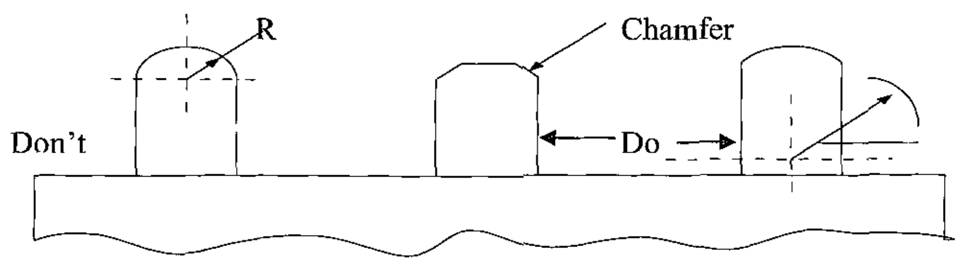


Fig. 4.29: It is better not to specify a blended radius on machined rails.

4.6.1.1 Face Milling

- Design parts which provide quicker and more economical processes. For example, spot facing is quicker and more economical than face milling (fig. 4.30) [71].
- Design parts which allow a bevel or chamfer rather than rounding if possible because rounding requires a form-relived cutter and more precise setup both of which are more costly than bevelling and chamfering (fig. 4.31) [71].
- In face milling, the ratio of the cutter diameter to the width of cut should be no less than 3:2.

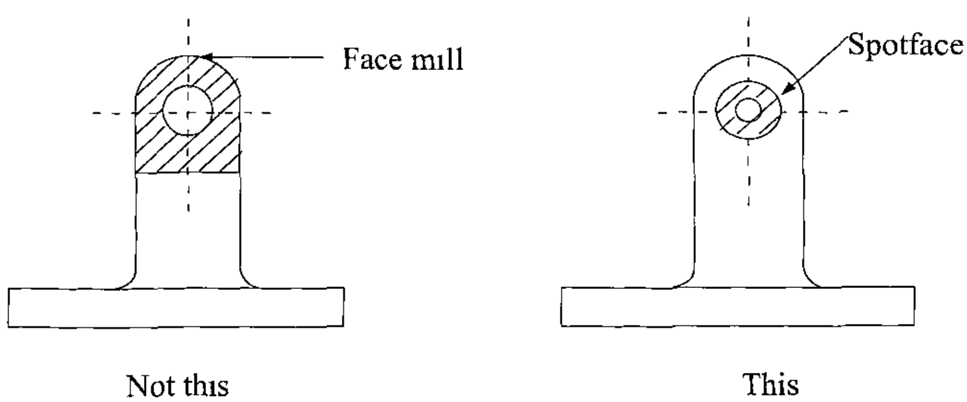
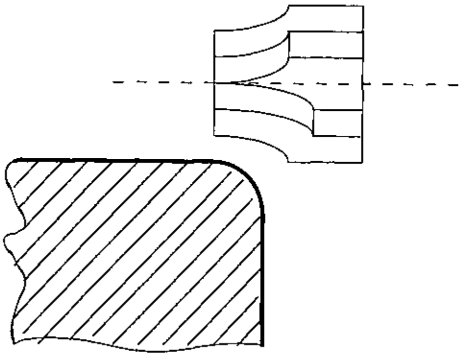
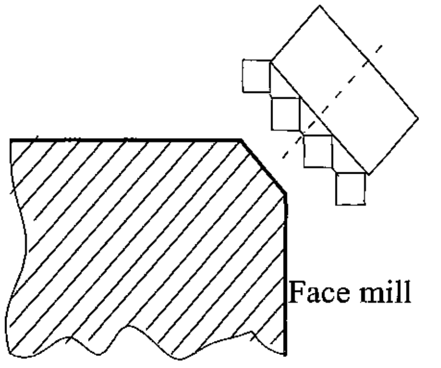


Fig. 4.30: Spotfacing is quicker and more economical than face milling for small flat surfaces.





Not this (Corner-rounding cutter) This

Fig. 4.31: Allowing a bevelled rather than a rounded corner can provide more economical machining.

4.6.1.2 Thread Milling

- Design parts that include hole diameters as large as possible because the cutter should not exceed one-third of the hole diameter.
- Avoid 90° flank thread forms which are impossible to mill.

4.6.1.3 End Milling

- Avoid end-milled slots deeper than the cutter diameter (fig. 4.32) [73].

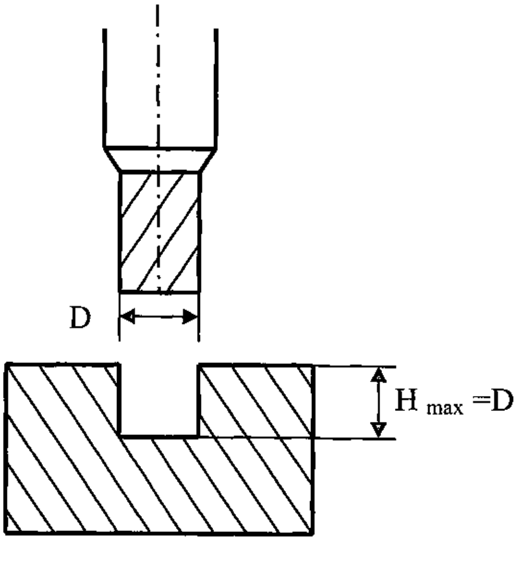


Fig. 4.32: End-mill slot in steel.

4.6.2 Planing, Shaping and Slotting

- Design parts which are not usually larger than 25m × 15m.

- Design parts with surfaces that are not shorter than 300 mm to machine on a planer except as part of a gang-machining operation.

- Design parts which provide machined surfaces in the same plane to reduce the number of operations required. Except for multitooled planers which can machine both surfaces simultaneously.

- Design parts not longer than 900 mm and with a minimum length of 13 mm for surface machining by shapers [74].

- Avoid designs with multiple surfaces which are not parallel in the direction of the reciprocating motion of the cutting tool since this would require additional setups.

- Design parts which allow a minimum size of hole in which a keyway, slot or other contour can be machined with a slotter or shaper of about 25 mm (fig. 4.33) [71].

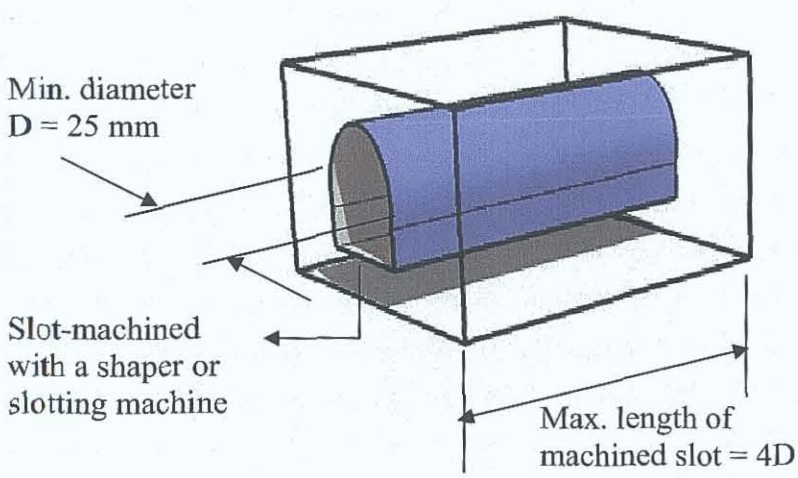


Fig. 4.33: The minimum-size hole in which a keyway, slot, or other contour can be shaper-machined is about 25 mm. Slots and contours should not be longer than 4 times the largest dimension of the opening or the hole diameter.

- Design parts that do not require contoured surfaces unless a tracer attachment is available and then specify gentle contours and generous radii as much as possible.

- Design parts which allow sufficient stock for a stress-relieving operation between rough and finish machining or if possible rough machine equal amounts from both sides. Allowance should be about 0.4 mm for machining.

- Avoid machined surfaces too close to an obstruction at the end of the cut. Shapers and slotters are able to cut within 6 mm of an obstruction or the end of a blind hole. A relieved portion should be allowed at the end of the machined surface (fig. 4.34) [73].

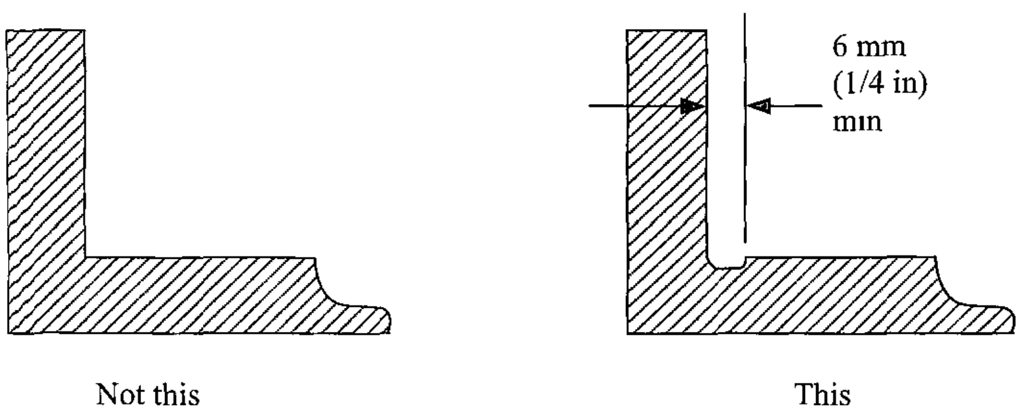


Fig. 4.34: Avoid machined surfaces too close to an obstruction at the end of the cut.

- Slots and contours should not be longer than 4 times the largest dimension of the hole (fig. 4.33) [73].

- Design parts in such a way that they can be easily clamped to avoid abrupt cutting force in planing and shaping and sturdy enough to withstand deflection during machining (fig. 4.35) [71].

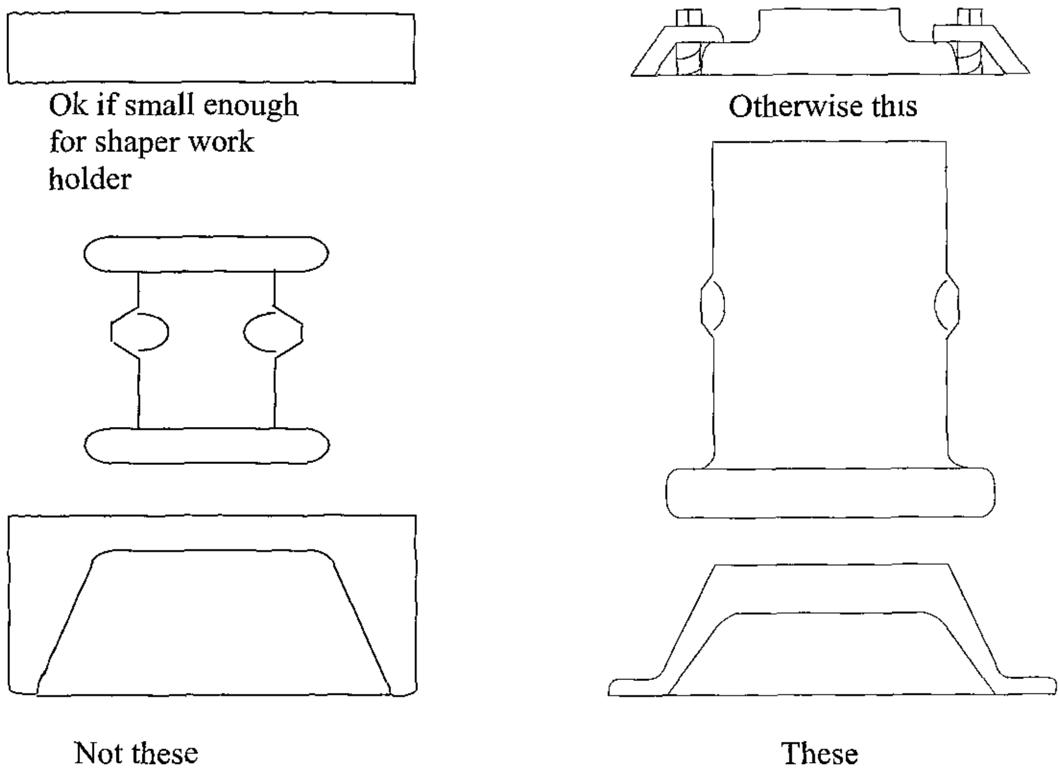


Fig. 4.35: Design planer-and shaper-machined parts to be sturdy enough to withstand cutting-tool forces and to be solidly clamped.

4.6.3 Broaching

- Parts should be rigid enough to withstand clamping and cutting forces during broaching operations.

- To minimise the setup time, such as tooling and holding fixtures, parts of similar operation should be designed in the same group.

4.6.3.1 Internal Broaching

- The design should consider that blind holes, sharp corners, dovetail splines, and large surfaces should be avoided. If splines or similar shapes are necessary in blind hole there should be a relief at the bottom of the broached area to allow the chip to break off (fig. 4.36) [73].

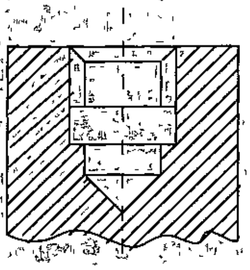


Fig. 4.36: Blind holes should have relief at the bottom of the broached area.

- To minimise tooth-edge wear and stress concentration points, sharp internal corners should be avoided. Chamfers are preferred but radii may be specified (fig. 4.37) [71].

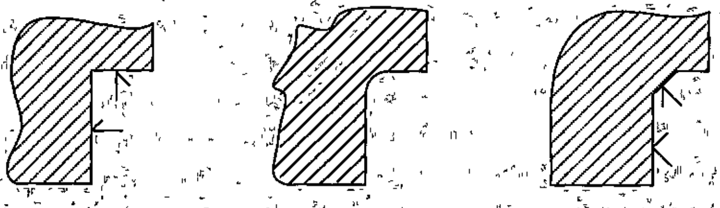


Fig. 4.37: Internal-corner design.

- The design should consider that symmetrically and irregularly shaped internal forms are usually broached by starting from round holes (fig. 4.38) [73].

Fig. 4.38: Irregularly shaped broached holes are started from round holes.

- It is preferable to design keyways to ISO specifications.

- Design pilot holes for internal keys of parts which are in the same centreline. A balanced shaped hole is preferable to prevent the broach from drifting to one side (fig. 4.39) [73].

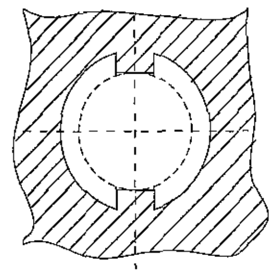


Fig. 4.39: A balanced shaped hole is preferable to prevent the broach from drifting to one side.

4.6.3.2 External Broaching

- Design parts that do not include relieves or undercut in the corners to simplify broaching operations of external surfaces (fig. 4.40) [71].

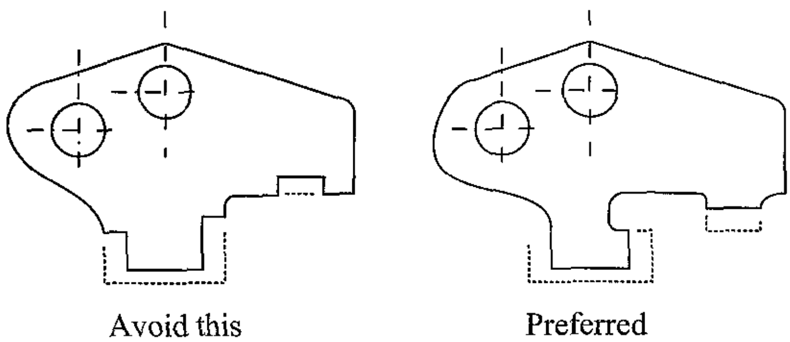


Fig. 4.40: Relieves or undercuts in the corners simplify broaching of external surfaces.

- Avoid sharp or narrow undercuts, if this is not possible they should be as shallow as possible.

- The design should consider that large surfaces should be broken into a series of bosses.

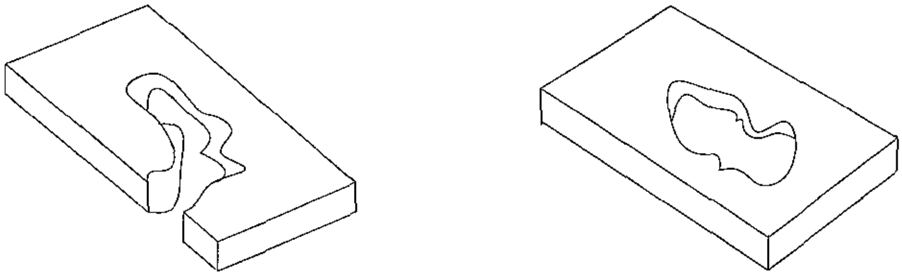
- Design parts that include chamfers rather than round corners.

