Automated tolerance verification for process plan generation: A knowledge-based approach.

Raymond. Cheng
University of Windsor

Follow this and additional works at: https://scholar.uwindsor.ca/etd

Recommended Citation
https://scholar.uwindsor.ca/etd/1251

This online database contains the full-text of PhD dissertations and Masters' theses of University of Windsor students from 1954 forward. These documents are made available for personal study and research purposes only, in accordance with the Canadian Copyright Act and the Creative Commons license—CC BY-NC-ND (Attribution, Non-Commercial, No Derivative Works). Under this license, works must always be attributed to the copyright holder (original author), cannot be used for any commercial purposes, and may not be altered. Any other use would require the permission of the copyright holder. Students may inquire about withdrawing their dissertation and/or thesis from this database. For additional inquiries, please contact the repository administrator via email (scholarship@uwindsor.ca) or by telephone at 519-253-3000ext. 3208.
NOTICE

The quality of this microform is heavily dependent upon the quality of the original thesis submitted for microfilming. Every effort has been made to ensure the highest quality of reproduction possible.

If pages are missing, contact the university which granted the degree.

Some pages may have indistinct print especially if the original pages were typed with a poor typewriter ribbon or if the university sent us an inferior photocopy.

Reproduction in full or in part of this microform is governed by the Canadian Copyright Act, R.S.C. 1970, c. C-30, and subsequent amendments.

AVIS

La qualité de cette microforme dépend grandement de la qualité de la thèse soumise au microfilmage. Nous avons tout fait pour assurer une qualité supérieure de reproduction.

S'il manque des pages, veuillez communiquer avec l'université qui a conféré le grade.

La qualité d'impression de certaines pages peut laisser à désirer, surtout si les pages originales ont été dactylographiées à l'aide d'un ruban usé ou si l'université nous a fait parvenir une photocopie de qualité inférieure.

La reproduction, même partielle, de cette microforme est soumise à la Loi canadienne sur le droit d'auteur, SRC 1970, c. C-30, et ses amendements subséquents.
AUTOMATED TOLERANCE VERIFICATION
FOR PROCESS PLAN GENERATION:
A KNOWLEDGE-BASED APPROACH

by

Raymond Cheng

A Thesis Submitted to the
Faculty of Graduate Studies and Research
through the Department of Industrial Engineering
in Partial Fulfillment of the Requirements for the
Degree of Master of Applied Science at the
University of Windsor

Windsor, Ontario, Canada
1991
The author has granted an irrevocable non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of his/her thesis by any means and in any form or format, making this thesis available to interested persons.

The author retains ownership of the copyright in his/her thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without his/her permission.

L'auteur a accordé une licence irrévocable et non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de sa thèse de quelque manière et sous quelque forme que ce soit pour mettre des exemplaires de cette thèse à la disposition des personnes intéressées.

L'auteur conserve la propriété du droit d'auteur qui protège sa thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

I hereby declare that I am the sole author of this thesis.
I authorise the University of Windsor to lend this thesis to
other institutions or individuals for the purpose of
scholarly research.

Raymond Cheng

I further authorize the University of Windsor to reproduce
this thesis by photocopying or by other means, in total or
in part, at the request of other institutions or individuals
for the purpose of scholarly research.

Raymond Cheng
The University of Windsor requires the signature of all persons using or photocopying this thesis. Please sign below, and give address and date.
ABSTRACT

The study develops an Expert System approach to generate alternative process plans for mechanical parts with tolerance requirements. The knowledge base incorporates six factors: process selection, machine specifications, tool selection, machining parameters, fixture selection and process capability. The EXSYS program has been used to manipulate the knowledge base. It starts to extract and verify dimensional and geometrical tolerances for each machining surface. Then, it selects all possible processes, machines and tools. Alternative groups of machines and tools are first verified by the machining parameters required to machine the part. Then, the operation index generated by statistical process control is used to verify the practical feasibility of each of these groups. The potential operations are then grouped to form alternative process plans. Operations in each process plan are finally sequenced using an optimization algorithm and a total cost for all operations is calculated. Application of the program to a mechanical part is illustrated.
To the Lord my God
ACKNOWLEDGMENTS

I wish to express my grateful appreciation to several people for their assistance in this research. Dr. George Abdou, who is my thesis advisor, provided me with excellent guidance and personal encouragement. Thanks go to all other committee members: Dr. S.P. Dutta, Mr. D. Cho, and Dr. Z.F. Reif for their precious suggestions and information. Thanks to Dr. N. Singh (Head of the Department for his support during my graduate study.

Much thanks are extended to Mr. Dieter Liebsch for his assistance in developing the tolerancing and machining techniques of the knowledge base rules. Thanks also to Mr. Tom William for his technical help in programming and to Ms. Jacquie Mummery (Secretary of the Department) for her assistance and cooperation in the preparation of the thesis. Special thanks to Ms. Amy Chiu from English Department for her help in the final preparation of the thesis.

To my parents, to my sister, Helen, and to my housemates, Emmy and Ken, I owe the most sincere debt of gratitude for their support, encouragement, and love.

Finally, I would like to give thanks to Ms. Grace Chan, who has been supporting me through her prayers and concern.
TABLE OF CONTENTS

ABSTRACT ......................................................... vi
DEDICATION ....................................................... vii
ACKNOWLEDGEMENTS .............................................. viii
LIST OF FIGURES .................................................. xi
LIST OF TABLES ................................................... xii

Chapter ......................................................... Page

I. PROBLEM DEFINITION ......................................... 1
II. LITERATURE REVIEW ........................................... 4
III. OBJECTIVES OF RESEARCH .................................... 13

IV. FRAMEWORK OF TOLERANCES IN PROCESS PLANNING ........ 16

4.1 Factors affecting process plan ......................... 17
  4.1.1. Process selection ................................. 18
  4.1.2. Machine selection ................................. 19
  4.1.3. Tool selection .................................. 20
  4.1.4. Machining parameter selection ............... 21
  4.1.5. Jig and Fixture selection .................... 21
  4.1.6. Operation Sequences ........................... 22
  4.1.7. Manufacturing Cost .............................. 25

4.2. Outline of the EXSYS program ......................... 26
4.3. Data required ............................................. 27

V. EXPERT SYSTEM DESCRIPTION ................................. 29

5.1. Databases .................................................. 29
  5.1.1. DXF file ......................................... 29
  5.1.2. Tolerance information file .................... 31
  5.1.3. Process limit files ............................. 33
  5.1.4. Machine data files .............................. 34
  5.1.5. Tool data files ................................ 38
  5.1.6. Machining data file ............................ 41
  5.1.7. Nose radius file relative to surface finish and feed rate . 42
  5.1.8. Jig and Fixture data file ..................... 43
  5.1.9. Database for statistical data from SPC .......... 44

5.2. Qualifiers and variables ................................ 45
5.3. Algorithms and external programs .................... 49
  5.3.1. Algorithms to extract data and verify dimensional and geometric tolerances ......... 49
5.1.1.1.  Data retrieval module  40
5.1.1.2.  Dimensional tolerance verification module  41
5.1.1.3.  Computer-aided geometric tolerance module  50