4.6.4 Sawing

- Design parts with radii of contours that are as generous as possible. The minimum internal radius of contour-sawed surfaces depends on the blade width.

- Designs should consider the kerfs losses in contour band sawing. Kerf widths range from 0.8 mm to about 4 mm, depending on the cutting process, saw tooth set, speed and other factors.

- Avoid contour-sawed holes if possible. Since normal band-sawing practice involves an endless blade, it is necessary when sawing such shapes to predrill a hole, thread the blade hole through the hole, and weld the blade (fig. 4.41) [73].



This Not this

Fig. 4.41: The part on the right requires cutting and rewelding of the band-saw blade.

- Design parts that include sufficient stock for finishing operations since contour sawing is a rough machining process.

- Designs should consider that materials too hard for conventional contour sawing can be processed advantageously by friction contour sawing.

4.7 Design Guidelines for Abrasive Machining Processes

4.7.1 Grinding

- Designs should consider that non-hardened materials usually grind more rapidly than hardened materials.

- Grinding processes are economically justifiable for any production volume.

- Design parts in such a way that they can be held securely, either in chucks, magnetic tables, or suitable fixtures and work holding devices to protect distortions during grinding in thin and tubular work pieces.

- Designs should consider that hard materials, highly abrasive materials and fragile materials are suitable for grinding. Thin walls, interrupted surfaces (such as holes and keyways) are difficult to machine by other processes.

- To prevent the fill-up of pores of the grinding wheel during grinding, very soft materials (aluminium, copper) should be avoided.

4.7.1.1 Surface Grinding

- The design should consider that nonmagnetic materials are held by vices, special fixtures, vacuum chucks or double-sided adhesive tapes.

- To avoid frequent wheel dressing accurate form grinding design should be kept simple.

- The design should consider that, as much as possible, surfaces should be ground in one set up of the workpiece.

- Avoid openings in the surfaces because the grinding wheel tends to cut slightly deeper at the edge of an interrupted surface when very flat surfaces are required

- Avoid blind cuts, designs that force the wheel to be stopped during the cut or reversed with too little clearance provided.

- In order to prevent wheel loading and growth differences dissimilar materials should be avoided (when possible).

- Designs should consider the conditions required for minimum stock removal by grinding.

- Design parts in such a way that all the parameters on the drawings are indicated clearly.

4.7.1.2 External Cylindrical Grinding (Center-type grinding)

- Design parts in such a way that for better finish and accuracy keep the parts well balanced, and long slender designs should be avoided.

- Design parts in such a way that a long small-diameter part which causes deflection is avoided.
  Length/Diameter< 8 is best.
  Length/Diameter> 20 causes problem.

- Avoid grinding deep, narrow groves. Wheel dressing is difficult, and wheels wear is very fast.

- Design parts in such a way that interrupted surfaces which cause grinding problems and tend to be ground more deeply are avoided.

- Designs should consider that undercuts on facing surfaces are difficult for cylindrical grinding machines except for shallow degrees and it will reduce the accuracy of cylindrically ground parts (fig. 4.42) [73].

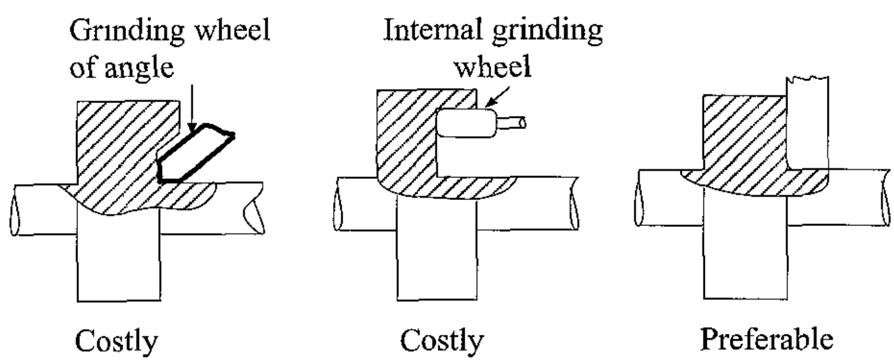


Fig. 4.42: Ground undercuts on facing surfaces are costly and should be avoided.

- Parts should be rigid enough to withstand deformations when held in a three jaw chuck.

- Avoid grinding sharp corners. Use fillet radii as large as possible. Even better, relief grooves could be used or the part could be machined by turning (fig. 4.43) [71].

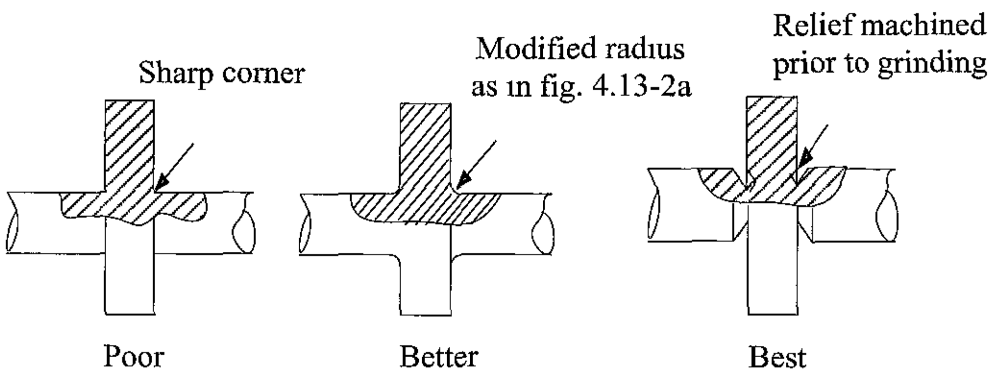


Fig. 4.43: The best practice is to machine or cast a relief at the junction of two surfaces before grinding.

- Designs should consider that for accurate cylindrical grinding, centre holes on work pieces held between centres should have an exact 60° angle and uniformity of shape.

- Designs should consider that profiles are better kept as simple as possible. Plunge type cylindrical grinding is only applicable for ground features that are of less width than the grinding wheel tool.

4.7.1.3 External Cylindrical Grinding (Centre-less grinding)

- As short pieces are more susceptible to having unspecified taper or concave or barrel-shaped surfaces, design parts in such a way so as to keep ground surfaces at least one diameter in length (if possible) to avoid problems.

- Designs should consider that parts with variable diameters, such as bolts, valve tappets, and distributor shafts, can not be ground by centre-less grinding.

- Parts with irregular shapes should be avoided if they do not have ground surfaces longer than the grinding wheel width unless the shape permits a combination of in feed and through-feed grinding (fig. 4.44) [73].

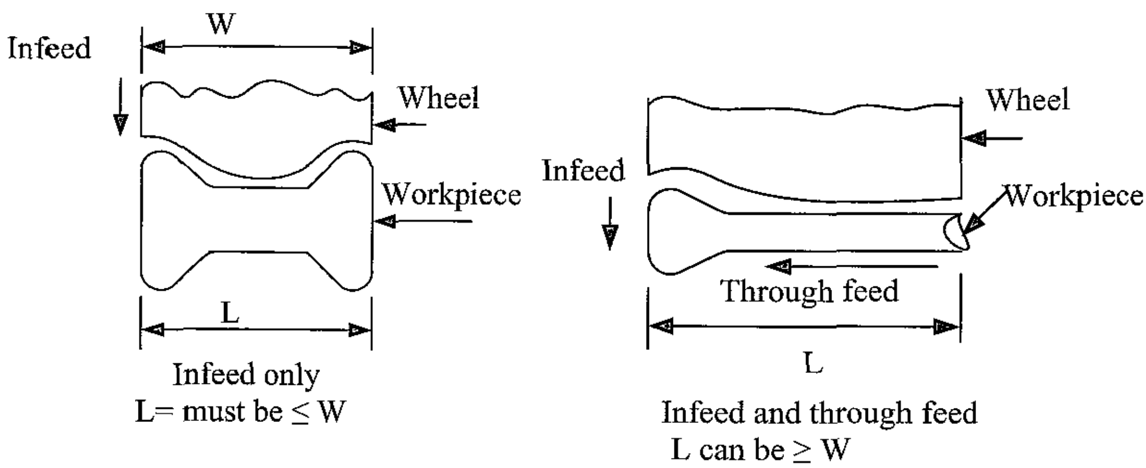


Fig. 4.44: Parts with irregular surfaces can not be longer than the width of the grinding wheel unless both infeed and through feed are used and the part is stepped in one direction as shown.

- Designs should consider that the largest diameter of the workpiece can be machined using through-feed centre-less grinding (fig. 4.45) [71].

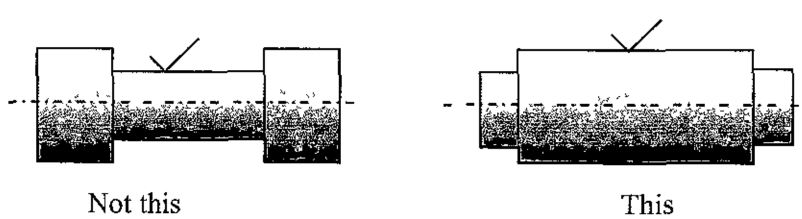


Fig. 4.45: Only the largest diameter of the workpiece can be through-feed centre-less-ground

- Avoid square, nearly square, or round ends if the end must be finished. The included angle of the pointed end should be 120° or less.

- The design should consider that to reduce wheel dressing and other costs in centre-less grinding (in feed method) the form should be kept as simple as possible.

- Design parts in such a way that they do not require keyways, flats, holes and other interruptions to the surface.

- Designs should consider that to prevent the tendency for a high spot and unbalanced conditions it is preferable to put flats on opposite sides of the part.

- Design parts in such a way that wheel dressing fillets and radii are avoided. If not possible keep them as large as possible (fig. 4.46) [73].

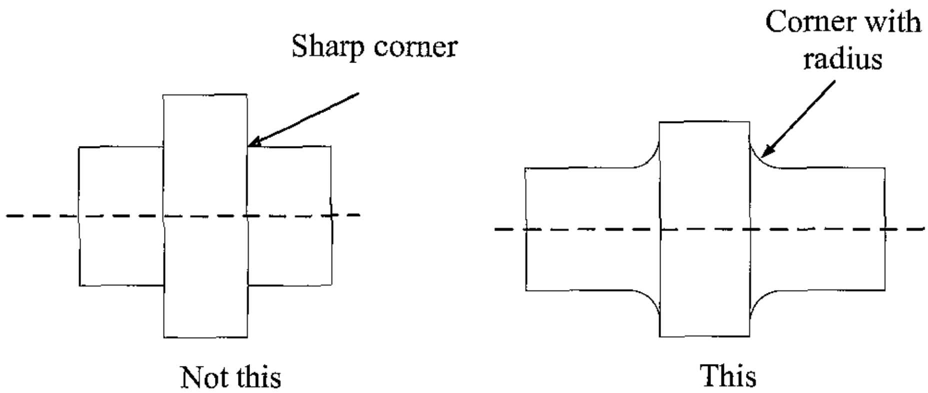


Fig. 4.46: Avoid grinding sharp corners.

4.7.1.4 Internal Grinding

- Design parts in such a way that prevents the increase in grinding time and the possibility of waviness and chatter. Deep, narrow holes should be avoided. Internal grinding is difficult if the hole Depth / Diameter > 6 (fig. 4.47).

- Designs should consider that axial interruption will incur a grinding-time penalty when the hole diameter is < 2 its length and the hole L/D > 3 [71].

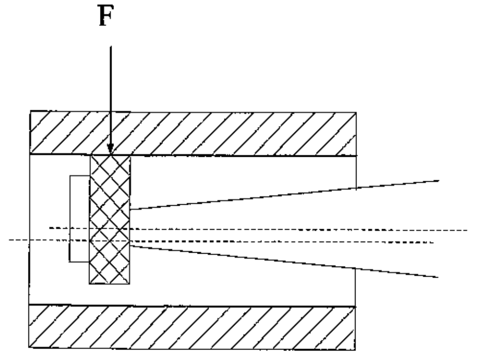


Fig. 4.47: Holes deeper than 6 times diameter and overly long-reach distances to the ground hole should be avoided unless the area is wide enough to provide rigid support for the wheel spindle.

- Designs should consider that face-clamping chucks are more forgiving of outside-diameter quality but require better face flatness to prevent a misshapen inside diameter unless either the inside diameter is very short or the part is very rigid. An area equal to 25% of the area to be ground internally is sufficient.

- Avoid sharp bottom corners of blind holes which take more time for grinding operations. A relief of at least 3-mm axial length will minimise straightness and taper problems (fig. 4.48) [73].

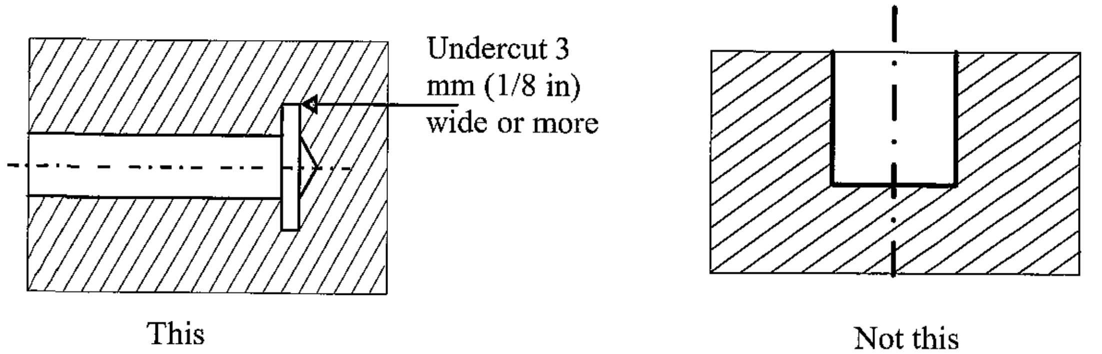


Fig. 4.48: Sharp bottom corner in blind holes should be avoided.

- Design parts in such a way as to minimise the tendency of the wheel to remove more stock in the vicinity of cross holes or to round corners of a keyway. Interruption should be avoided.

- The design should consider that for a pass of a reasonable wheel size, the entrance must be as large as possible and interference with the quill or spindle should be avoided.

4.7.1.5 Creep Feed Grinding

- Designs should consider that for improvement of surface finish and to keep temperature low, grinding wheels are mostly softer grade resin bonded with open structure (fig. 4.49) [74].

- Special features, such as high power (up to 225 KW), high stiffness (because of the high forces due to the depth of material removed), high damping capacity, variable and well-controlled spindle and work-table speeds, and ample capacity for grinding fluids should be included in design considerations.

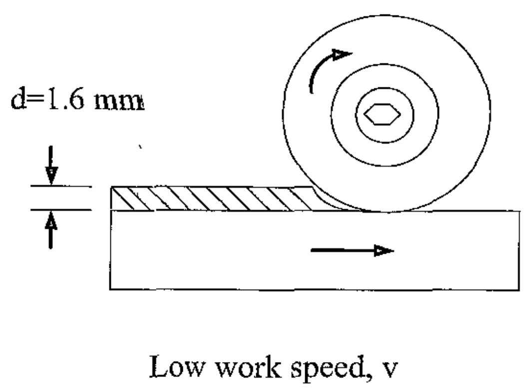


Fig. 4.49: Schematic illustration of the creep-feed grinding process.

4.8 Design Guidelines for Metal casting processes

- Design parts in such a way that they contain allowances for shrinkage during solidification as it causes induced stresses, distortion and reduces work piece dimensions compared with the size of the mould cavity (fig. 4.50) [71].

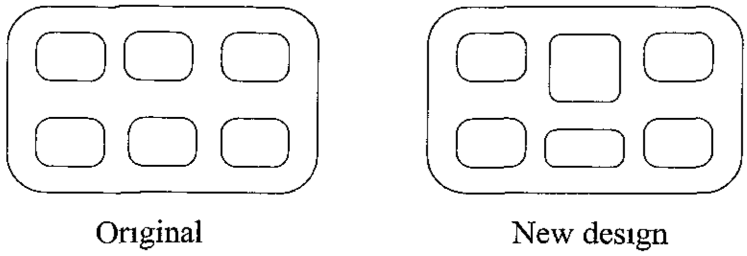


Fig. 4.50: Modification of design to avoid shrinkage cavity in castings.

- Designs should be protected against warping because of temperature gradient during cooling or poor surface finish because of uneven flow of metal during pouring. Large plain surfaces should be avoided.

- In order to prevent cracking and tearing during solidification of the metal sharp corners, angles, and fillets should be avoided. Fillet radii usually range from 3 mm to 25 mm (1/8 in to 1 in) which should be selected to reduce stress concentrations and ensure proper liquid-flow during the pouring process (fig. 4.51) [74].