5.1.1. Surface roughness verification
   program (SURF.VAS, RAS)  54
5.1.1. Process selection (PROCES.RAS)  66
5.1.4.  JOIN.TAS.RAS  67
5.1.5.  FINISH.RAS  10
5.1.6.  Machine selection algorithm  11
5.1.7.  Tool selection algorithm  16
5.1.8.  Machining parameter selection  19
5.1.9.  Holding device selection  27
5.1.10.  Process capability analysis  32
5.1.11.  Alternative process plan
   generation (OPMPLN.RAS)  37
5.1.12.  Operation sequence generation
   (REFERRED.RAS)  37
5.1.14.  Cost estimation program
   (COST.RAS)  37

5.4.  Knowledge base rules
5.4.1.  Rules for process selection
   based on geometric tolerances  31
5.4.2.  Rules for process selection
   based on feature  32
5.4.3.  Rules for process selection
   based on dimension  34
5.4.4.  Rules for combining selected
   processes from CMBR.M and
   PML.(PROGRAM)  35
5.5.  Influence engine

7.1.  APPLICATION OF THE DPM.P System
   THE CASE STUDY  16
6.1.  Description of the sample part  15
6.2.  Dimension extraction and verification  15
6.3.  Process, machine, tool and machining
   parameter selection  15

7.1.  CONCLUSIONS  17

REFERENCES  18
APPENDICES  14
VITA MINimus  14
# List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1</td>
<td>Relationships among factors in process planning with tolerance considerations</td>
<td>17</td>
</tr>
<tr>
<td>4.2</td>
<td>Comparison between the effect of jigs on the roundness of a rotational part</td>
<td>21</td>
</tr>
<tr>
<td>5.1</td>
<td>System components of the expert system</td>
<td>30</td>
</tr>
<tr>
<td>5.2</td>
<td>System diagram for TOLEKXV module</td>
<td>51</td>
</tr>
<tr>
<td>5.3</td>
<td>Relationship between total tolerance of a part and tolerance of an individual surface</td>
<td>54</td>
</tr>
<tr>
<td>5.4</td>
<td>Flow chart of TOLEKXV in case 1 and 2</td>
<td>55</td>
</tr>
<tr>
<td>5.5</td>
<td>Flow chart of TOLEKXV in case 1</td>
<td>57</td>
</tr>
<tr>
<td>5.6</td>
<td>Flow chart of CAGT module</td>
<td>59</td>
</tr>
<tr>
<td>5.7</td>
<td>Flow chart of logic in PROCESS.RAS and JOIN-T&amp;S program</td>
<td>60</td>
</tr>
<tr>
<td>5.8</td>
<td>Flow chart of external program FINISH.RAS</td>
<td>71</td>
</tr>
<tr>
<td>5.9</td>
<td>Flow chart of machine selection algorithms</td>
<td>74</td>
</tr>
<tr>
<td>5.10</td>
<td>Flow chart of tool selection algorithms</td>
<td>77</td>
</tr>
<tr>
<td>5.11</td>
<td>Flow chart of machining parameter selection</td>
<td>81</td>
</tr>
<tr>
<td>5.12</td>
<td>Flow chart of process-capability analysis program (SPC.RAS)</td>
<td>86</td>
</tr>
<tr>
<td>5.13</td>
<td>System flow chart of proposed expert system</td>
<td>99</td>
</tr>
<tr>
<td>6.1</td>
<td>Geometry of the sample part</td>
<td>102</td>
</tr>
<tr>
<td>6.2</td>
<td>New drawing of sample part after tolerance verification</td>
<td>104</td>
</tr>
</tbody>
</table>
### LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-1</td>
<td>Comparison between expert systems in process planning</td>
</tr>
<tr>
<td>4.1</td>
<td>Tolerance and surface finish range for machining processes</td>
</tr>
<tr>
<td>4.2</td>
<td>Comparison between tool characteristics of regular end mills</td>
</tr>
<tr>
<td>5.1</td>
<td>DXF group code for dimensioning in AutoCAD</td>
</tr>
<tr>
<td>5.2</td>
<td>Description of data in tolerance information file</td>
</tr>
<tr>
<td>5.3</td>
<td>Data in PROCESS.DAT file</td>
</tr>
<tr>
<td>5.4</td>
<td>Data structure for LATHE.DAT file</td>
</tr>
<tr>
<td>5.5</td>
<td>Data structure for MCCENTER.DAT file</td>
</tr>
<tr>
<td>5.6</td>
<td>Data in LATHE.BAS file</td>
</tr>
<tr>
<td>5.7</td>
<td>Data in MCCENTER.BAS file</td>
</tr>
<tr>
<td>5.8</td>
<td>Data structure of INSERT.DAT file</td>
</tr>
<tr>
<td>5.9</td>
<td>An example of INSERT.DAT file</td>
</tr>
<tr>
<td>5.10</td>
<td>Data structure of MILLS.DAT file</td>
</tr>
<tr>
<td>5.11</td>
<td>An example of MILLS.DAT file</td>
</tr>
<tr>
<td>5.12</td>
<td>Tool accuracy for general purpose twist drills and reamers</td>
</tr>
<tr>
<td>5.13</td>
<td>The format of machining data file</td>
</tr>
<tr>
<td>5.14</td>
<td>An example of nose radius data file</td>
</tr>
<tr>
<td>5.15</td>
<td>An example of holding device data file</td>
</tr>
<tr>
<td>5.16</td>
<td>Data in SPC.DAT file</td>
</tr>
<tr>
<td>5.17</td>
<td>List of qualifiers in the expert system</td>
</tr>
<tr>
<td>5.18</td>
<td>List of variables in the expert system</td>
</tr>
</tbody>
</table>
5.19 Relative values of surface finish with usual tolerances .... 65
5.20 Data location of the COST.BAS program ........ 89
5.21 Output of COST.BAS program ...................... 90
5.22 Corresponding machining processes for each feature ............. 93

6.1 DIMTOL.DAT from TOLEXV module ................. 103
6.2 Verified operation generated from PARA.BAS .. 106
6.3 Remaining alternative operations after process capability analysis ............... 107
6.4 List of operations in both PREPLANS .............. 107
6.5 Summary of alternative process plan ............... 109
6.6 Process plan for each alternative process plan ............. 110
CHAPTER I

PROBLEM DEFINITION

Process planning, which is one of the main functions in manufacturing, is the link between the design function and the manufacturing function. It plays an important role in the implementation of computer-integrated manufacturing (CIM) system. It is supposed to define the optimum production methods for mechanical parts in strict accordance with tolerance requirements. The overall process of production planning consists of many decisions related to selection of machine, tool, operation sequence and verification of design tolerances.

Tolerances are important since they define the acceptable manufacturing error. Tolerance technology is a wide-ranging topic. It can be divided into two areas, tolerance design and tolerance control. Tolerance design is a technique to assign optimum tolerances in the design stage for the least manufacturing cost. Tolerance control is performed for ensuring the existing machines and tools to produce a part within the specified tolerances.

The specification of tolerances on the dimensions of manufacturing parts has a significant impact on the final production cost. Tight tolerances can result in excessive process costs, while loose tolerances may lead to increased waste and assembly problems. Nowadays, improper tolerances are often assigned by less experienced designers, regardless
of the machine capabilities and process feasibility. Since geometric tolerancing involves the understanding of the functionality of a part and of the manufacturing constraints from part designers, geometric tolerances are not often correctly assigned. Moreover, people in manufacturing areas also ignore the effects of tolerances on manufacturing processes. Since improper design tolerances have not been detected before the process plans are developed, the existing process planning systems still cannot generate any efficient process plan. Consequently, high production cost, (including machining, tooling, and labor costs), may be involved in the production due to the presence of unnecessarily close tolerances or rejected products without appropriate tolerances which have not been verified.

In addition, the gap between CAD (Computer-Aided Design) and CAPP (Computer-Aided Process Planning) has not been resolved. Since existing CAD systems do not have the capability of storing technological characteristics, such as geometric tolerances and surface finish, human interaction for transferring data between CAD databases and CAPP systems is still required.

Therefore, tolerance verification is necessary to make sure whether the right tolerances are properly assigned before the machines and the tools are selected to achieve the design tolerance. So, tolerance verification in CAD, which involves a lot of knowledge and experience, is a task in between tolerance design and tolerance control.
Finally, process planning is an unstructured problem which lacks the required knowledge. It is nonprogrammable because its nature and structure are elusive or complex. Many knowledge-based expert systems in process planning are now being developed to support this unstructured task. However, there is only few systems considered the tolerances in generating more process plans.
CHAPTER II

LITERATURE REVIEW

The tolerance design in CAD has received some attention in the literature. Most studies have focused on the importance of tolerance verification. They have attempted to highlight the significance of tolerance in manufacturing and to apply various techniques for tolerance design.

Fainguelernt, et al. [1], mentioned that determination of manufacturing tolerances is one of the most important issues in production planning. They proposed a strategy of "tolerance transfer" to optimize tolerance ranges according to functionality and equipment capability by means of computerized tolerancing and dimensioning method. The developed computer program provides higher reliability of tolerance transfer and shorter processing time than that of manual tolerancing.

Weill [2] applied this strategy and developed an optimization module to ensure that consideration of tolerance is effective to process planning. However, this module was limited to one-dimensional tolerancing. Weill [3] also emphasized that the knowledge of geometrical tolerancing would be a powerful means to control manufacturing processes, and to avoid process and machine inaccuracies. The fact is because tolerances are used to
represent functional requirements of design for a process planner to select production process and tooling. However, knowledge of the designer about process capabilities is not available, so the need of developing technological data banks for this purpose is clearly felt. He also recommended that standards and technical rules of dimensional and geometrical tolerancing must also be performed by means of more intelligence of computer technology.

Recently, more researches have focused on the expert system approach in process planning. The state-of-the-art survey in computer-aided process planning was presented by Alting and Zhang [4]. At the end of 1980s, knowledge based expert systems were considered to have the greatest potential for the development of CAPP systems. Some of the expert process planning systems had been developed, and their characteristics are discussed as follows:

**PROPLAN** is an expert process planning system suitable for symmetric rotational parts. It uses a CAD system database as a primary source of data input. The knowledge base consists of production rules. Since PROPLAN employs a graph search strategy to generate a process plan, the system is not suitable for complex parts.

**EXCAP** system has been designed to generate process plans for simple rotational parts. The system generates a 'tree' of a possible sequence of operations and the machining parameters by a backward chaining inference strategy. EXCAP can also produce more than one process plan
for users to choose. However, it has not been interfaced with any CAD models.

SIPP is a knowledge base that includes information about the characteristics of different machinable surfaces and the capabilities of the various machining processes. The information is represented by using hierarchical frames. The system considers the process with its tolerance and geometric restrictions. Using a least-cost-first search strategy, the system will choose the one with the least cost.

TIPPS is a generative process planning system implemented by Chang and Wysk [5]. It integrates a CAD database which represents an object by surfaces and boundaries. TIPPS allows the user to modify the surface identification interactively, because tolerance information is not available in the CAD database. The system gives a feasible sequence of processes, tools, and machining parameters as a report.

Zhang and Alting [6] developed an expert process planning system XPLAN-R for rotational parts. The imperfection of this expert system is that it cannot retrieve technical information for the part drawing. For example, batch size, material properties, dimensions, tolerances and roughness, and geometrical features, etc., are required to be entered interactively from the users.

Joshi et al. [7] presented an expert process planning system with a solid model interface. However, the output of
that CAD interface provides all the information necessary for process planning except the tolerancing information which has been ignored in most of the solid modeling. Tolerance, datum, and surface finish information are augmented manually with the CAD output. A system which can generate and store this missing information is expected from further research.

The current development and the future of the expert system approach were reviewed by Gupta [8]. He recommended that geometric reasoning should form the basis for the CAD interpreter and tolerance information is important for process selection. Wang and Wysk [9-10], who introduced Turbo-CAPP as an intelligent process planning system, attempted to provide a solution in these two areas. The Tolerance Input Module (TIM1) in Turbo-CAPP is a knowledge acquisition procedure for tolerance information. However, this module only allows the user to input and modify the dimensional and geometrical tolerances. Even though a frame of product qualification (surface finish and tolerances) knowledge is available for storing this tolerance information through the CAD database by the user, the improperly assigned tolerances are still not detected by less experienced process planners. Therefore, they concluded that a generalized algorithm for automatic extraction of surfaces features from CAD database and an automated tolerance verification mechanism still need to be developed.

Since the knowledge base in Turbo-CAPP is not
sufficient, certain aspects of the concept are highlighted along with possible modifications to generate a more efficient process plan. In Turbo-CAPP, frames and production rules are defined first as the two types of knowledge formalism schemes for process planning. The frames are used for declarative facts which usually identify the object with its characteristics. Five different types of frames included the facts about the workpiece qualification, machining process, machine, tool, and holding device. Another type of knowledge is represented in the form of rules (procedural knowledge), such as the rules for surface feature extraction, selection of machining operation, operation sequence, machine, tool, jig & fixture, and machining parameters. The frame-based representation of knowledge provides a means for coupling declarative knowledge with its procedural rival. The frame structure for declarative knowledge is represented by means of a number of restrictions and conditions.

The rule-based representation or production system is a knowledge base which consists of production rules, in the form of premise-action pairs. Rules are expressed in the form of IF (premise), THEN, and ELSE (action). If all of the premises are true, then the actions are considered to be true and added to what the program knows. If any of the premises is false, then the rule is considered to be false, and any statements in the ELSE part are then taken to be true and added to what the program knows.
Since a detailed explanation of the frame is not found, several problems occur and thus, generate an inefficient process plan. An example of a knowledge frame is shown as follows.

A Frame for Describing Rough_Bore
Process Name: Rough_Bore
List of Conditions:
1. smallest_dia(H) = 0.375
2. largest_dia(H) = 10
3. pos_dia_tol(H) > 0.0003
4. neg_dia_tol(H) > 0.0003
5. straightness(H) > 0.0003*(len/dia^3 + 0.0010)
6. roundness(H) > 0.002
7. parallelism(H) > 0.0004*(len/dia^3 + 0.0015)
8. true_position(H) > 0.0002
9. sur_fini(H) > 30

First, the formulae for calculating geometric tolerance limits, shown in the frame, are based on Wysk [11]. The process boundary theory provides an empirical formula for the dimensional and geometric tolerances. However, the formulae define the values of dimensional and geometric tolerances with constants or as the function of part length (len) and the diameter (dia). No reference has been provided for the minimum allowable tolerance values. In fact, geometric tolerance is also strictly related to the functionality and the dimensional tolerances.