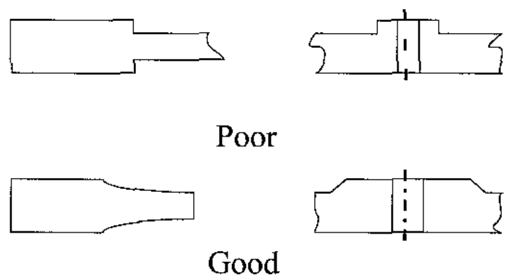


Fig. 4.51: Suggested design modifications to avoid defects in castings.

- Design should be in such a way that the parting line can be on a flat plane rather than contoured, which is more economical and more accurate. The parting line separates the two halves of the mould of the desired part. The location of the parting line is important because the greater the degree of contouring, the greater the problems and costs.

- In order to remove each pattern easily without damaging the mould, the pattern must have some degree of taper, or draft.
  Draft range from 5 mm/m to 15 mm/m (1/16 in. /ft to 3/16 in. /ft).
  Draft angles usually range from 0.5° to 2° (fig. 4.52) [73].

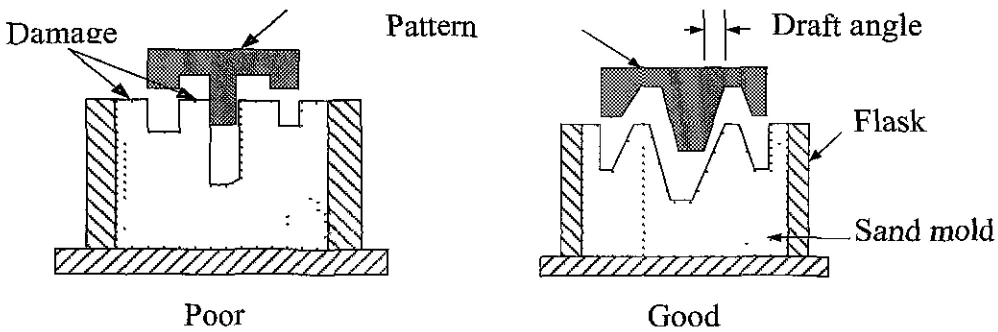


Fig. 4.52: Taper on patterns for ease of removal from the sand mould.

- Design parts with liberal tolerances. The permissible variations in the dimensions of a part depend on the particular casting process, size of the casting, and type of pattern used.
  Tolerances range of ± 0.8 mm (1/32 in.) for small casting.
  Tolerances range of ± 6 mm (1/4 in.) for large casting.

- In order to avoid unnecessary problems and promote sounder casting it is best to have sections and walls as uniform as possible in thickness. Problems occur when the wall thickness is less than 6 mm in all metals, that is why it is cheaper to pay for an increased section size than to pay an increased price required to cover foundry scrap losses.

- Design parts in such a way that the interior walls and sections are 20% thinner than the external walls since they cool slowly and to reduce thermal and residual stresses which minimise metallurgical changes (fig. 4.53) [71].

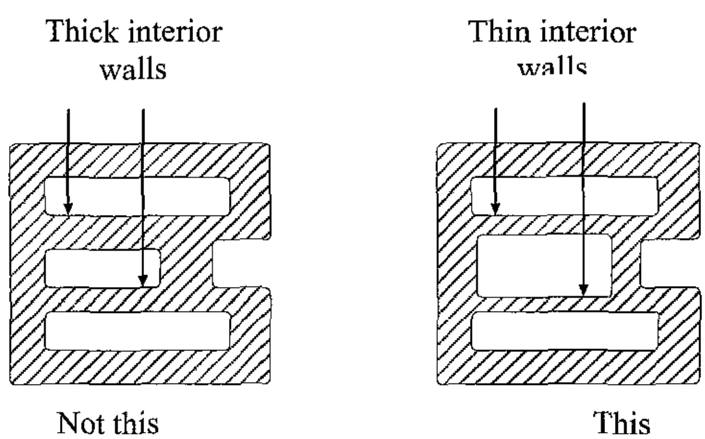


Fig. 4.53: Interior walls should be 20% thinner than exterior walls since they cool more slowly.

- In order to maintain additional finishing operations a stock allowance must be added to surfaces which are to be machined during designing of parts. Machining allowances, which are included in pattern dimensions, depend on the type of casting and increase with the size and section thickness of castings. Allowances usually range from:
  about 2 mm to 5 mm for small castings.
  to more than 25 mm for large castings.

- Designs should consider that small holes are usually cheaper and more satisfactory to drill than mould or core. Holes less than 19 mm in diameter are cheaper and better if drilled after casting (fig. 4.54) [71].

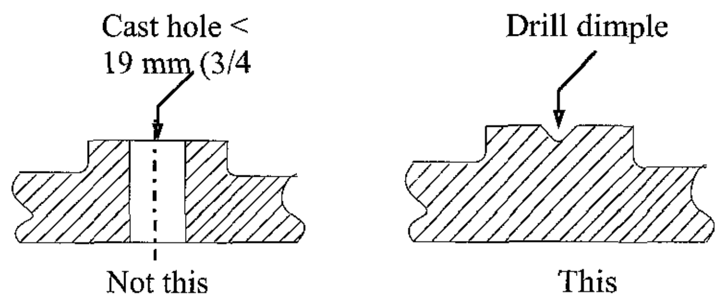


Fig. 4.54: Holes less than 19 mm in diameter are cheaper and better if drilled after casting.

- Designs should consider that through holes and pockets are more straightforward and economical in sand mould casting. Sand mould casting reduces the cost of the casting by saving material.

- Pockets that are much deeper than their width can be drawn with high-quality pattern equipment in shell mouldings (fig. 4.55) [73].

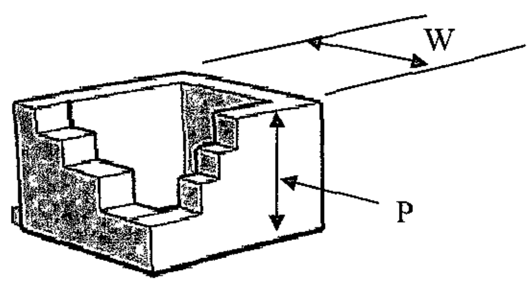


Fig. 4.55: Design rules for the correct portions of rectangular.

- In the case of expandable pattern casting, it should be taken into account that the flow of molten metal is basically laminar with Reynolds numbers in the range of 400 to 3000 and the estimated velocity in the range of 0.1-1.0 m/s.

- Design parts in such a way that in plaster mould casting, wall thickness, inserts, markings, draft, holes, and machining allowance should be taken into consideration [71]

Walls with projected areas:	Minimum thickness:
Up to 650 mm^2 (1 in^2)	1-mm (0.040-in)
Above 650 mm^2 to 1950 mm^2 (1 to 3 in^2)	1.5 mm (0.060 in)
Above 1950 mm^2 to 9750 mm^2 (3 to 15 in^2)	2.4 mm (0.090 in)

Draft angle 1/2° or more for outside surfaces
Draft angle 1 to 3° for inside surfaces (at least)
Maximum temperature 1200°C (2200°F)

- In order to get good dimensional accuracy and surface finish over a wide range of sizes and intricate shapes Ceramic-mould casting is applicable and all other design considerations are as for plaster mould casting.

- In Investment casting, minimum wall thickness, flatness and straightness, radii, curved surfaces, parallel sections, key and keyways, holes, blind holes, through holes, drafts, screw threads, and undercuts should be taken into account during the design of parts.

- In pressure casting (also called pressure pouring or low-pressure casting) the effect of pressure should be take in consideration.

- Design parts in such a way that, in Die casting, wall thickness, ribs and fillets, drafts, holes, core slides, threads, inserts, machining allowance, surface design, die sinking economics, and integral assembly are taken in consideration.

- In die casting the ejector pin locations should be taken into account and preferably be in reference with the die caster.

- Die casting is a high pressure (1000kg/cm^2) fluid injected process which takes 1-2 months die set-up time and has production rates of 20-200 pieces/hr-mould. A single mould produces over 500,000 castings during its productions life time.

- To avoid stress concentrations, generous radius should be specified in Centrifugal casting. The necessary centrifugal force should also be considered.

- For True centrifugal castings, cylindrical parts ranging from 13 mm to 3 mm and 16 m long with wall thickness 6-13 mm are to be considered.

4.9 Design Guidelines for Forming and Shaping Process

4.9.1 Rolling

- Design parts in such a way that the radii of both inside and outside corners should be as generous as possible. One stock thickness is minimum and 2 times stock thickness is preferable (fig. 4.56) [73].

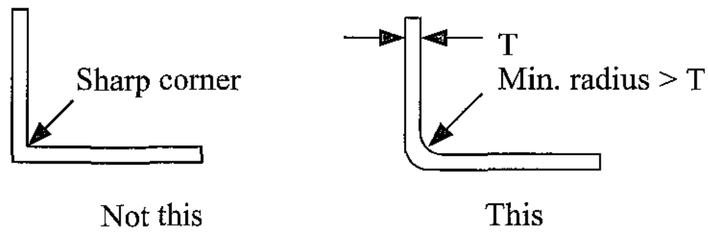


Fig. 4.56: The minimum bend radius for roll- formed components is one stock thickness, but 2 times stock thickness is preferable.

- Design parts which comprise an entrance and exit flare distortion at each end, whenever minimum length limitations are not possible (flare amounts to about 1.3 mm and extend 75 mm from each end).

- Parts shorter than 3 times the centreline spacing of rolls of the machine employed will not feed or form satisfactorily.

- Design parts with minimum length 3 times of the stock thickness.
- Avoid blind corners if precise bends are needed. Contact with both sides of the stock with rollers is more accurate than blind corners (fig. 4.57) [71.

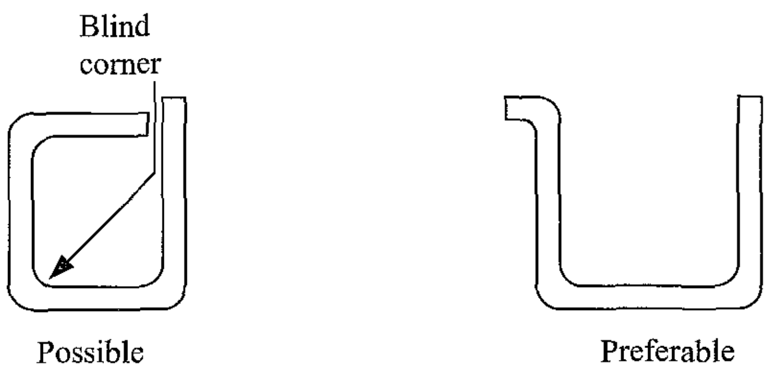


Fig. 4.57: Blind corners should be avoidable if possible.

- Vertical sidewalls should be avoided (fig. 4.58) [73].

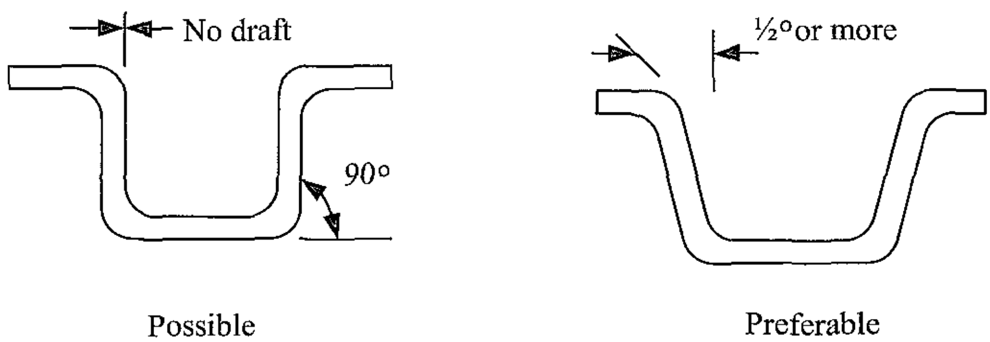


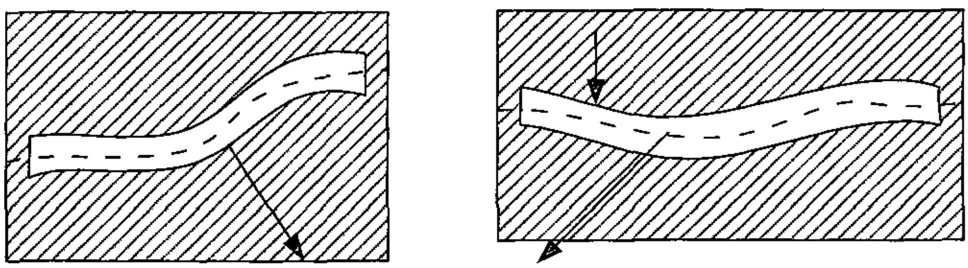
Fig. 4.58: Avoid exactly vertical sidewalls.

- In flat rolling the higher the friction and the larger the roll radius, the greater the maximum draft and reduction in thickness.

4.9.2 Forging

- Designs should be in such a way that for forged pieces that are produced in two-part impression dies, the parting line, the draft, the presence of ribs, bosses, recesses and webs and the machining allowance are taken into account.

- To avoid high side- thrust forces on the dies the parting line should be in one plane (fig. 4.59).



Not this This

Fig. 4.59: Preferable design of parting line.

- It should be taken into account that the angle of the surface parting line should not exceed 75° from the principal parting line. Much shallower angles are desirable.

- Design parts which include a minimum draft angle of 0° ± 0.5°for high tolerance and 1° ± 0.5° for standard draft angles. For aluminum and brass low draft and no draft forgings are allowed.

- To avoid process defects the rib thickness should be equal to or less than the web thickness. In general, the ratio of rib height to thickness is 6:1.

- Avoid small fillet radii as a sharp internal die is required which causes rapid wear, increases the possibility of break, and the metal flow is restricted.

- In order to avoid forging difficulty thin web and deeper ribs should be debarred (fig. 4.60) [73].

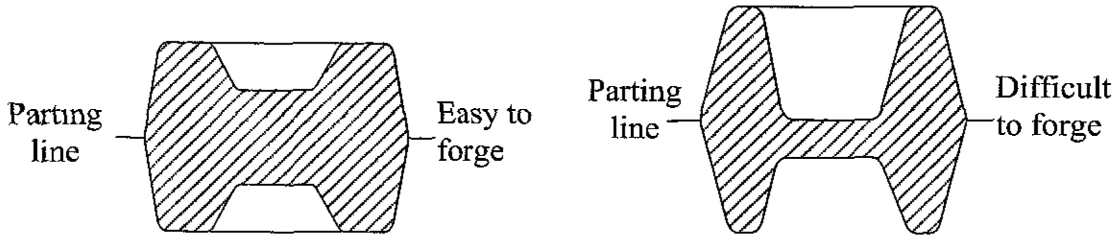


Fig. 4.60: As the web becomes thinner and the ribs become deeper, forging difficulty increases.

4.9.3 Extrusion

- Design parts with generous radii which is advantageous for both internal and external corners.

- Avoid sharp corners whenever possible. If necessary the angle should be as large as possible and always more than 90°.

- Variations from flatness of long sections are reduced by adding ribs to the sections.

- Holes in nonsymmetrical shapes should be avoided with steels and other less extrudable materials (fig. 4.61) [71].

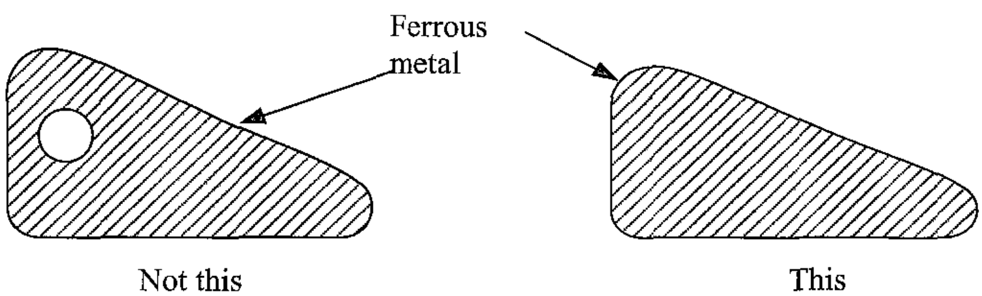


Fig. 4.61: Nonsymmetrical shapes of holes are avoidable.

- Avoid abrupt changes in section thickness with all metals but particularly with steel and less extrudable metals (fig. 4.62) [73].

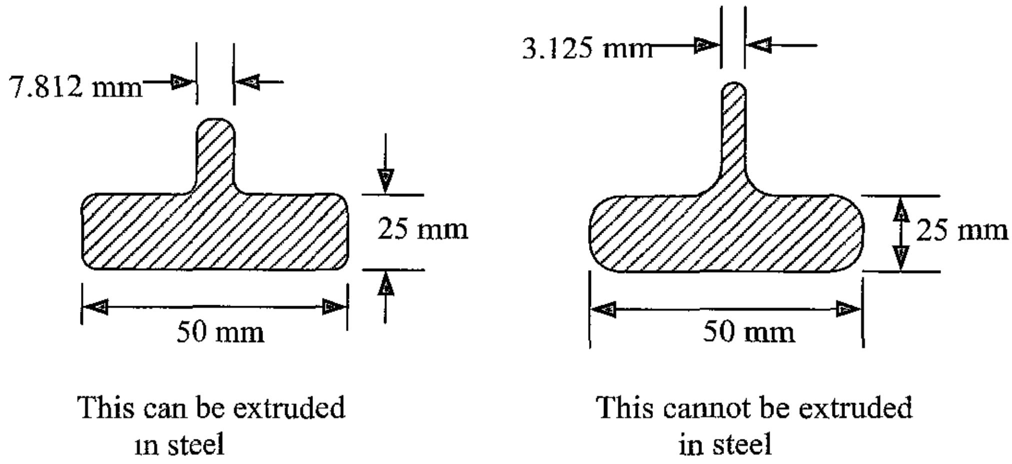


Fig: 4.62: Avoid abrupt changes in section thickness.

- To provide sufficient strength in the tongue portion of the extruded die the indentation depth relative to its width should be taken in account.
  For steels, the maximum indentation depth is 1 width.
  For copper alloys, the maximum indentation depth is 1.5 widths.
  For magnesium and aluminum alloys, the maximum indentation depth is 3 widths.

- Consider the length to thickness ratio of the part. The ratio of length to thickness of any segment should not exceed 14:1. For magnesium it can be 20:1 (fig. 4.63) [71].

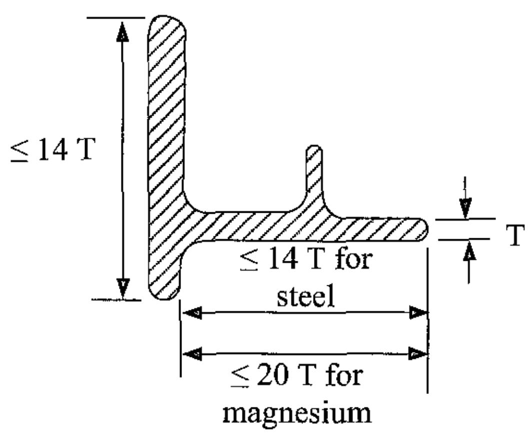


Fig. 4.63: Right thickness ratio.

- Sections walls should be balanced, especially with hollow sections.

- To avoid unbalanced stresses and warpage, symmetrical cross sections are preferable to Nonsymmetrical designs.

4.10 Design Guidelines for Sheet Metal Process

4.10.1 Bending

- Design parts with the gentlest and shallowest bend. Large-radius bends with less severe angles are more economical than tight bends (small radii) and large-angle bends.

- In order to avoid slower and more expensive bending, the design should be in such a way as to contain larger bend radii, which are easier to manufacture without a mandrel.

- Design parts with the minimum bend radius of 3 times sheet thickness in which bending can be performed without cracking on the outer surface.

- It is easier to make a tight bend if the part is bent 45° than it is if the part is bent 120°.

- Design parts which allow straight lengths between multiple bends in more than one plane minimum 1 or 2 times the diameter of the metal (fig. 4.64) [73].

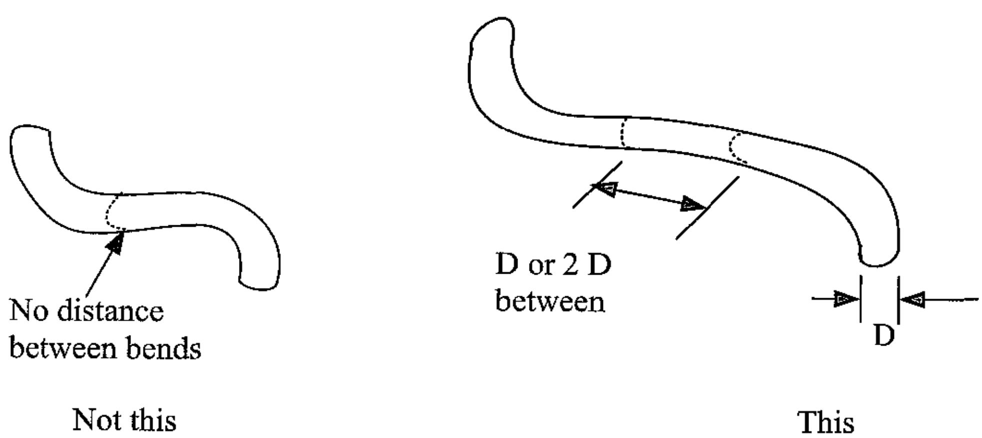
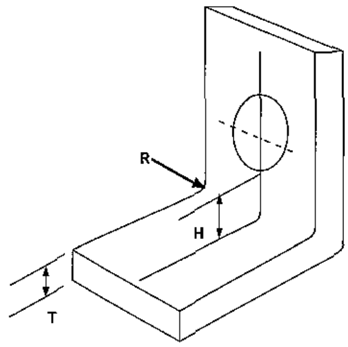


Fig. 4.64: Allow a straight length between bends.

- To avoid distortion of the holes design parts which contain a minimum spacing between the lowest edge of a hole and a bend surface of 1.5 times the sheet thickness plus the bending radius (fig. 4.65) [73].



$H_{min} = 1.5.T+R$

Fig. 4.65: Corrected hole design in bending.

- The design should consider that the final bend angle after spring back is smaller and the radius is larger than that of the bending tool.

- Designs should consider that it is difficult to align the holes if they are punched before bending. Instead (a) the holes can be punched (drilled) after bending (b) one of the holes can be oversized or oval (c) or pilot holes that align the strip symmetrically can be used (d) pilot holes assure that the blank is centred in the forming die (fig. 4.66).

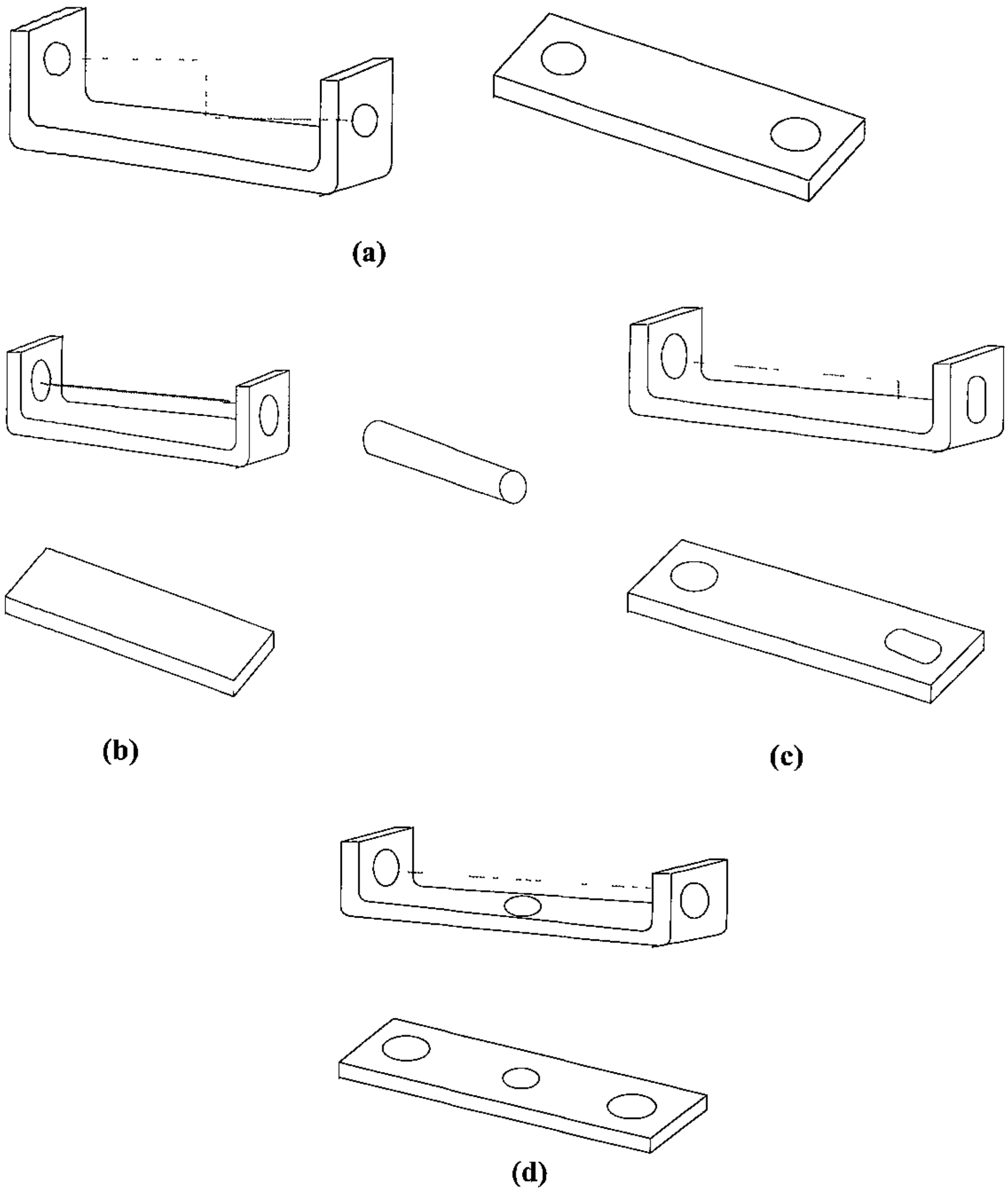


Fig. 4.66: Alignment of hole in sheet bending.

- The gain direction of the material should be taken into consideration. Parts should be aligned on the sheet (strip) so that the direction of the maximum stress coincides with the grain direction.

- In order to perform more sophisticated bending operations higher bendability metal can be used. Bendability increases with ductility.

4.10.2 Punching (piercing)

- If possible, avoid designs that require sharp corners, both internal and external, of punches or die. Sharp corners tend to break down prematurely, wear faster, and have larger burrs, rougher edges of the blanked part in the area of the corner and are stress concentrators. The minimum corner radius is 0.5 sheet thickness but never less than 0.8 mm.

- Design parts in such a way that a punched hole diameter is not less than the stock thickness, otherwise, the hole can be oversized.

- Avoid designs which require space between two holes less than 2 times the stock thickness. 3 times thickness is preferable from a die-strength standpoint.

- In order to prevent part bulging in the edge of the area adjacent to the hole, the design should be in such a way that the minimum distance between a hole edge and the adjacent edge of the blank is at least the stock thickness. 1.5-2 times thickness is preferable (fig. 4.67) [73].

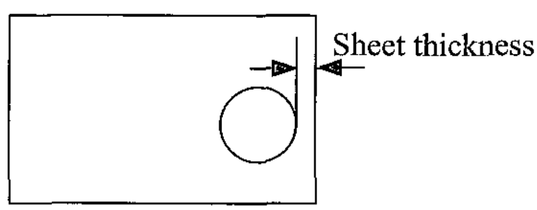


Fig. 4.67: Minimum distance between the hole and edge should be sheet thickness but 1.5-2 times are preferable.

- If possible, pierce a hole before forming as it is less costly than a secondary operation.

- Avoid designs of long, narrow projections which are subjected to distortion and require thin, fragile punches.

- In order to prevent tooling costs specify round holes (if possible) instead of holes of square, rectangular, or other shapes. Round punches are cheaper.

4.10.3 Blanking

- In order to perform fine blanking operations the sheet is first locked tightly by a V-shaped stringer close to the die's perimeter before shearing takes place. Clearance between the punch and die is reduced (about 1% of the sheet thickness) [71].

- Design fine blanked parts with round corners to avoid tears in the material over the shear zone. The minimum radius depends on the corner angle, material thickness and type of material. Generally [73]
  Obtuse angles: radius 5-10% of material thickness.
  Right angles: radius 10-15% of material thickness.
  Acute angles: radius 25-30% of material thickness.

- Designs should be in such a way that small parts with the same thickness and shape can be made from a piece of stock left over from blanking of large parts which saves raw material.

- In order to provide better nesting of blanks and improved utilisation of material proper placement of the part along the sheet is required as it affects the volume of the scrap and necessary instrumentation.

- Designs should be provided for slots in fine blanked parts such that:
  Minimum width of the slot is equal to 0.6 times the thickness.
  Minimum distance between the slot and the edge of the parts is equal to 0.6 times the thickness.
  Maximum length of the slot is equal to 12 times the width of the slot [73].

- Designs should be in such a way that the width of the tooth (forms for gears, ratchets, etc.), on the pitch circle radius is 60% of material thickness produced by fine blanking.

4.10.4 Deep drawing

- Durıng the desıgn of parts in deep drawing, the characteristics of materials should be taken into consideration, such as high ductility, low strain, high tensile strength and uniform grain size.

- Avoid deep drawing operatıons in small lots, it ıs more sophisticated, more expensıve toolıng is required and more development work and tıme are necessary than for simply bent stamping.

- Shallow drawıngs may be produced without blank holding, suitable for low-volume production. The maxımum depth/diameter ratio ıs 10% (fig. 4.68) [71].

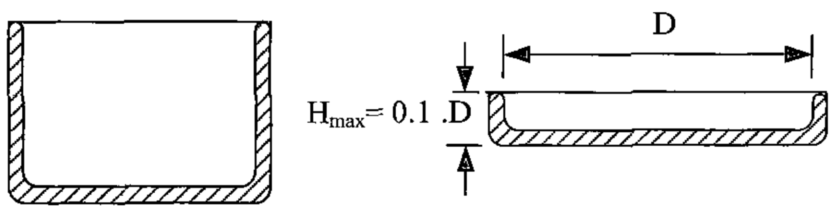


Fig. 4.68: The maximum depth/diameter ratio should be 10%.

- Designs should be in such a way that do not allow tapered-wall shells and/or flanged shells because these are much more expensive than cylindrıcal ones (fig. 4.69).

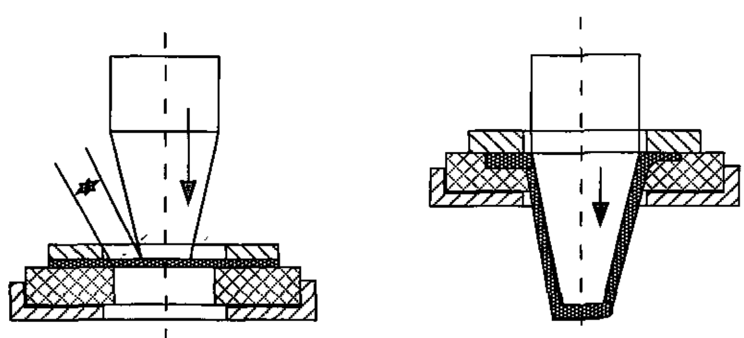


Fig. 4.69: Avoid tapered-wall shells.

- Sharp corners should be avoided in the bottoms of the drawn parts. A minimum radius of 4 times stock thickness is acceptable.

- Because of variations in wall thickness the dimensions of both inside and outside diameters can not be controlled.

- Rectangular boxes should be specified with corner radii to be a minimum of 0.25 times the depth drawn [71].

- Avoid design parts with countersinking and counter boring unless they are really necessary because these features are costly as they require additional tooling.

4.10.5 Spinning

- A taper angle should be used if the part has cylindrical sides and a chuck.
  For wood chuck taper angle is 2° or more.
  For steel chucks taper angle is 1/4°.

- Designs should consider that outside beads of the part are more economically spun than inside beads.

- Design parts with conical and curvilinear shapes which are suitable for conventional spinning. Part diameters may range up to 6 m.

- Avoid sharp corners which cause thinning of the stock. Blended radii and fillets are preferable. Minimum radius is 6 mm although 3 mm usually causes no problem (fig. 4.70) [73].

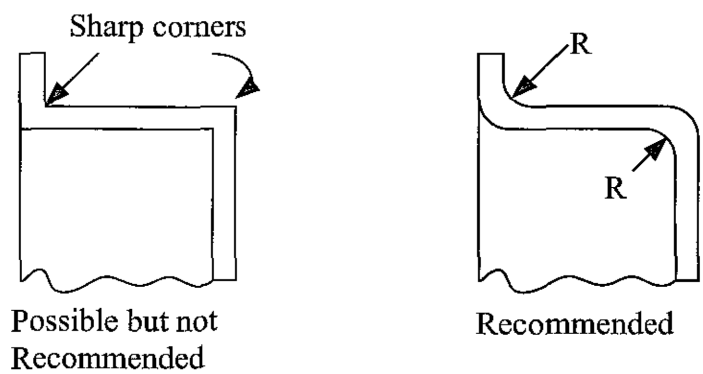


Fig. 4.70: Avoid sharp corners, if possible.