Second, the quality of surface finish is commonly specified along with tolerances. This is becoming more common as product demands increase because surface quality often determines how well a part performs. According to the frame given as above, if a given surface has a surface finish which is 100 times greater than the specified lower limit, and dimensional tolerance is just about the minimum
tolerance limit, then it means that there is a mismatch between surface finish and dimensional tolerance given from the design.

Third, the frame for rough boring gives an unclear technological characteristic for that machining process. Considering the limits of the dimensional tolerances specified in the frame, the availability for a process is presented arbitrarily. For instance, the frame for describing rough boring implies that this process can be adopted for all the surfaces specified with the positive dimensional tolerance which is greater than 0.0003 mm. Since a lot of machining processes may be selected other than rough boring, the missing upper limit for the dimensional tolerances will cause an effect that more machining processes will be recommended but not all are suitable. If the specified tolerance on the surface is large, then it may not be necessary to choose rough boring as a machining process, some other processes may also be suitable and more economical. Therefore, upper limit for dimensional tolerance which is missing in the frame is also necessary for choosing efficient processes.

Finally, Turbo-CAPP only applies tolerance information for process selection and operation sequencing. In fact, tolerance information is also a piece of valuable information and a criterion for selecting machines, tools, machining parameters, and holding devices.

A comparison among different expert systems for process planning in today's industries is shown in Table 2.1. There
are seven criteria in the comparison. They are classified as the input requirements (part shape, CAD involvement and dimension) and the output performances (knowledge representation and tolerance verification) of the expert system. As the objective of this study, a proposed expert system is also compared with the existing expert systems with respect to the criteria. The proposed expert system will apply production rules, algorithms, databases, and statistical analysis to generate more efficient process plan with tolerance verification.

As a summary, these publications have provided insight into the tolerance verification in CAD. The researches show that a tolerancing system requires standardized information to assist the user, and has to be user-friendly. However, automatic feature extraction including storage of technical information is necessary for the tolerancing system.

Therefore, in order to develop a generalized algorithm which is more applicable to real life industry, further studies are required.
<table>
<thead>
<tr>
<th>EXPERT SYSTEM</th>
<th>Task Shape</th>
<th>CAD Involvement</th>
<th>Dimension</th>
<th>Knowledge Representation</th>
<th>Dim. Tol.</th>
<th>Geo. Tol.</th>
<th>Verify Tolerance</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROPLAN (1984)</td>
<td>Rot.</td>
<td>CAD Database</td>
<td>2D</td>
<td>Rules</td>
<td>NO</td>
<td>NO</td>
<td>NO</td>
<td>Inefficient for complex parts</td>
</tr>
<tr>
<td>EXCAP (1984)</td>
<td>Rot.</td>
<td>No</td>
<td>2D</td>
<td>Rules</td>
<td>NO</td>
<td>NO</td>
<td>NO</td>
<td>Not interfaced with CAD, not good for complex parts</td>
</tr>
<tr>
<td>SIPP (1985)</td>
<td>n/a</td>
<td>n/a</td>
<td>3D</td>
<td>Frames</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Use tolerances for process selection</td>
</tr>
<tr>
<td>TIFPS (1985)</td>
<td>Rot. &amp; prismatic</td>
<td>CAD Database</td>
<td>3D</td>
<td>Rules</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Interactively tolerance input</td>
</tr>
<tr>
<td>TURBO-CAPP (1987)</td>
<td>Symmetric rot.</td>
<td>CAD Database</td>
<td>2D</td>
<td>Rules</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Verification of tolerances by user only, no standard, tolerance has no effect on machine, tool, machining parameter selection</td>
</tr>
<tr>
<td>XPLAN-R (1988)</td>
<td>Rot.</td>
<td>No</td>
<td>2D</td>
<td>Hybrid (rules/frames)</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>CAD is not integrated for feature extraction</td>
</tr>
</tbody>
</table>

n/a: information not available
CHAPTER III

OBJECTIVES OF RESEARCH

Three areas are considered to be important in the tolerancing problems which have not been properly investigated in the previous studies. The first area concerns the extraction of surfaces and tolerance information from CAD database, i.e. an efficient CAD/CAPP integration. The second area concerns the implementation of dimensional and geometrical tolerances verification in CAD. The third area concerns the design of the rule-based expert system in process planning.

1. Extract dimensions, tolerances, and surfaces from CAD database. Since the dimensions and tolerances of CAD drawings are not stored uniquely in a database for process planning, extraction module is a first module to be developed for CAD/CAPP integration. A CAPP can automatically retrieve tolerance information from the database to generate a process plan.

2. Implement dimensional and geometrical tolerance verification in CAD. A tolerance verification module has been developed in a CAD system. Tolerance verification is based on calculation of stackup and allocation of dimensional tolerance, and CSA standards in geometrical tolerancing.
3. Design a rule-based expert system in process planning. A prototype of an expert system for process planning will be designed with the involvement of surface qualification, i.e. tolerance and surface finish, into the selection of processes, machines, tools, machining parameters and holding devices. This knowledge will be represented by production rules in an expert system.

The procedures of this study consisted of the following steps:

1. Develop a program to create a tolerance information file by extracting each entity as a surface, its dimension and tolerance from the CAD database.

2. Develop a program for verification of dimensional tolerances and to gather more related information about workpiece's raw material and surface finish.

3. Design a computer-assisted geometric tolerancing module based on functionality which is using the rules from CSA standards to recommend some reasonable tolerance ranges for the user.

4. Construct a knowledge base structure in logical flow for process planning.

5. Survey the knowledge in selection of machining processes, machines, tools, machining parameters and holding device with the consideration of tolerances.

6. Develop rules and external programs to support the process planning task in the knowledge base.
7. Develop process capability analysis and cost evaluation for the generated alternatives from the expert system.

8. Demonstrate the feasibility of the proposed expert system through a case study in process planning.
CHAPTER IV

FRAMEWORK OF TOLERANCES IN PROCESS PLANNING

Tolerance is used to show acceptable variations in dimensions. Since it is required to produce the exact dimension of a part within certain variations specified by the product designer, tolerancing is very important for the manufacturer. Production engineers are generally aware of the significance between design tolerances and manufacturing costs. However, from the designers' point of view, they may tend to be conservative and specify tolerances that are too tight. So, if the production engineers discover the specified tolerances that are difficult to be achieved during manufacturing, they refer the situation back to the designer. Therefore, loss of precious time and increase of the costs will happen. The reason for the high cost is the lack of the designers' awareness of the cost implications of the tolerance specification and the available machine and process capabilities. Therefore, tolerances should not be considered only on the functional point of view at the design stage, but also on the manufacturing point of view at the production stage.
4.1 Factors Affecting Process Plan

Based on many studies in the relationship between tolerances and process planning, there are many factors which are highly affected by the tolerances in production process. A chart showing the relationships between tolerances and factors in process planning is presented in Figure 4.1. Factors which are directly and indirectly affected by tolerances are shown as follows:

1. Machining processes
2. Machines
3. Tools
4. Machining parameters
5. Jigs and fixtures
6. Sequence of operations
7. Manufacturing cost

Figure 4.1 Relationships among factors in process planning with tolerance verification
4.1.1 Process Selection

Every machining process produces a certain type of surfaces at a certain performance level. The performance level can be represented by tolerances and nominal surface finish. Table 4.1 given from Jones [7] shows that each process is capable of producing parts within certain limits of tolerances and surface finishes. Some processes at their best can compete with another process, but can never match the results of the best of those processes. For instance, according to the given limits of dimensional tolerances in machining processes, a hole required with just a rough finish and a wide tolerance range of 0.005 can be produced by drilling on a drill press. However, if a larger and a tighter tolerance range of hole diameter is specified, milling operation should be chosen rather than drilling.

Apart from dimensional tolerances, some characteristics of geometric tolerances, like straightness, roundness, concentricity and parallelism of the part shape are directly related to the performance capability of some machining process. For example, finish boring can only produce an internal surface with concentricity which is greater than 0.0002 inch and roughness of internal diameter with 0.0002 inch.
Table 4.1  Tolerance and surface finish for machining processes

<table>
<thead>
<tr>
<th>MACHINING PROCESS</th>
<th>TOLERANCE (inch)</th>
<th>SURFACE FINISH (micro-inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. BORING</td>
<td>± 0.001</td>
<td>63</td>
</tr>
<tr>
<td>2. BROACHING</td>
<td>± 0.002</td>
<td>63</td>
</tr>
<tr>
<td>3. COUNTERBORING</td>
<td>± 0.005</td>
<td>94</td>
</tr>
<tr>
<td>4. DRILLING</td>
<td>+ 0.005, - 0.002</td>
<td>125</td>
</tr>
<tr>
<td>5. GRINDING</td>
<td>± 0.001</td>
<td>16</td>
</tr>
<tr>
<td>6. HOBBING</td>
<td>± 0.010</td>
<td>94</td>
</tr>
<tr>
<td>7. MILLING</td>
<td>± 0.010</td>
<td>94</td>
</tr>
<tr>
<td>8. REAMING</td>
<td>± 0.001</td>
<td>63</td>
</tr>
<tr>
<td>9. THREADING</td>
<td>± 0.003</td>
<td>125</td>
</tr>
<tr>
<td>10. TURNING</td>
<td>± 0.005</td>
<td>94</td>
</tr>
</tbody>
</table>

4.1.2. Machine Selection

In the process planning procedures, the process selection is followed by the machine selection. In order to fulfill the requirements of the part designer, the process planner has to select a machine based on the technological characteristics, such as dimensional and geometrical tolerances and surface finish. Every machine has its specification of accuracy in machining process. Since any movement on each axis will cause certain mechanical inaccuracy, its mechanical inaccuracy should not be greater than the allowable variation of size of a given part; otherwise, the part will be rejected. On the other
hand, it is not economical to use a machine with high precision for high machining cost to produce just a part where tolerance is considered not important. Therefore, without considering the process capability of the machine based on the tolerance requirement of the part, the generated process plan is inefficient.

4.1.3. Tool Selection

Cutting tool is a critical element in machining. The minimum deviation for cutting a surface and the process capability can be determined by combining the accuracy of the machine and cutting tool. In fact, every tool has its tolerance on its geometry. For instance, a 1/4 inch diameter drill bit usually has a certain tolerance of 0.0005 inch. The tolerances vary based on the drill size as listed in tool's handbooks [12]. An example derived from Machinery's handbook [15] demonstrates the difference between the tool accuracy of two types of half inch regular end mills as shown in Table 4.2.

<table>
<thead>
<tr>
<th>TOOL CHARACTERISTIC</th>
<th>DIMENSION S W L</th>
<th>TOLERANCE S W L D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Multiple flute</td>
<td>3/8 1 43/16</td>
<td>0.0005 1/32 1/16 0.005</td>
</tr>
<tr>
<td>Two flute</td>
<td>1/2 5/4 13/4</td>
<td>0.0005 1/32 1/16 0.003</td>
</tr>
</tbody>
</table>

When the flatness of the surface is given as 0.004 inch, the two flute high speed end mill is relatively better than
another one. On the other hand, it is unnecessary to pay a high tooling cost for just machining a surface where tolerance is not important. Therefore, justification of tool tolerance is economical.

4.1.4 Machining Parameter Selection

There is a rule of thumb saying that a fine surface can be cut by means of high spindle speed and low feed rate. Moreover, the relationship between feed rate and surface finish required on the part can be found in the nomograph given from the machining handbook [12], and a relationship between surface finish and tolerance has also been given by Doyle [13]. The finer the finished surface, the smaller the tolerance on the workpiece should be maintained.

4.1.5 Jig and Fixture Selection

Basically jigs are used in hole-making processes while fixtures are used in flat surface making. The shape, basic dimensions, and especially the technological requirements of a workpiece are the basic issues in selecting an appropriate holding device. Given an example of a rotational part, machining of holes with enlarging by boring is required. If the roundness of the diameter of the rotational part is specified in a small value, the selected jig should not cause any significant deformation on the external and internal surfaces of the part in order to avoid non-circular shape in the internal
surface. An example with a microscopic analysis to the external surface of a rotational part is presented. A diagram on Figure 4.2 shows the effects of jig's screws on the roundness of the rotational part. More screws to hold the part in the jig will affect the circularity of the part. As the design requirement for the roundness of the part is 0.001 inch, the choice of a holding device is critical in determining whether to accept or reject the finished parts. Therefore, different types of fixture design have to be considered when certain values of tolerances are required to be met.

4.1.6 Operation Sequences

Operation Sequence is one of the steps which follows the selection of process, machine and cutting tools in process planning. There are several general rules being established that will assist a production-design engineer to plan an optimum operation sequence. They are listed as follows:

a) Operations involving close dimensional tolerances or fine surface finishes are usually done last in the process sequence. Since both tolerance and surface finish of surfaces can be adversely affected by subsequent clamping, material handling, and processing, those operations that provide fine finishes and/or close tolerances should be performed as late as possible in the operation sequence. For example, a part held to 0.0001-in tolerance can subsequently be rejected if it is merely
dropped or severely bumped, because the impact can cause the part to lose its dimensional accuracy.

Figure 4.2  Comparison between the effect of jigs on the roundness of a rotational part.
b) It is almost always desirable to perform rough work involving heavy cuts and liberal tolerances early in the sequence. Heavy cuts will reveal defects in the raw material (usually castings or forgings) much more readily than light cuts. Furthermore, heavy cuts involving coarse finishes are usually performed faster with less-expensive workmen than fine finishes. It is also advantageous to find out that work is being performed on defective material as soon as possible with the least investment in secondary processes.

c) Internal operations are performed in advance of external operations. The principal reason for performing internal operations early is that internal surfaces are less likely to be damaged in material handling and subsequent processes, so, their surfaces can be completed earlier in the sequence. Another reason is that internal surfaces frequently provide a better means of holding the work and thus help ensure concentricity between inside and outside diameters. When internal work is performed, the logical sequence of operation is drilling, boring, recessing, reaming, and tapping. The logical sequence of external work is turning, facing, grooving, forming, and threading.