- In order to control repeated operations and annealing, deep cylindrical designs should be avoided. A spinning ratio (depth/diameter ratio) of less than 1:4 is preferable. Spinning ratios of different types of design are:

  Shallow: less than 1:4

  Moderate: 1:4 to 3:4

  Deep: 3:4 to 5:4

- In conventional spinning some thinning of the material is normal. Specifying material 25 or 30% thicker than the finished-part thickness is usually sufficient to allow for such reduction in wall thickness.

- Avoid designing parts with internal flanges and other configurations of reentrant shapes which are more costly to produce as the operation requires special, more complex chucks or spinning.

- An axisymmetric conical or curvilinear shape is suitable for shear spinning while maintaining the part's maximum diameter and reducing the part's thickness. Parts up to 3 m in diameter can be formed by shear spinning.

- Design parts which do not require reverse-form designs since they require additional operations with separate chucks (fig. 4.71) [71].

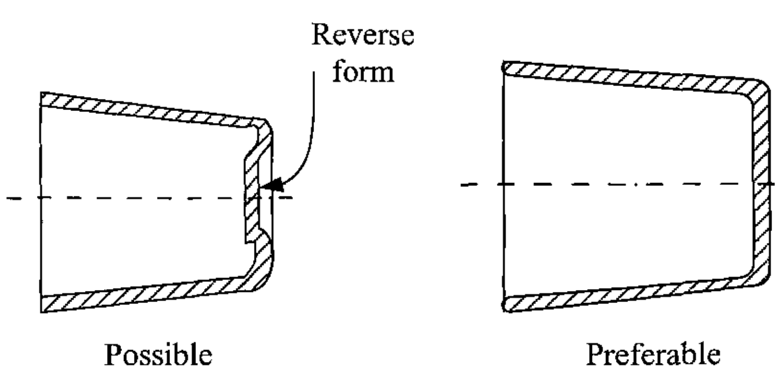


Fig. 4.71: Avoid reverse bends, if possible.

- Design parts with a cone angle of 5°, which provide rigidity if bottom rigidity is important and flatness is not required.

4.10.6 Forming

- Avoid design with sharp contours and reentrant angles. Stretch forming is more suitable to parts with shallow, gentle bends.
- Dies for stretch forming operations are generally made of zinc alloys, steel, plastics, or wood.
- Design parts in such a way that in stretch forming the blank is a rectangular sheet rather than round, triangular, trapezoidal, etc.
- Avoid designs with deep forming in the direction of the free edges which are not feasible in stretch forming [74].
- Avoid the design of nonconcentric shapes in explosive forming which requires costly tooling and control of process conditions.
- Design of complex shapes in smaller parts is often practicable in explosive forming but for large parts keep shapes as simple as possible. Steel plates 25 mm thick and 3.6 m in diameter have been formed by this method [74].
- Avoid sharp corners which cause stress concentration in the forming die and shortene die life.
- Avoid designs which contain slots or other cutouts in the area to be formed since it to has to be electrically formed.
- The higher the electrical conductivity of the workpiece, the higher the magnetic forces.

- In rubber forming, the die should be made of a flexible material, such as a rubber or polyurethane membrane, because of their resistance to abrasion, resistance to cutting by burrs or sharp edges of the sheet metal, and have a long fatigue life. Estimated pressure is usually of the order of 10 MPa (1500 psi).

- Rubber forming is suitable for parts with low cost tooling. It results in flexibility and ease of operation, low die wear, no damage to the surface of the sheet, and is capable of forming complex shapes.

- Super plastic forming offers the advantage of low tooling costs, because of the low strength of the material at forming temperatures, the ability to produce complex shapes, weight and material savings, and a virtual absence of stress within the formed parts.

- Super plastic forming improves productivity by eliminating mechanical fasteners and produces parts with good dimensional accuracy and low residual stresses.

- Designs should consider that in peen forming the surface of the sheet is subjected to compressive stresses, which tend to expand the surface layer.

4.11 Design Guidelines for Finishing Processes

4.11.1 Coated Abrasives

- Design should consider that coated abrasives which have a much more open structure are used extensively in finishing flat or curved surfaces of metallic and nonmetallic parts [74].

- For high rate material removal, coated abrasives are used with a belt. Belt speed is usually in the range of 700-1800 m/min.

4.11.2 Honing

- Design parts with no keyways, ports, undercuts, and other surface interruptions if possible because they cause problems on the honed surface. Wherever they are essential they should be kept as small as possible so that the abrading elements can pass with minimal effect.

- Designs should consider that the abrading elements must overrun the ends of the bore by an amount equal to one-fourth to one-half of the length of the abrasive in case of inside diameter honing (fig. 4.72) [73].

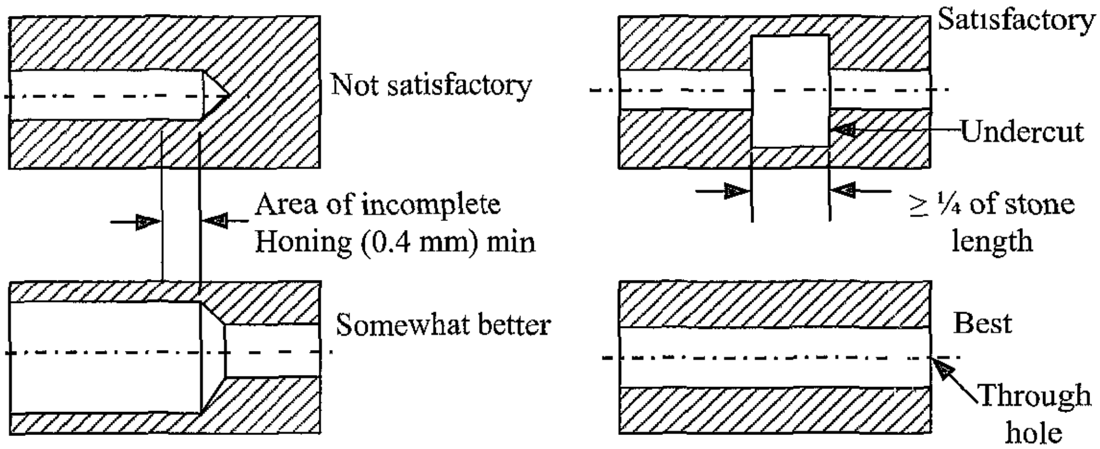


Fig. 4.72: Design recommendations for internal cylindrical surfaces which are honed.

- Design parts which are rigid enough to withstand the radial force with a reciprocating axial motion because the honing tool, mounted on a mandrel, rotates in the holes.

- Design parts in such a way that projections such as shoulders, bosses, spherical surfaces, flat surfaces, and outside diameter, are avoided [73].

4.11.3 Lapping

- In order to apply lapping operations in any shoulders, projections, or other interruptions, projected interferences should be avoided when the lap is moved back and forth across the work surface.

- The design should consider that to make unobstructed contact with the lap and the machine table the two surfaces should extend beyond other surfaces of the workpiece (4.73) [73].

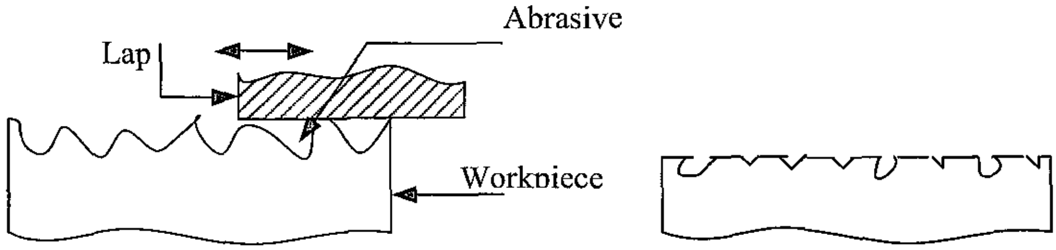


Fig. 4.73: The lapping process.

- The design should consider that curved surfaces, such as spherical objects, glass lenses and running-in mating gears can be done by lapping. Lapping pressures range from 7-140 KPa (1-20 psi) depending on the hardness of the workpiece.

4.11.4 Polishing and Buffing

- Design parts in such a way that for belt polishing, inside or outside sharp corners, deep recesses and compound curves are avoided.

- To prevent snagging or cutting of the polishing wheel or belt, parts with hooked edges or sharp projections should be avoided [74].

- To maintain free access of the wheel or belt to the surface for polishing bosses, handles, and other obstructions in the surface should be avoided.

- Avoid large surfaces (if possible) which provide uniformly polished surface as otherwise it is difficult to polish uniformly.

- Design parts which are easy for holding by hand. Fixtures are preferable for small parts and for those difficult to hold for fine polishing.

4.11.5 Barrel polishing

- Avoid designs containing small holes, slots, or recesses of parts which are difficult to barrel polish because they can trap pieces of the tumbling medium either directly or by bridging [74].

- The design should consider that large holes or shielded areas are not polished well in the barrel polishing process because the abrasive motion of the medium is affected in such spaces.

- Avoid designs containing springs and other wire or strip parts which are susceptible to interlock and tangle during barrel polishing.

- A secondary operation is required for effective barrel polishing of large flat surfaces.

- Stock removal in barrel polishing is normally of the order of 5 μm.

4.11.6 Electropolishing

- Designs should consider that irregular shapes are suitable for electropolishing because electrolyte attacks projections and peaks on the workpiece surface at a higher rate, thus producing a smooth surface [74].

- In order to get uniform appearance, electropolished and mechanically polished surfaces should not be placed together.

- Designs should consider that laser polishing works in more prominent surfaces of holes, recesses, and slots of work pieces than electropolishing.

- Designs should considered that specially shaped and placed electrodes can be used for fine finishing of a surface (if essential), but it is more costly.

- In electropolishing, the removal of 0.025 mm in the 0.2-1.2 μm range reduces surface roughness by about one-half.

4.11.7 Polishing using magnetic fields

- Designs should consider that for lower polishing times, no defect or few defect surfaces and economical processing magnetic float polishing of ceramic ball is suitable.

- Design parts to be rigid enough to withstand clamping and rotating forces because in magnetic-field-assisted polishing, magnetic poles are oscillated and they introduce a vibratory motion to the magnetic-abrasive conglomerate. For example: Bearing steels of 63 HRC have been mirror finished in 30 seconds by this process [74].

4.12 Summary of This Chapter

In this chapter, DFM rules have been explained broadly. Rules at the higher level of the hierarchical system are applied to more generic manufacturing features, and more specific rules are applied to more detailed features. This system leads to a minimised number of rules and helps to avoid repetition of rules in different applications. Design for manufacture rules play an important role in cooperation between the designer and the manufacturer in the design stage.

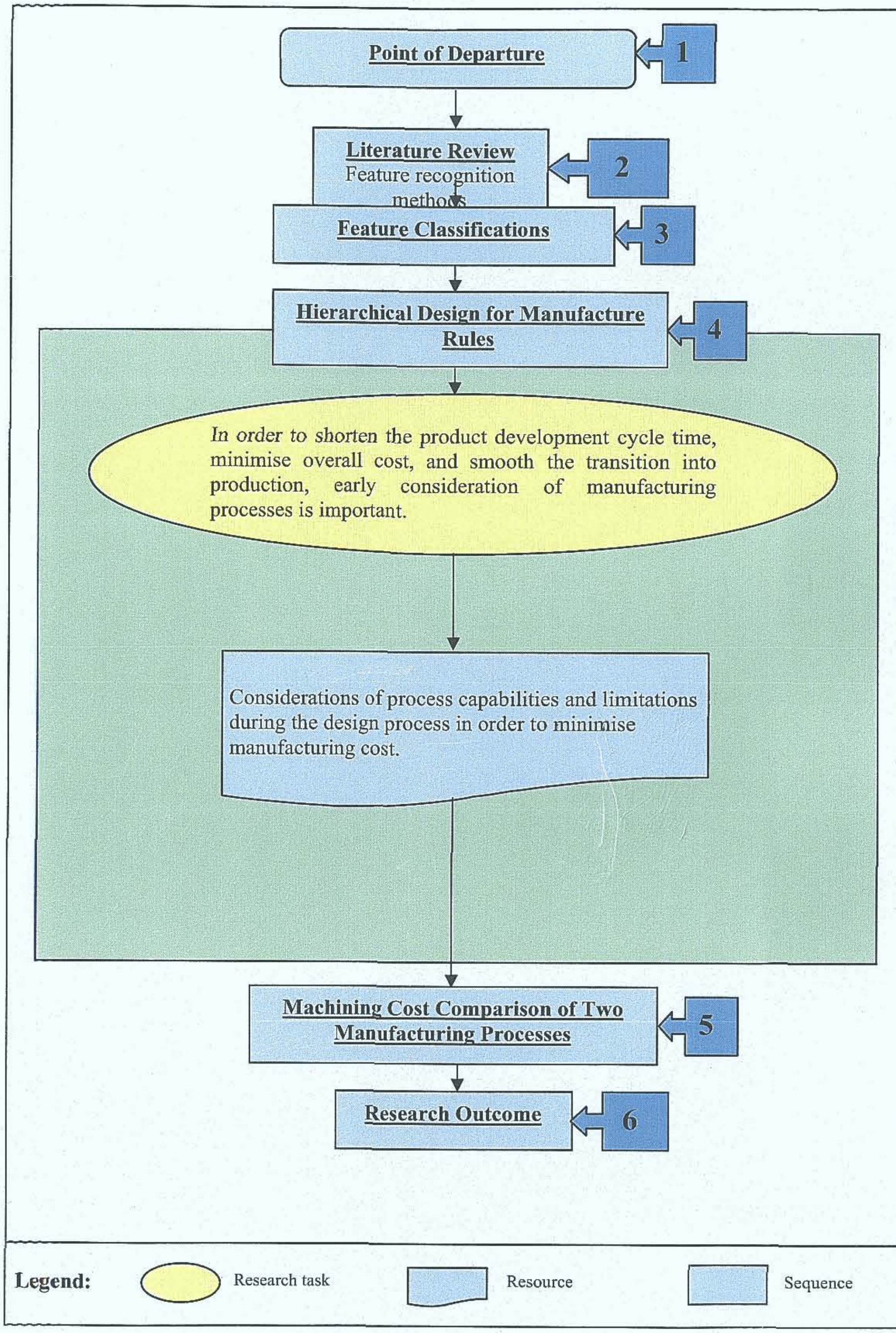


Fig. 4.74: Summary of the research process in chapter four.

CHAPTER 5 MACHINING COST COMPARISON OF TWO MANUFACTURING PROCESSES

5.1 Introduction

The machining process used plays a significant role in determining product quality, total manufacturing cost and impact on the environment. However, simultaneous improvement of cost, quality and environmental impact is sometimes possible. For example near-net-shape casting potentially eliminates some machining operations and their corresponding cost.

In general total cost depends on two factors, variable cost and fixed cost. Variable cost includes casting, labour (such as milling and drilling operations), lubricants, tooling and materials. On the other hand fixed cost includes initial investment, setup and overhead cost. In this section machining cost calculation of two machining process (drilling and milling) have been taken into consideration. To determine unit level cost for milling and drilling machining operations two factors, feature parameters and cutting parameters, are considered. Feature parameters include hole depth and diameter, end mill diameter, drill length. Cutting parameters include spindle speed, feed rate and depth of cut.

5.2 Cutting Condition

The three factors, cutting speed, feed rate and depth of cut, are known as cutting conditions. Cutting conditions are determined by the machinability rating of the material. Machinability is the comparison of materials based on their ability to be machined. From machinability ratings we can derive recommended cutting speeds.

5.2.1 Cutting Speed

Cutting speed is the speed at the outside edge of the tool as it is cutting. This is also known as surface speed which is directly related to surface area. If two tools of different sizes are turning at the same revolutions per minute (RPM) rate, the larger tool has a greater surface speed. Surface speed is measured in surface feet per minute (SFPM).

Cutting Speed for Milling is the speed at the outside edge of the milling cutter as it is rotating. This is also known as surface speed. Surface speed, surface footage, and surface area are all directly related.

All cutting tools work on the surface footage principle. Cutting speeds depend primarily on two things, the kind of material being cut and the kind of cutting tool being using. The hardness of the work material has a great deal to do with the recommended cutting speed. The harder the work material the slower the cutting speed and the softer the work material the faster the recommended cutting speed.

The recommended cutting speed charts for drilling operations with high-speed steel drills in relation to their hardness is presented in Table.5.1[75].

Table 5.1: Recommended cutting speeds for drilling with high-speed steel drills (fpm).