Of course, there are some other rules to assist the sequencing of operations. For example, it is usually desirable to do as much work as possible once the work is located and held in a jig or fixture. For example,
if there are two surfaces of a rectangular part with a strict parallelism in geometric tolerances, they must be machined in the same setup in order to reach the tolerance requirement. Besides, if two holes are stated with a close concentricity in geometric tolerance, it is desirable to keep the part in the same drill jig during the drilling operations of both holes, and one drilling operation should follow the other in order to minimize the tie-up of tooling.

4.1.7 Manufacturing Cost

Loose tolerances are held more easily and economically than tight tolerances. However, if the tolerances of a part are increased by 100%, the part will not function as originally designed. Therefore, if design tolerances are maximized based on the functions of the part, the following cost savings may result from:

a) reduction in the number of manufacturing operations,
b) reduction of scrap and rework,
c) reduction of processing time,
d) reduction of tooling costs by higher tool tolerances,
e) reduction of cost in gauging for inspection,
f) reduction of labor costs for highly skilled men.

With the consideration of tolerances on the process, machine, tool, and holding device selection, a certain amount of money will be saved by a more efficient process plan. It is because unnecessary machining processes, machines, cutting tools and/or fixtures can be eliminated by the tolerance consideration in process plan.
4.2 Outline of the EXSYS Program

The Expert System that will be adopted is the EXSYS system. EXSYS is a program for manipulating expert system knowledge bases. It is a rule-based, backward-chaining, goal directed development tool. It is written in C and runs on an IBM or compatible computer. EXSYS has the ability to call external programs to carry out a wide range of calculation tasks. The external program may pass data to EXSYS from/to a database or spreadsheet and run a custom program for graphics or calculations. Upon completion of the calculations, EXSYS displays all goal parameters and their respective resultant probabilities.

The system involves three steps which are shown as follows:

1. Creating a text base, to include:
   (a) subject of knowledge base
   (b) choices of data structure
   (c) number of rules to be used in data derivation
   (d) input text to explain the created expert system
   (e) input text for recommendations
   (f) choices of rules to be displayed
   (g) choice to call the external program
   (h) choice of decisions
   (i) checking new rules against the previous one

2. Creating a rule base.

   The way that rules are built in EXSYS is different from those in most other expert system development tools. The knowledge engineer creates qualifiers (like parameters) and their respective values. Once qualifiers are created, they can be used repeatedly by simple recall. Upon defining the qualifiers, rules can then be constructed. Each rule
is expressed in the form of "IF, THEN, ELSE", and consists of:

(a) the name of the rule which instructs the development engine to insert a new rule into the knowledge base,
(b) the premise "IF" of the rule,
(c) the action "THEN" of the rule in which the knowledge engineer assigns a probability value to the choices designated in Step 1.

3. Defining a flag variable that indicates whether or not a rule is activated, and changes to be incorporated when the rule is activated.

If all of the premises are true, then the rule is considered to be true and the actions are considered to be true and added to what the program knows. If any of the premises is false, the rule is considered false, and any statements in the ELSE part are then taken to be true and added to what the program knows.

4.3 Data Required

The expert system requires a lot of information to generate alternatives for the user to make a decision. Some data are directly given by the user, while some data are extracted implicitly from an external program and inputted into the expert system.

Data from user are:

1. Type of workpiece material and tool material.
2. Hardness of workpiece material.
3. Decimal places of dimensions and tolerances.
4. Dimensions and tolerances of the part drawing.
5. Minimum clearances of the part.
6. Surface finishes on the surfaces when needed.
7. Dimension and its tolerance of the matching parts.
8. Features of the part.
9. Functionality of the part or features. The user answers a set of questions about assembly and the geometric requirement for the part.
10. Production rate and volume.
CHAPTER V

EXPERT SYSTEM DESCRIPTION

The structure of the EXSYS system for process planning generation is shown in Figure 5.1. The components of the expert system are discussed as follows.

5.1 Databases

It consists of data files, multiple qualifiers and variables, algorithms in process planning, and external programs for data searching and calculations.

5.1.1 DXF Files

PART.DXF (Data eXchange File) file is an ASCII file which all the geometry information, specifically dimensions and their tolerances, are stored. The variable names of the data are identified by codes in the DXF file. The variable is retrieved by its corresponding code. The code descriptions for dimensioning information in the DXF file are listed in Table 5.1. The most important information given from the codes are the dimension and the tolerance values, and the coordinates.
Figure 5.1 System components of the computer aided process planning system with expert system
Table 5.1  DXF Group Code for Dimensioning in AutoCAD

<table>
<thead>
<tr>
<th>CODE</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>&quot;Dimension&quot; Statement</td>
</tr>
<tr>
<td>1</td>
<td>Text (Dimension &amp; Dimensional Tolerance)</td>
</tr>
<tr>
<td>2</td>
<td>Name of the block being measured</td>
</tr>
<tr>
<td>8</td>
<td>Layer Name</td>
</tr>
<tr>
<td>10</td>
<td>X-definition point</td>
</tr>
<tr>
<td>11</td>
<td>X-coordinate of middle point of dimension</td>
</tr>
<tr>
<td>12</td>
<td>X-coordinate of insert point</td>
</tr>
<tr>
<td>13</td>
<td>X-coordinate of definition point</td>
</tr>
<tr>
<td>14</td>
<td>X-coordinate of definition point</td>
</tr>
<tr>
<td>15</td>
<td>X-coordinate of definition point</td>
</tr>
<tr>
<td>16</td>
<td>X-coordinate of definition point</td>
</tr>
<tr>
<td>20</td>
<td>Y-coordinate of definition point</td>
</tr>
<tr>
<td>21</td>
<td>Y-coordinate of middle point of dimension</td>
</tr>
<tr>
<td>22</td>
<td>Y-coordinate of insert point</td>
</tr>
<tr>
<td>23</td>
<td>Y-coordinate of definition point</td>
</tr>
<tr>
<td>24</td>
<td>Y-coordinate of definition point</td>
</tr>
<tr>
<td>25</td>
<td>Y-coordinate of definition point</td>
</tr>
<tr>
<td>26</td>
<td>Y-coordinate of definition point</td>
</tr>
</tbody>
</table>

5.1.2  Tolerance Information File

DIMTGL.DAT is generated by the Data Retrieving Module which will be described in the external program section. Three types of tolerance information are generated in the file:

a) Data that are retrieved from PART.DXF;
b) Data that are created from tolerance verification module;
c) Data that are entered by user.