Material	Hardness, (Bhn)	Cutting Speed, (fpm)
Plain Carbon Steels AISI–1019, 1020, 1030, 1040, 1050, 1060, 1070, 1080, 1090	120–150 150–170 170–190 190–220 220–280 280–350 350–425	80–120 70–90 60–80 50–70 40–50 30–40 15–30
Alloy Steels AISI-1320, 2317, 2515, 3120, 3316, 4012, 4020, 4120, 4128, 4320, 4620, 4720, 4820, 5020, 5120, 6120, 6325, 6415, 8620, 8720, 9315	125–175 175–225 225–275 275–325 325–375 375–425	60–80 50–70 45–60 35–55 30–40 15–30
Tool Steels Water Hardening Cold Work Shock Resisting Mold High-Speed Steel	150–250 200–250 175–225 100–150 150–200 200–250 250-275	70–80 20–40 40–50 60–70 50–60 30–40 15–30
Gray Cast-Iron	110–140 150–190 190–220 220–260 260–320	90–140 80–100 60–80 50–70 30–40

On the other hand, the recommended cutting speed for milling machine operations with high-speed steel milling cutters in relation to their hardness is presented in Table.5.2 [76].

Table 5.2: Recommended Cutting Speed for Milling with high-speed steel milling cutter (fpm).

Material	Hardness, (Bhn)	Cutting Speed, (fpm)
Plain Carbon Steel, AISI1010 to AISI 1030	to 150 150 to 200	110 100 to 140 100 80 to 120
Gray cast iron	120 to 180 180 to 225 225 to 300	80 70 30
All Alloy Steels Having .3% or Less Carbon Content: AISI 1320, AISI 3120, AISI 4130, AISI 4020, AISI 5020, AISI 4118, AISI 9310, etc.	180 to 220 220 to 300 300 to 400	80 65 to 100 60 30 to 80 40 30 to 50
All Alloy Steels Having More Than .3% Carbon Content: AISI 1340, AISI 2340, AISI 4140, AISI 4150, AISI 4340, AISI 5140, AISI 5150, AISI 52100, AISI 8660, AISI 9260, etc.	180 to 220 220 to 300 300 to 400	80 60 to 100 55 30 to 80 30 20 to 50

5.2.2 Feed Rate

The speed of the cutter's movement is called the feed rate. The feed rate depends on many factors, including the type of material being cut, the type of cutter used, and the condition of the CNC machine. The spindle feed rate on drilling machines is given in terms of Millimetres Per Revolution (MPR). Millimetres per revolution are the rate at which the tool advances into the work at every revolution of the tool. The feed rate that can be used is determined mainly by the size of the chip that the drill can withstand. As the size of the drill increases, the feed rate of the drill also increases.

The table feed rate on milling machines is given in terms of Millimetres Per Minute (MPM). Inches per minute are the rate at which the tool will advance into the work. The feed rate, that can be used, is determined by the speed of the rotation of the cutter (RPM), the number of cutting teeth on the cutter, and by the size of the chip that the cutter can withstand. The chip size is called the feed rate in inches per tooth or chip load [77] which is shown in fig. 5.1.

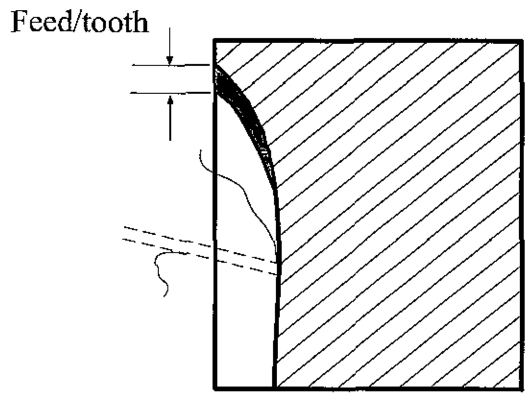


Fig. 5.1: Feed rate in inches per tooth or chip load.

The drilling machine operation with high-speed steel drills which also depends on the feed rate is presented in Table.5.3 in relation to the drill diameter [78].

Table 5.3: Recommended feed rate for high-speed steel Twist drills.

Drill diameter (mm)	Feed (mm/rev)
1.587 to 3.175	0.0254 to 0.0762
3.175 to 6.35	0.0508 to 0.1524
6.35 to 12.7	0.1016 to 0.254
12.7 to 25.4	0.1778 to 0.381
Over 25.4	0.381 to 0.635

The recommended feed inch/tooth (mm/tooth) for milling machine operations with a high-speed steel mill cutter in relation to its hardness, depth of cut and cutter diameter is presented in Table.5.4 [79].

Table 5.4: Recommended feed in inch /tooth (mm/tooth) for milling with high-speed steels cutters.

Material	Hardness (HB)	End Mills		
		Depth of cut, 0.250 in. (6.35 mm)		
		Cutter diameter in. (mm)		
		½ in. (12.7 mm)	¾ in. (19.05)	1 in. and up (25 mm and up)
		Feed per Tooth, in. (mm)		
Plain carbon steels, AISI 1006 to 1030, 1513 to 1522	100-150	0.001 (0.0254)	0.003 (0.0762)	0.003 (0.0762)
	150-200	0.001 (0.0254)	0.002 (0.0508)	0.003 (0.0762)
	120-180	0.001 (0.0254)	0.003 (0.0762)	0.003 (0.0762)
	180-220	0.001 (0.0254)	0.002 (0.0508)	0.003 (0.0762)
	220-300	0.001 (0.0254)	0.002 (0.0508)	0.002 (0.0508)
Alloys steels having less than 3% carbon. Typical examples: AISI 4012, 4023, 4027, 4118, 4320, 4422, 4427, 4615, 4620, 8620, 93b17	125-175	0.001 (0.0254)	0.003 (0.0762)	0.003 (0.0762)
	175-225	0.001 (0.0254)	0.002 (0.0508)	0.003 (0.0762)
	225-275	0.001 (0.0254)	0.002 (0.0508)	0 003 (0.0762)
	275-325	0.001 (0.0254)	0.002 (0.0508)	0.002 (0.0508)
Alloys steels having 3% carbon or more. Typical examples: AISI 1330, 1340, 4032, 4037, 4130, 4140, 8640, 94b30	175-225	0.001 (0.0254)	0.002 (0.0508)	0.003 (0.0762)
	225-275	0.001 (0.0254)	0.002 (0.0508)	0.003 (0.0762)
	275-325	0.001 (0.0254)	0.002 (0.0508)	0.002 (0.0508)
	325-375	0.001 (0.0254)	0.002 (0.0508)	0.002 (0.0508)
Gray cast iron	120 - 180	0.003 (0.0762)	0.003 (0.0762)	0.004 (0.1016)
	180 - 225	0.001 (0.0254)	0.002 (0.0508)	0.003 (0.0762)
	225 - 300	0.001 (0.0254)	0.002 (0.0508)	0.002 (0.0508)

5.2.3 Depth of Cut

The depth of cut is the thickness of material removed in a machining operation. The depth of cut has a significant influence on side deflection. The depth of cut and the feed directly influence the performance and tool life of an insert. Using a small depth of cut with a wide insert may result in a deflection that is too small to be effective. This may result in vibration. If the depth of cut is too large for the width of an insert, or the feed too high, the insert may be overloaded, causing immediate breakage. In the finishing operation, when the depth of cut is normally minimal, it is important to select the proper insert with a small width and a small corner radius.

5.3 High-Speed Steels

In the early 1900s, high-speed steels were the most highly alloyed of the tool steels. High-speed steel (HSS) tools are so named because they were developed to cut at higher speeds. High-speed steels have high toughness and resistance to fracture, which are especially suitable for high positive-rake-angle tools, interrupted cuts, and for machine tools with low stiffness that are subject to vibration and chatter. They can be hardened to various depths, have good wear resistance, and are relatively inexpensive.

Two basic types of high-speed steels are available. The *molybdenum* (M series), which contains up to about 10 % molybdenum with chromium, vanadium, tungsten, and cobalt as alloying elements. The other type is the *tungsten* (T-series), which contains 12-18 % tungsten, with chromium, vanadium, and cobalt as alloying elements. High-speed steel tools are available in shaped, cast and sintered (powder metallurgy) forms. To improve performance, high-speed steel tools can be coated.

5.4 Drilling

In recent times, nearly 25% of all the cutting tools in the world are used for drilling operations. Some operations are strictly drilling operations. However drilling machines can be used to perform other operations such as reaming, tapping, countersinking and counterboring. The same rules and principles of cutting speed and RPM calculations apply for all the operations which are performed in drilling machines. For example, the reamer needs half the cutting speed and twice the feed as drilling. The most important requirement is to pay attention to the cutting speeds which have the greatest impact on tool life.

Table 5.5: Feature machining process parameters for drilling.

Feature parameters	**Input for feature**
Category k	I
Number of features	10
Diameter (mm)	12.7 mm (0.5 in)
Hole depth (mm)	10 mm (0.393 in)
Drill length (mm)	80

Table 5.6: Alternative machining parameters for drilling.

Material	**Cast iron**
Feed rate (mm/rev)	0.1778 mm/rev
Cutting speed (ft/min)	130 ft/min (39.62 m/min)

5.4.1 Total Time for Drilling Operations

The feed-based approach is based on the feed rate and length of cut for the process [80]. The drilling time for a 12.7 mm hole is obtained from:

$$T = \frac{L}{F} \qquad \textbf{(5-1)}$$

Where

T = Machining time in minutes or second also referred to as cutting time

F = Feed rate in in/min or mm/min

L = Length of cut in the feed direction, inch or mm.

The value of the cutting speed V is taken from Table 5.6 and the diameter of the cutter is 0.5 inch.

$$N = \frac{\text{Cutting speed} \times 4}{\text{Diameter of the cutter}}$$

$$= \frac{130 \times 4}{0.5}$$

$$= 1040 \text{ rpm}$$

Most drilling machines are set up to feed in inches per revolution. If, however, the feed rate for the machine is setup in inches per minute (IPM), the operator needs to multiply the operating R.P.M. of the drill by the feed rate in inches per revolution.

The value of f_r = 0.1778 mm/rev or 0.007 in/rev, from the Table 5.6, and the RPM is calculated above. Then,

$$\text{Feed (in/min)} = \text{RPM} \times \text{Feed in inches per revolution}$$

$$= 1040 \times 0.007$$

$$= \mathbf{7.28 \text{ in/min}}$$

The hole length is $L =$ 10 mm or 0.393 inch from the Table. 5.5. Then from equation (5-1), the drilling time for the ½ in. (12.7 mm) holes is

$$T = \frac{L}{F}$$

= 0.0539 min/hole = **0.539 minutes** for 10 holes

The handling time can be estimated from a database such as that by Ostwald or Boothroyd and Knight [81]. The handling time, which also includes the indexing time, is 0.30 minutes from Table. 5.7. Allowances of 9% for machining and 15% for handling time are applied.

Table 5.7: Basic loading and unloading times (in min.) for various work holding devices and for different workpiece weights.

Work holding device	Loading and unloading times per piece for workpiece weight ranges	
	0 – 10 lbs	**10 – 30 lbs**
Between centers (no dog)	0.30 min	0.40 min
Between centers (with dog)	0.67 min	0.96 min
Universal chuck	0.39 min	0.53 min
Independent chuck	0.69 min	0.83 min
V block	0.50 min	0.59 min
Vice	0.30 min	0.40 min

Then,

$$T = 1.09 \times 0.539 + 1.15 \times 0.30$$

$$= \mathbf{0.932\ min.}$$

Table 5.8: Setup times for basic machining operations.

Operation description	Holding device	Setup	Time (min)
Milling Plane surface	Vice Collet or chuck	1 tool	9-11
Drilling Sensitive	Table or vice	2 spindles	5-7

The set up time for the two spindle drill press is 15 minutes, as taken from Table 5.8 [82]. If the100% adjustment is applied, then the total time to produce a part is

Total time = Setup time + Time/piece

= 5 + 0.932

= 5.932 min. or 0.098 hours

If it is assume that the hourly machine (labour) rate is minimum wage of Ireland **(€ 8.65)** per hour then the total cost for labour is **€ 0.855** for 1 part with 10 holes.

5.4.2 Power Calculation

The material removal rate (MRR) in drilling is the volume of material removed by the drill per unit time. Suppose for a drill with diameter D, the cross-sectional area of the drilled hole is $\frac{\pi D^2}{4}$. The velocity of the drill perpendicular to the workpiece is the product of the feed f and the rotational speed N [74],

Then, $$\text{MRR}=\frac{\pi D^2}{4}\times(f)\times(N) \quad \textbf{(5-2)}$$

The value of D, f is in Table 5.5 and Table 5.6. And the value of N is from quation (5-2), then

$$\text{MRR}=\frac{3.1416\times(12.7)^2}{4}\times(0.1778)\times(1040)$$

$= 23{,}424\ \text{mm}^3/\text{min}$

$= 390.40\ \text{mm}^3/\text{s}$ for 1 hole

$= 3904.02\ \text{mm}^3/\text{s}$ for 10 holes

The average unit power of 5 W per s/mm^3 for cast iron is taken from Table 5.9 [74]. Hence the power required is

Power = (3904.02) × (5) × 0.932

= 18192 W

= **18.19 KW**

From the Ireland Electricity Supply Board, the unit price cost for general purpose tariff is €0.1610 / KWh. So the total price of power used in drilling is

= (18.19) (0.1610)

= € 2.92

= **€ 3.31** (with 13.5 % vat)

Table 5.9: Approximate energy requirements in cutting operations.

Material	Specific energy	
	W. s/mm^3	**hp. min/in^3**
Aluminum alloys	0.4 – 1.1	0.15 – 0.4
Cast irons	1.6 – 5.5	0.6 – 2.0
Copper alloys	1.4 – 3.3	0.5 – 1.2

So, the total machining costs for drilling category "k" on a CNC machine is the sum of the machine cost and the tooling cost for category k [83].

$$C_{duCNC} = \left(\frac{1.2.d_k}{f_k}.(1+E).H_k.M\right) + \left(\frac{1.2.d_k.(C_{pt}+t_{tc}.M)}{f_k.T_k}\right).H_k \qquad \textbf{(5-3)}$$

In equation (5-3),

$C_{tk} = \left(\frac{1.2.d_k(C_{pt}+t_{tc}.M)}{f_k.T_k}\right).H_k$, is the tooling cost which can be found from the Table 5.12 [74] and the cost for a ½ inch high-speed twist drill is **€ 3.86.**

Then, from equation (5-3),

$$C_{duCNC} = \left(\frac{1.2\times 10}{0.1778}\times(1)\times 1\times 0.855\right) + \text{Tooling cost}$$

$$= € \ 57.36 + 3.86$$

$$= \textbf{€ 61.22 for 10 holes in drilling (approximate)}$$

5.5 Milling

Milling machines are used to perform a wide variety of machining operations. There are some operations that are strictly milling operations, but milling machines can be used to perform other operations such as drilling, reaming, tapping, and boring. The rules and principles of cutting speeds and R.P.M. calculations that apply to these "other" operations performed on milling machines are still used in the same manner.

Table 5.10: Feature machining process parameters for milling.

Feature parameters	Input for feature
Category k	I
Number of operations	10
End mill diameter (mm)	12.7 mm (1/2 in.)
Axial depth of cut (mm)	6.35 mm (0.25 in.)
Radial rake angle (deg)	10
Number of teeth/cutter	2
Depth of hole	10 mm (0.393 in)

Table 5.11: Alternative machining parameters for milling.

Material	Cast iron
Feed rate (mm/tooth)	0.762 mm/tooth (0.003 in/tooth)
Cutting speed (ft/min)	80 ft/min

5.5.1 Total Time for Milling Operation

The feed-based approach is based upon the feed rate and length of cut for the process [80]. The milling time for the ½ in. (12.7 mm) holes is obtained using the same equation as for drilling.

$$T = \frac{L}{F}$$

The value of cutting speed V is taken from Table 5.11 and the diameter of the cutter is 0.5 inch.

$$N = \frac{\text{Cutting speed} \times 4}{\text{Diameter of the cutter}}$$

$$= \frac{80 \times 4}{0.5}$$

$$= \mathbf{640\ rpm}$$

The feed rate in inches per tooth must be converted into feed rate in inches per minute (IPM) to ensure the correct feed rate setting on the machine. The formula for converting feed rate in inches per tooth into inches per minute is as follows:

The value of f_t = 0.0762 mm/tooth or 0.003 in/tooth from the Table. 5.11, which is treated as chip load (CL). Number of teeth (flute) is 4 and the RPM is calculated above.

$$\text{Feed (in/min)} = \text{RPM} \times \text{Chip load (CL)} \times \#\ \text{Teeth (flute)}$$
$$= 640 \times 0.003 \times 4$$
$$= \mathbf{7.68\ in/min}$$

The hole length is $L =$ 10 mm or 0.393 inch from Table. 5.10. Then from the equation,

$$T = \frac{L}{F}$$

$$= 0.0511 \text{ min/hole} = \mathbf{0.511\ min.} \text{ for all 10 holes}$$

The handling time can be estimated from a database such as that by Ostwald or Boothroyd and Knight [81]. The handling time, which also includes the indexing time, is 0.30 minutes from Table. 5.7. Assuming allowances of 9% for machining and 15% for handling time are applied, then thetotal time is:

$$T = 1.09 \times 0.511 + 1.15 \times 0.30$$
$$= \mathbf{0.901\ minutes}$$

The set up time for 1 tool milling vice is 69 minutes, as taken from Table 5.8 [82]. If the100% adjustment is applied, then the total time to produce per parts is

Total time = Setup time + Time/piece

= 9 + 0.901

= **9.901 min or .165 hours**

If it is assume that the hourly machine (labour) rate is minimum payment of Ireland **(€ 8.65)** per hour then the total cost for labour is **€ 1.42** for 1 part with 10 holes.