Description of the variables in the file is presented in Table 5.2.
<table>
<thead>
<tr>
<th>VARIABLE_NAME</th>
<th>DESCRIPTION</th>
<th>ENTRY FROM</th>
</tr>
</thead>
<tbody>
<tr>
<td>N</td>
<td>Total number of surfaces</td>
<td>TOLEXV module</td>
</tr>
<tr>
<td>I</td>
<td>Surface number</td>
<td>TOLEXV module</td>
</tr>
<tr>
<td>D(I)</td>
<td>Dimension Value</td>
<td>DXF file</td>
</tr>
<tr>
<td>T(I)</td>
<td>Dimensional Tolerance Value</td>
<td>DXF file</td>
</tr>
<tr>
<td>STATE(I)</td>
<td>Type of Dimension (L/D)</td>
<td>TOLEXV module</td>
</tr>
<tr>
<td>X1(I)</td>
<td>X-coord. of 1st ext. point</td>
<td>DXF file</td>
</tr>
<tr>
<td>X2(I)</td>
<td>X-coord. of 2nd ext. point</td>
<td>DXF file</td>
</tr>
<tr>
<td>Y1(I)</td>
<td>Y-coord. of 1st ext. point</td>
<td>DXF file</td>
</tr>
<tr>
<td>Y2(I)</td>
<td>Y-coord. of 2nd ext. point</td>
<td>DXF file</td>
</tr>
<tr>
<td>SR(I)</td>
<td>Surface Roughness</td>
<td>TOLEXV</td>
</tr>
<tr>
<td>MINCL(I)</td>
<td>Minimum Clearance</td>
<td>TOLEXV</td>
</tr>
<tr>
<td>MAXST(I)</td>
<td>Maximum Straightness</td>
<td>CAGT module</td>
</tr>
<tr>
<td>MINST(I)</td>
<td>Minimum Straightness</td>
<td>CAGT module</td>
</tr>
<tr>
<td>MAXCI(I)</td>
<td>Maximum Circularity</td>
<td>CAGT module</td>
</tr>
<tr>
<td>MAXCY(I)</td>
<td>Maximum Cylindricity</td>
<td>CAGT module</td>
</tr>
<tr>
<td>MINCY(I)</td>
<td>Minimum Cylindricity</td>
<td>CAGT module</td>
</tr>
<tr>
<td>MAXCO(I)</td>
<td>Maximum Concentricity</td>
<td>CAGT module</td>
</tr>
<tr>
<td>MINCO(I)</td>
<td>Minimum Concentricity</td>
<td>CAGT module</td>
</tr>
<tr>
<td>LENGTH</td>
<td>Total Length of Part</td>
<td>TOLEXV</td>
</tr>
<tr>
<td>T_PART</td>
<td>Dimensional Tolerance of total length of the part</td>
<td>TOLEXV</td>
</tr>
<tr>
<td>RMLENG</td>
<td>Length of Raw Material</td>
<td>TOLEXV</td>
</tr>
<tr>
<td>RMTOL</td>
<td>Tolerance of Raw Material</td>
<td>TOLEXV</td>
</tr>
</tbody>
</table>
5.1.3 Process Limit Files

PROCESS.DAT is a data file which consists of a number of machining processes with both upper and lower limits of dimensional tolerances and surface roughness that are frequently applied in industry. The data for PROCESS.DAT, which are derived by Doyle [13], are presented for the general type of products in Table 5.3.

<table>
<thead>
<tr>
<th>MACHINING PROCESS</th>
<th>DIMENSIONAL TOLERANCE LIMIT</th>
<th>SURFACE ROUGHNESS LIMIT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>LOWER (in)</td>
<td>UPPER (in)</td>
</tr>
<tr>
<td></td>
<td>LOWER (µ in)</td>
<td>UPPER (µ in)</td>
</tr>
<tr>
<td>HONING</td>
<td>0.00005</td>
<td>0.00020</td>
</tr>
<tr>
<td>F_GRINDING</td>
<td>0.00017</td>
<td>0.00170</td>
</tr>
<tr>
<td>R_GRINDING</td>
<td>0.00170</td>
<td>0.00200</td>
</tr>
<tr>
<td>REAMING</td>
<td>0.00050</td>
<td>0.00200</td>
</tr>
<tr>
<td>BROACHING</td>
<td>0.00500</td>
<td>0.00200</td>
</tr>
<tr>
<td>F_BORING</td>
<td>0.00100</td>
<td>0.00500</td>
</tr>
<tr>
<td>R_BORING</td>
<td>0.00400</td>
<td>0.01000</td>
</tr>
<tr>
<td>F_TURNING</td>
<td>0.00170</td>
<td>0.01000</td>
</tr>
<tr>
<td>R_TURNING</td>
<td>0.01000</td>
<td>0.02000</td>
</tr>
<tr>
<td>MILLING</td>
<td>0.00170</td>
<td>0.02000</td>
</tr>
<tr>
<td>DRILLING</td>
<td>0.00200</td>
<td>0.10000</td>
</tr>
<tr>
<td>HAND_GRINDING</td>
<td>0.01700</td>
<td>0.05000</td>
</tr>
<tr>
<td>FLAME_CUTTING</td>
<td>0.05000</td>
<td>0.10000</td>
</tr>
</tbody>
</table>
5.1.4 Machine Data Files

Two database files [14], in which two classes of machines were defined, had been developed. The database includes machines which accommodate parts up to 4 feet long and run the motors in maximum power up to 30 hp. LATHE.DAT file includes the CNC lathes while MCCCENTER.DAT file includes the machining centers. All the turning operations are assigned to CNC lathes. The others will be assigned to machining centers. If the machine specifications satisfy the workpiece requirements, the machines will be selected from the data file. Accuracies of machining axis are also considered when the dimensional and geometric tolerances are given from the design of workpiece.

The data structures of LATHE.DAT and MCCCENTER.DAT files are presented in Tables 5.4 and 5.5, respectively. The lists of 15 lathes and 9 vertical machining centers with their corresponding parameters are illustrated in Tables 5.6 and 5.7, respectively.
### Table 5.4  
**Data Structure for LATHE.DAT file**

<table>
<thead>
<tr>
<th>FIELD NAME</th>
<th>UNIT</th>
<th>VARIABLE NAME</th>
<th>FIELD SPACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Machine Code</td>
<td>----</td>
<td>CODE(m)</td>
<td>4</td>
</tr>
<tr>
<td>Machine Name</td>
<td>----</td>
<td>NAME(m)</td>
<td>30</td>
</tr>
<tr>
<td>Maximum Turn Diameter</td>
<td>inch</td>
<td>MTD(m)</td>
<td>4</td>
</tr>
<tr>
<td>Maximum Turn Length</td>
<td>inch</td>
<td>MTL(m)</td>
<td>4</td>
</tr>
<tr>
<td>Maximum Spindle Speed</td>
<td>rpm</td>
<td>MAXSP(m)</td>
<td>4</td>
</tr>
<tr>
<td>Minimum Spindle Speed</td>
<td>rpm</td>
<td>MINSP(m)</td>
<td>4</td>
</tr>
<tr>
<td>Feeds</td>
<td>ipm</td>
<td>F(m)</td>
<td>3</td>
</tr>
<tr>
<td>X-axis accuracy</td>
<td>inch</td>
<td>XACC(m)</td>
<td>7</td>
</tr>
<tr>
<td>Y-axis accuracy</td>
<td>inch</td>
<td>YACC(m)</td>
<td>7</td>
</tr>
<tr>
<td>Power</td>
<td>hp</td>
<td>P(m)</td>
<td>3</td>
</tr>
</tbody>
</table>