5.5.2 Power Calculation

The material removal rate (MRR) in milling is the volume of material removed by the cutter per unit time. Suppose for an end mill cutter with diameter D, the cross-sectional area of the end milled hole is $\frac{\pi D^2}{4}$. The velocity of the end mill cutter perpendicular to the workpiece is the product of the feed f and the rotational speed N [74],

Then, $$\text{MRR}=\frac{\pi D^2}{4}\times(f)\times(N)$$

The value of D and f is in Table 5.10 and Table 5.11. However, the value of N is in the above equation.

Then, $$\text{MRR}=\frac{3.1416\times(12.7)^2}{4}\times(0.0762)\times(640)$$

$$= 6{,}177.79 \text{ mm}^3/\text{min}$$

$$= 102.96 \text{ mm}^3/\text{s for 1 hole}$$

$$= 1029.63 \text{ mm}^3/\text{s for 10 holes}$$

The average unit power of 5 W per s/mm^3 for cast iron is taken from Table 5.9 [74]. Hence the power required is

$$\text{Power} = (1029.63) \times (5) \times 0.901$$

$$= 4638.48 \text{ W}$$

$$= \mathbf{4.63\ KW}$$

From the Ireland Electricity Supply Board, unit price cost for general purpose tariff is €0.1610 / KWh. So the total price of power used in drilling is

= (4.63) (0.1610)

= € 0.746

= **€ 0.846** (with 13.5 % vat)

So, the total machining costs for milling category "k" on a CNC machine is the sum of the machine cost and the tooling cost for category k [83].

$$C_{muCNC} = \left(\frac{(d_m + D_m/2)\times\pi}{f}\right)\times(1+E)\times H_k \times M + \left(\frac{(d_m + D_m/2)\times\pi\times T_{mk}}{f}\right)\times(C_{pt} + t_{tc}\times M)\times H_k \tag{5-4}$$

In equation (5-4),

$C_{tk} = \left(\frac{(d_m + D_m/2)\times\pi\times T_{mk}}{f}\right)\times(C_{pt} + t_{tc}\times M)\times H_k$, is the tooling cost which can be found from Table 5.12 [74] and the cost for ½ inch high-speed end mill cutter is **€ 6.44.**

Table 5.12: Approximate cost of selected tools for machining.

Tools	Size, in. (mm)	Cost, (€)
Drills, HSS, straight shank	¼ (6.35) ½ (12.7)	0.664 - 1 32 1.93 – 3.86
Tapered shank	¼ (6.35) 1 (25 4)	1.61 – 4.50 9.66 – 28 98
End mills, HSS	½ (12.7) 1 (25.4)	6.44- 9.66 9.66 – 19.32
Carbide-tipped	½ (6.35) 1 (25.4)	19.32 – 22.54 28.98 – 38.64
Solid carbide	½ (12.7) 1 (25.4)	19.32 – 45.08 115.92
Burs, carbide	½ (12.7) 1 (25.4)	6.44 – 12.88 32.2 – 38.64
Milling cutters, HSS, staggered tooth, 3/8 in.	4 (101.6) 8 (203.2)	22.54 – 48.3 83.72 – 167.44

Then, from equation (5-4),

$$C_{muCNC} = \left(\frac{(12.7 + 12.7/2) \times 3.1416}{0.762}\right) \times (1) \times 1 \times 1.42 + \text{Tooling cost}$$

$$= € \ 111.52 + 6.44$$

$$= \mathbf{€ \ 117.96 \ for \ 10 \ holes \ in \ milling \ (approximate)}$$

In the above calculations, the value of E (ratio between productive and non-productive time) and H_k (holes per hit in category k) was taken to be 1. As 10 holes with the same dimensions in one part were calculated, there was no efficiency factor. All the setup time was calculated and there was no tool change time. Tool life was 60-120 (for high-speed steel tool) minutes but the operation took place for 5.932 minutes (for drilling) and 9.90 minutes (for milling). One tool setup was sufficient to perform this operation. However, holes per hit in category k was taken as 1 as all the holes were in the same part. The above calculated cost may not be suitable for the Industry level as they have their own cutting conditions, tool selection, labour cost and materials.

5.6 Roughness

Roughness consists of surface irregularities which result from the various machining processes. These irregularities combine to form surface texture. In general the quality of machined surface is characterised by the accuracy of its manufacture with respect to the dimensions specified by the designer. Characteristic evidence on the machined surface is found after machining operations. This evidence is in the form of finely spaced micro irregularities left by the cutting tool. Different types of pattern are found for different types of cutting tool which can be identified after the machining operation.

On the other hand, ideal surface roughness is a function of only feed and geometry which represent the best possible finish and can be obtained for a given tool shape and feed. The theoretical surface roughness can only be achieved if the built-up-edge, chatter and inaccuracies in the machine tool movements are eliminated completely [84]. For a sharp tool without nose radius, the maximum height of unevenness is given by:

$$R_{max} = \frac{f}{\cot\phi + \cot\beta} \tag{5-5}$$

Then the surface roughness value is given by:

$$R_a = \frac{R_{max}}{4} \tag{5-6}$$

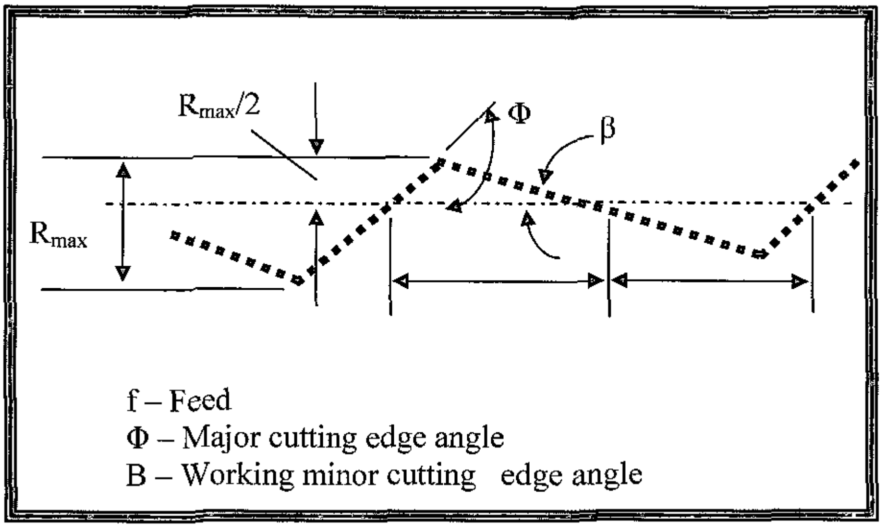


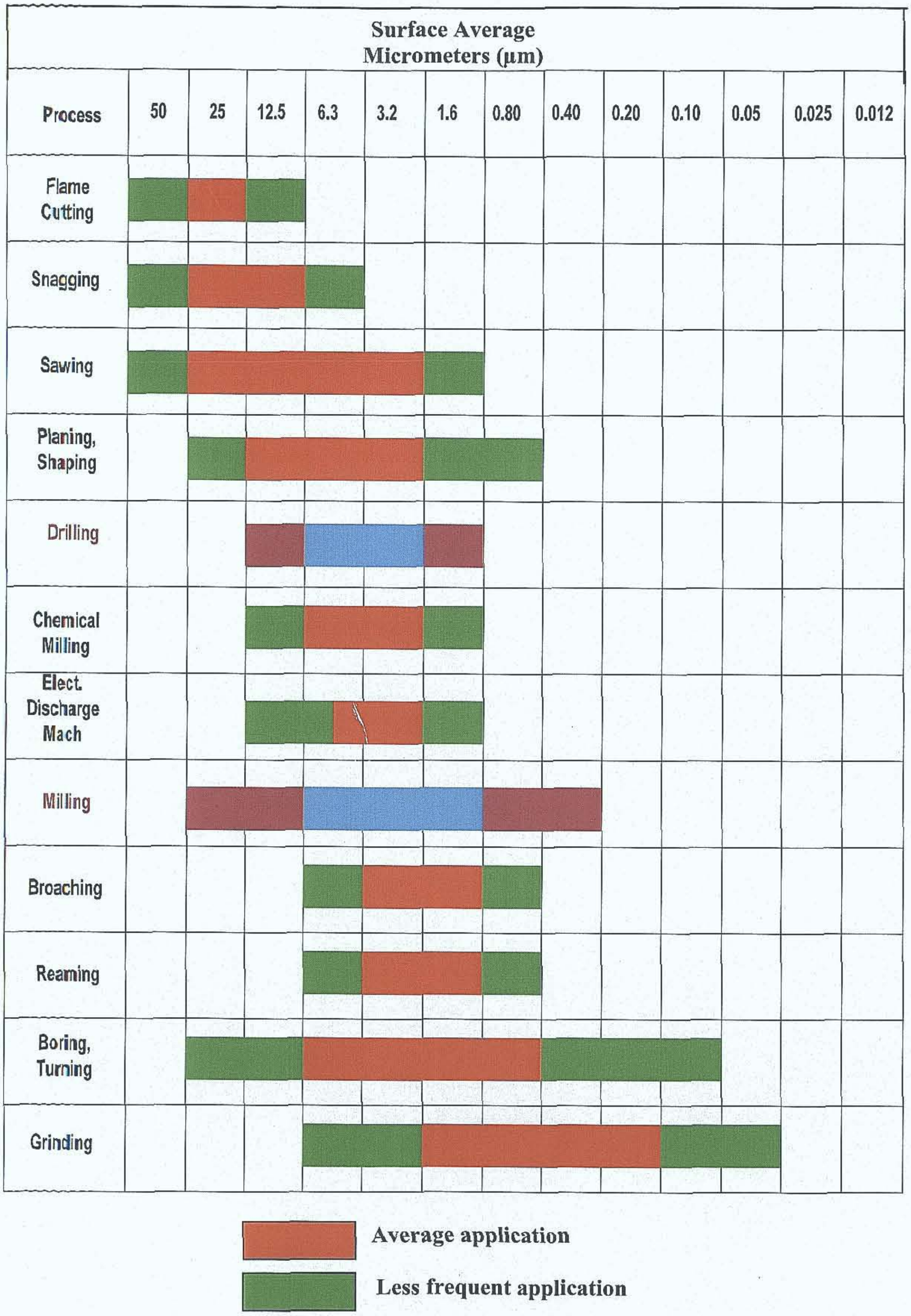
Fig. 5.2: Idealised model of surface roughness.

It can be shown that the roughness value is related to the feed and corner radius by the following expression:

$$R_a = \frac{0.0321 f^2}{r} \tag{5-7}$$

The surface roughness produced by milling and drilling operations is (0.80-6.3 μm) and (1.6-6.3 μm) respectively from the Table 5.13.

Table 5.13: Surface Roughnesses Produced By Common Production Processes [74].



5.7 Summary of This Chapter

It can be seen that, after calculating machining cost of two different machining processes, milling and drilling, the cost of the drilling process is much lower than the cost of the milling process. The main reason behind the higher cost of the milling process is the lower cutting speeds, feeds and the higher tooling cost. However, more setup time is needed to perform the machining operation. So, from this calculation it is easy to conclude that the drilling process is more economically justifiable than milling.

But for the high precision surface roughness, milling process is best suited as the surface roughness ranges from 0.8-6.3 μm.

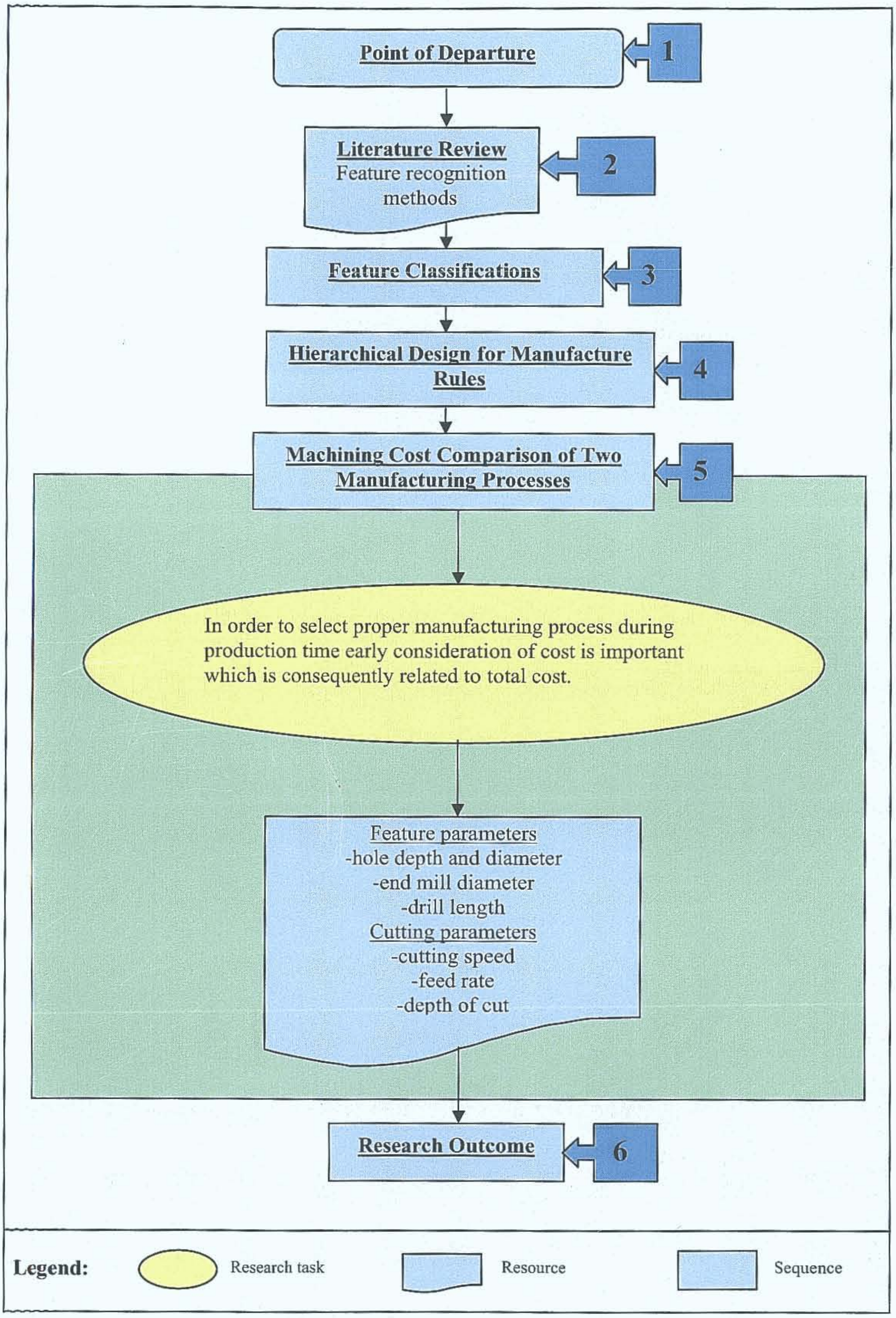


Fig. 5.3: Summary of the research process in chapter five.

CHAPTER 6 CONCLUSIONS AND FUTURE WORK

6.1 Conclusions

The performance of production processes suffered poorly in the manufacturing sector due to insufficient reconciliation of process capabilities with design requirements. Special processes are often poorly understood and frequently modified during the production time. In order to avoid the practice of "do it anyway" instead of "do it right" for set up plan requirements, Design for Manufacture (DFM) can be used. Due to the complexity of detailed design and processing, it is still impossible to completely replace the human decision factor with an automatic manufacturing analysis system. Poor designs increase the product cost Product cost includes the design costs and the manufacturing costs. However, labour cost (direct and indirect) amounting to 2-15% of the total cost, materials and manufacturing processes of up to 50-80% of the total cost, and overheads 15-45% of total are the manufacturing costs. Implementation of DFM in an organisation is heavily dependent on the effectiveness of its Product Design Process (PDP).

A new approach of feature classification has been shown in this thesis. Features such as hole feature, pocket feature, slot feature, boss feature and step feature have been associated with their possible characteristics and manufacturing processes. The developed system helps the designer to select proper manufacturing processes during the design phase. This relates to production cycle times and cost.

Although most manufacturing process guidelines have been in existent for diverse manufacturing applications, there is still a lack of hierarchical DFM guidelines and rules. This thesis contributes to the development of a structured, hierarchical design for manufacture guideline system. This allows to appreciate and consider process capabilities and limitations during the design process in order to minimise manufacturing cost.

This thesis also contributes to the selection of the proper manufacturing processes in relation to machining cost. A simple machining cost estimation of producing ten holes using milling and drilling process has been shown in detail. The estimated cost shows that the drilling process is suited but for a high precision surface roughness the milling process should be in consideration.

6.2 Future Work

The developed system can be added to a design software tool. The designer, using the tool, will then be able to perform what-if scenarios and evaluate the design from a manufacturing point of view. Each manufacturing process contains design recommendations from which a designer can easily get an idea about which processes are suitable for which feature for manufacture making it easier to design any product. It is to be mentioned that the DFM system will not restrict the design process, but will give practical information about the manufacturing constraints which may occur during the product manufacture. The designer can also chose whatever materials the manufacturer would prefer for manufacturing the parts. At the end the user would be aware of the producibility of the product with regard to the choice of material, production type and feature's characteristics.

The design for manufacture (DFM) rules system can be embedded to the Pro-engineer or similar design software. In Pro-engineer there could be a manufacturing feature library in which all the rules could be added. Whenever a designer starts a design this system will notify the designer if the specified design rules are violated. Not only will it show a message but also it will indicate the correct specification for the design.

REFERENCES

[1] Pratt M. J. and Wilson P. R. "Requirements for support of form features in a solid modelling system", *CAM-I, R-85-ASPP-01*, 1985.

[2] Henderson M. R. Chuang S. N., Ganu P. and Gavankar P. "*Graph-based feature extraction*", Arizona State University, 1990.

[3] Kao C. Y., Kumara S. R. T. and Kasturi R. "Extraction of 3D object features from CAD boundary representation using super relation graph method", *IEEE Transaction on Pattern Analysis and Machine Intelligence,* vol. 17(12), pp. 1228-1233, 1995.

[4] Chang T. C. "Expert process planning for manufacturing", *Reading, Massachusetts: Adision-Wesley Publishing Company*, 1990.

[5] Gossard D. C., Zuffante R. P. and Sakurai H. "Representing dimensions, tolerances and features in mcae system", *IEEE Computer Graphics & Applications,* vol. 8(2), pp. 51-59, 1988.

[6] Henderson M. R. and Chang G. J. "FRAP: automated feature recognition and process planning from solid model data", *American Society of Mechanical Engineering (ASME)*, pp. 529-536, 1988.

[7] Kayacan M. C. and Celik S. A. "Process planning system for prismatic parts", *Integrated Manufacturing Systems*, vol. 14(2), pp. 75-86, 2003.

[8] Roy U. and Liu C. R. "Feature-based representational scheme of a solid modeller for providing dimension and tolerencing information", *Robotics and Computer-Integrated Manufacturing*, vol. 4(3/4), pp. 335-345, 1988.

[9] Wang N. and Ozsoy T. M. "A scheme to represent features, dimensions and tolerances in geometric modelıng", *Journal of Manufacturıng System*, vol. 10(3), pp. 233-240, 1991.

[10] Nasr E. S. A. and Kamrani A. K. "A new methodology for extracting manufacturing features from CAD system", *Computer and Industrial Engineerıng,* Elsevier, vol. 51, pp. 389-415, 2006.

[11] Nagaraj H. S. and Gorumoorthy B. "Machinable volume extraction for automatic process planning", *IIE transaction*, vol. 34(4), pp. 393-410, 2002.

[12] Sharma R. and Gao J. X. "Implementatıon of step application protocol 224 ın an automated manufacturıng planning system", *Proceedings of The Institution of Mechanical Engineers, part B: Journal of Engineering Manufacture,* vol. 216(1), pp. 1277-1289, 2002.

[13] Joshi S. and Chang T. C. "Graph based heuristics for recognition of machined features from a 3-D solid models", *Computer aided design*, vol. 20, pp. 58-66, 1988.

[14] Han J. H., Pratt M. K., and Regli W. C. "Manufacturıng feature recognition from solid models", *IEEE transaction on robotics and automation*, Vol-16, No-6, December 2000.

[15] Trika S. N. and Kashyp R. L. "Geometric reasoning for extraction of manufacturing features in ıso-oriented polyhedrons*", IEEE transaction Pattern Analysis Machine Intel,* vol. 16, no. 11, pp. 1087-1100, 1994.

[16] Choi B. K., Barash M. M. and Anderson D. C. "Automated recognition of machined surfaces from a 3D solid model", *Computer Aided Design*, Vol. 16, No. 2, pp. 81-86, March 1984.

[17] Barwick S. P. and Bowyer A. "Feature Technology," Universıty of Bath, *Technical report* 001/1993.

[18] Henderson M. R. and Anderson D. C. "Computer recognition and extraction of form features", A *CAD/CAM link' Computers in Industry,* vol. 5, pp. 329-339, 1984.

[19] Henderson M. R. and Anderson D. C. "Computer recognition and extraction of form features", *A CAD/CAM link: Computers in Industry*, vol. 5, pp. 329-339, 1984.

[20] Kyprianou L. K. "Shape classification in computer-aided design", *PhD thesis,* Chirst College, University of Cambridge, Cambridge, United Kingdom, July 1980.

[21] Han J. H., Prat M. K. and Regli W. C. "Manufacturing feature recognition from solid models", *IEEE Transaction on Robotics and Automation,* Vol. 16, No.6, December 2000.

[22] Woo. T. "Feature extraction by volume decomposition", in *Proceeding of Conf. CAD/CAM Technology in Mechanical Engineering,* 1982

[23] Kim Y. "Convex decomposition and solid geometric modelling", *PhD. Dissertation,* Stanford University, 1990.

[24] Kim Y., "Recognition of form features using convex decomposition", *Computer Aided Design,* vol. 24, no. 9, pp. 461-476, 1992.

[25] Waco D. and Kim Y. S., "Geometric reasoning for machining features using convex decomposition", in *Proceedings of 2nd ACM Solid Modeling Symposium,* pp. 323-331, 1993.

[26] Waco D. and Kim Y. S., "Considerations in positive to negative conversation for machining features using convex decomposition", in *Proceedings of ASME Computer in Engineering Conference,* pp. 35-45, 1993.

[27] Waco D. and Kim Y. S., "Geometric reasoning for machining features using convex decomposition", *Computer Aided Design*, vol.26, no.6, pp. 477-489, 1994.

[28] Kim Y., Wang E., Lee C. and Rho H. "Feature-based machining precedence reasoning and sequence planning", in *Proceeding of 1998 ASME Design Engineering Technical Conference (DETC98/CIE-5707)*, 1998.

[29] Martino S. and Kim Y. "Cylindrical features in form feature recognition using convex decomposition", in *Proceeding of IFIP conference on Feature Modeling and Recognition in Advanced CAD/CAM Systems*, 1994

[30] Wang E. and Kim Y. "1997 status of the form feature recognition method using convex decomposition", in *Proceeding of ASME Computers in Engineering Conference*, 1997.

[31] Christensen N. C., Emory J. D. and Smith M. L., "Phoenix method for automatic conversion between geometric models", *Allied Signal Incorporated*, Kansas City, MO. US Patent 728367, 1983.

[32] Sakurai H. and Chin C., "Definition and recognition of volume features for process planning", in *Advances in Feature Based Manufacturing* , Shah J. J., Mantyla M. and Nau D. S. Eds, Amsterdam, The Netherlands: Elsevier science B. V., pp. 65-80, 1994.

[33] Coles J., Crawford R. and Wood K. "Form feature recognition using base volume decomposition", in *Proceedings ASME Design Automation Conference*, pp. 281-297, 1994.

[34] Shah J. J., Shen Y. and Shirur A., "Determination of machining volumes from extensible sets of design features", in *Advances in Feature Based Manufacturing*, Shah J. J., Mantyla M. and Nau D. S., Eds, Amsterdam, The Netherlands: Elsevier Science B .V., pp. 129-157, 1994.

[35] Sakurai H. and Dave P. "Volume decomposition and feature recognition, part II: Curved objects", *Computer-Aided Design*, vol. 28, no. 6-7, pp. 519-537, 1996.

[36] Kraker K. J. De., Dohmen M. and Bronsvoort W. F. "Maintaining multiple views in feature modeling", in *Proceeding of Solid Modeling' 97*, pp. 123-130, 1997.

[37] Bidarra R., Karker K. J. De. and Bronsvoort W. F. "Representation and management of feature information in a cellular model", *Computer-Aided Design*, vol. 30, no. 4, 1998.

[38] Requicha A. A. and Vandenbrande J. H. "Form features for mechanical design and manufacturing", *ASME Computers in Engineering Conference*, Anaheim, CA, USA, Book number G0502A-1989, eds. Riley. D et al., pp. 47-52, Aug 1988.

[39] Arab M., "Requirements and architecture of CAM oriented CAD systems for design and manufacture of mechanical parts", *PhD Dissertation*, University of California, Los Angles, USA, 1982.

[40] Li R. and Yu M., "A framework for prismatic part-data generation-unit-machined loop concept", in *International Journal Computer Integrated Manufacturing*, Vol. 3, No. 2, pp. 96-111, 1990.

[41] Perng D., Chen Z. and Li R., "Automatic 3D machining feature extraction from 3D CSG solid input", *Computer Aided Design*, Vol. 22, No. 5, pp. 285-295, June 1990.

[42] Barwick S. P. and Bowyer A., "Feature technology", *Technical Report 001/1993*, University of Bath.

[43] Cybenko G., Bhasin A. and Cohen K. D. "Pattern recognition of 3D CAD objects: Towards an electronic yellow pages of mechanical parts", *Smart Engineering Systems Design*, vol. 1, pp. 1-13, 1997.

[44] Vandenbrande J. H. and Requicha A. A. G. "Special reasoning for the automatic recognition of machinable features in solid models", *IEEE Trans. Pattern Analysis and Machine Intelligence*, Vol.15, pp. 1-17, Dec 1993.

[45] Vandenbrande J. H. and Requicha A. A. G. "Geometric computation for the recognition of spatially interacting machinable features", In Shah J. J., Mantyla M. and Nau D. S. *Advances in Feature Based Manufacturing,* Elsevier Science, B. V., Amsterdam, The Netherlands, pp. 83-106, 1994.

[46] Regli W. C. "Geometric algorithms for recognition of features from solid models", *PhD dissertation,* University of Maryland, College Park, MD, 1995.

[47] Han J. and Requicha A. A. G. "Hint generation and completion for feature recognition", in *Proceedings of International Symposium Automotive Technology and Automation (ISATA),* pp. 89-96, 1996.

[48] Brooks S. L. and Greenway Jr R. B. "Using STEP to integrate design features with manufacturing features", in *ASME Computers in Engineering Conference,*" A. A. Busnaina, Ed. Boston, MA, pp. 579-586, Sept 1995.

[49] Regli W. C., Gupta S. K. and Nau D. S. "Toward multiprocessor feature recognition", *Computer-Aided Design,* vol. 29, no. 1, pp. 37-51, 1997.

[50] Simmons R. "A theory of debugging plans and interpretation", in *Proceedings AAAI,* pp. 94-99, 1988.

[51] Van Houten F. J. A. M. "PART: A computer-aided process planning system", *PhD. Dissertation,* University of Twente, 1991.

[52] Nau D. S., Regli W. C. and Gupta S. K. "Al planning versus manufacturing-operations planning: A case study", in *Proceeding of 14 th Int. Joint Conf. Artificial Intelligence,* Montreal, PQ, Canada, vol. 2, pp. 1670-1676, Aug. 1995.

[53] Das D., Gupta S. K. and Nau D. S. "Generating redesign suggestion to reduce setup cost: A step toward automated redesign", *Computer-Aided Design,* 1996.

[54] Gupta S. K. and Nau D. S. "A systematic approach for analyzing the manufacturability of machined parts", *Computer-Aided Design,* vol. 27, no. 5, 1995.

[55] Regli W. C., Gupta S. K. and Nau D. S. "Extracting alternative manufacturing features: An algorithmic approach", *Res. Eng. Design*, vol. 7, no. 3, pp. 173-192, 1995.

[56] Gupta S. K., Regli W. C. and Nau D. S. "Imacs: A case study in real-world planning", *IEEE Intel System*, Vol. 13, no. 3, pp. 39-51, 1998.

[57] Han J. "On multiple interpretations", in *Proceedings of 4th ACM SIGGRAPH Symposium Solid Modeling and Application*, pp. 311-321, 1997.

[58] Han J. and Han I. "Manufacturable feature recognition and its integration with process planning", in *Proceeding of 5th ACM SIGGRAPH Symposium Solid Modeling Applications*, pp. 108-118, 1999.

[59] Brousseau E. B., Dimov S. S. and Setchi R. M. "Learning and reasoning techniques for automatic feature recognition from CAD models", *The Manufacturing Engineering Centre*, Cardiff University, Cardiff, CF24 3AA, UK.

[60] Gao S. and Shah J. "Automatic recognition of intersecting machining features based on minimal condition subgraphs", *Computer-Aided Design*, vol. 30, no. 9, pp. 727-739, 1998.

[61] Falcidieno B. and Gainnini F. "Extraction and organization of form features into a structured boundary model," *Eurographics '87, Editor Marechal G.*, pp. 249-259.

[62] Jacquel D. and Salmon J. "Design for manufacturability: a feature-based agent driven approach," *Proceedings C Instn. Mech Engrs.*, vol. 214, part B, 2000.

[63] Lee J. Y. and Kim K. "A feature-based approach to extracting machining features," *Elsevier*, 1999.

[64] Pham D. T. and Dimov S. S. "An approach to concurrent engineering," *Proc Instn Mech Engrs.*, vol. 212 part B, pp. 13-27.

[65] Fu M. W., Lu W. F., Ong S. K., Lee I. B. H. and Nee A. Y. C. "An approach to identify design and manufacturing features from a data exchanged part model," *Computer-Aided Design*, vol. 35(11), pp. 979-993, 2003.

[66] Jami J. S. and Mantyla M. "Parametric and feature-based CAD/CAM," 1995.

[67] Gao S. and Shah J. J. "Automatic recognition of interacting machining features based on minimal condition subgraph," *Elsevier*, 1999.

[68] Davies B. J. "CIM software and interfaces," *Computers in Industry*, vol. 33, pp. 91-99, 1997

[69] Xu X. W. and Ding J. J. "Integrating design with manufacturing," *Proceedings of pacific Conference on Manufacturing*, August 1998.

[70] Yan X., Yamazaki K. and Liu J. "Recognition of machining features and feature topologies from NC programs," *Computer-Aided Design*, vol. 32, pp. 605-616, 2000

[71] Bralla J. J. (Ed.), "*Design for Manufacturability Handbook*", 2nd Edition, McGraw-Hill, New York.

[72] O'Driscoll M., "Design for Manufacture", *Journal of Materials Processing Technology*, vol.122, pp. 318-321, 2000.

[73] Bralla J. J., "*Handbook of Product Design for Manufacturing*", McGraw-Hill Book Company, New York, 1986.

[74] Kalpakjian S., "*Manufacturing Engineering and Technology Third Edition*", Addison-Wesley publishing company, New York, 1995.

[75] Machine shop 1 Drilling machines, "*Cutting speeds and RPM calculations*", Available from, http://its.fvtc.edu, (Accessed on 15th March, 2008).

[76] Machine Shop 3 Milling machines, "*Cutting speeds and RPM calculations*", Available from, http://its.foxvalleytech.com, (Accessed on 17th March, 2008).

[77] Machine Shop 3 Milling machines, "*Feed rates calculations*", Available from, http://its.foxvalleytech.com, (Accessed on 17th March, 2008).

[78] Machine shop 1 Drilling machines, "*Feed rates calculations*", Available from, http://its.fvtc.edu, (Accessed on 15th March, 2008).

[79] Oberg E., Jones F. D., Horton H. L. and Ryffel H. H, "*Machinery's Handbook 27 th Edition* ", Industrial Press Inc. p. 1054, New York, 2004.

[80] Cresse C. R., Adithan M. and Pabla B. S., "*Estimating and Costing for the Metal Manufacturing Industries*", Marcel Dekker, Inc. p. 118, New York 1992.

[81] Cresse C. R., Adithan M. and Pabla B. S., "*Estimating and Costing for the Metal Manufacturing Industries*", Marcel Dekker, Inc. p. 125, table 9-1, New York 1992.

[82] Ostwald P. F., "*American Machinist Manufacturing Cost Estimating Guides*", Mc Graw Hill, P. 382, Edition 1982.

[83] Nicolaou P., Thurston D. L. and Carnahan J. V., "Machining Quality and Cost: Estimation and Tradeoffs", *Manufacturing Science and Engineering*, transaction of ASME, pp. 845, November 2002.

[84] Roughness, "*Terminology*", Available from, http://www.mfg.mtu.edu/ (Accessed on 15th March, 2008).