### Table 5.5  
**Data Structure for MCCENTER.DAT file**

<table>
<thead>
<tr>
<th>FIELD NAME</th>
<th>UNIT</th>
<th>VARIABLE NAME</th>
<th>FIELD SPACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Machine Code</td>
<td>----</td>
<td>CODE(m)</td>
<td>4</td>
</tr>
<tr>
<td>Machine Name</td>
<td>----</td>
<td>NAME(m)</td>
<td>30</td>
</tr>
<tr>
<td>Maximum X Dimension</td>
<td>inch</td>
<td>MAXX(m)</td>
<td>4</td>
</tr>
<tr>
<td>Maximum Y Dimension</td>
<td>inch</td>
<td>MAXY(m)</td>
<td>4</td>
</tr>
<tr>
<td>Maximum Z Dimension</td>
<td>inch</td>
<td>MAXZ(m)</td>
<td>4</td>
</tr>
<tr>
<td>Maximum Spindle Speed</td>
<td>rpm</td>
<td>MAXS(m)</td>
<td>4</td>
</tr>
<tr>
<td>Minimum Spindle Speed</td>
<td>rpm</td>
<td>MINS(m)</td>
<td>4</td>
</tr>
<tr>
<td>Feeds</td>
<td>ipm</td>
<td>F(m)</td>
<td>3</td>
</tr>
<tr>
<td>Accuracy</td>
<td>inch</td>
<td>ACC(m)</td>
<td>7</td>
</tr>
<tr>
<td>Power</td>
<td>hp</td>
<td>P(m)</td>
<td>3</td>
</tr>
<tr>
<td>CODE</td>
<td>MACHINE Name(m)</td>
<td>MTD(m)</td>
<td>MTL(m)</td>
</tr>
<tr>
<td>------</td>
<td>-----------------</td>
<td>--------</td>
<td>--------</td>
</tr>
<tr>
<td></td>
<td>CNC LATHE</td>
<td>inch</td>
<td>inch</td>
</tr>
<tr>
<td>M101</td>
<td>C. Milacron 208</td>
<td>8.00</td>
<td>11.75</td>
</tr>
<tr>
<td>M102</td>
<td>C. Milacron 1212</td>
<td>12.00</td>
<td>nil</td>
</tr>
<tr>
<td>M103</td>
<td>C. Milacron 1400</td>
<td>8.00</td>
<td>24.00</td>
</tr>
<tr>
<td>M104</td>
<td>G &amp; L Numeric</td>
<td>14.50</td>
<td>11.00</td>
</tr>
<tr>
<td>M105</td>
<td>J &amp; L FMS Delta</td>
<td>4.00</td>
<td>32.00</td>
</tr>
<tr>
<td>M106</td>
<td>J &amp; L TNC-A 210A</td>
<td>4.00</td>
<td>41.30</td>
</tr>
<tr>
<td>M107</td>
<td>MAZAK Q. T. 8</td>
<td>6.30</td>
<td>9.80</td>
</tr>
<tr>
<td>M108</td>
<td>MAZAK Q. T. 10</td>
<td>10.00</td>
<td>19.69</td>
</tr>
<tr>
<td>M109</td>
<td>MAZAK S. T. 25</td>
<td>11.80</td>
<td>39.50</td>
</tr>
<tr>
<td>M110</td>
<td>MAZAK S. T. 30</td>
<td>11.80</td>
<td>39.50</td>
</tr>
<tr>
<td>M111</td>
<td>MAZAK S. T. 40N</td>
<td>14.50</td>
<td>40.75</td>
</tr>
<tr>
<td>M112</td>
<td>OKUMA LB15C</td>
<td>8.27</td>
<td>19.69</td>
</tr>
<tr>
<td>M113</td>
<td>OKUMA LC20</td>
<td>7.87</td>
<td>39.37</td>
</tr>
<tr>
<td>M114</td>
<td>OKUMA LC30</td>
<td>17.72</td>
<td>39.37</td>
</tr>
<tr>
<td>M115</td>
<td>Takisawa TS-20</td>
<td>12.00</td>
<td>39.30</td>
</tr>
<tr>
<td>CODE</td>
<td>MACHINE Name(m)</td>
<td>MXX(m)</td>
<td>MXY(m)</td>
</tr>
<tr>
<td>------</td>
<td>-----------------</td>
<td>--------</td>
<td>--------</td>
</tr>
<tr>
<td></td>
<td>MACHININGCENTER</td>
<td>inch</td>
<td>inch</td>
</tr>
<tr>
<td>M201</td>
<td>ALENCO VMC330</td>
<td>22</td>
<td>13</td>
</tr>
<tr>
<td>M202</td>
<td>BRIDGEPORT btclI</td>
<td>30</td>
<td>15</td>
</tr>
<tr>
<td>M203</td>
<td>B &amp; S 1000VC</td>
<td>25</td>
<td>40</td>
</tr>
<tr>
<td>M204</td>
<td>B &amp; S 1500VC</td>
<td>60</td>
<td>30</td>
</tr>
<tr>
<td>M205</td>
<td>B &amp; S 2000VC</td>
<td>80</td>
<td>30</td>
</tr>
<tr>
<td>M206</td>
<td>C &amp; M 10VC</td>
<td>80</td>
<td>26</td>
</tr>
<tr>
<td>M207</td>
<td>C &amp; M 20VC</td>
<td>72</td>
<td>30</td>
</tr>
<tr>
<td>M208</td>
<td>C &amp; M 5VC 750</td>
<td>30</td>
<td>15</td>
</tr>
<tr>
<td>M209</td>
<td>C &amp; M 10VC 1000</td>
<td>40</td>
<td>20</td>
</tr>
</tbody>
</table>
5.1.5 Tool Data Files

A tool library is a database for tool specifications. All available tools are classified into three categories according to their geometric shapes and their basic functions. They are:

a) cylindrical surface-making tools;

b) flat surface-making tools; and

c) hole-making tools.

INSERT.DAT file consists of tool specifications of indexable inserts for turning. They are the most widely used tools for turning operations. Uncoated and coated carbide, high-speed steels, ceramics, polycrystalline diamonds, and cubic boron nitride inserts are commonly used for many applications. The data structure of INSERT.DAT is presented as Table 5.8. According to the identification system for indexable inserts in ANSI Standard B94.4 [12], a part of the INSERT.DAT file is shown in Table 5.9.
Table 5.8  Data structure of INSERT.DAT file

<table>
<thead>
<tr>
<th>FIELD NAME</th>
<th>DESCRIPTION</th>
<th>VALUE</th>
<th>FIELD SPACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>TID</td>
<td>Tool code based on attributes</td>
<td>T101,..., etc</td>
<td>4</td>
</tr>
<tr>
<td>SHAPE</td>
<td>Geometry of the tool</td>
<td>A,B,C,...,W</td>
<td>1</td>
</tr>
<tr>
<td>TOLT</td>
<td>Insert bilateral tolerance</td>
<td>± x.xxx&quot;</td>
<td>6</td>
</tr>
<tr>
<td>TYPE</td>
<td>Special feature on tool</td>
<td>A,B,C,...,S</td>
<td>1</td>
</tr>
<tr>
<td>RAD</td>
<td>Nose radius at cutting point</td>
<td>1,2,3,...,8</td>
<td>1</td>
</tr>
<tr>
<td>HAND</td>
<td>Left or right hand tool</td>
<td>L,R</td>
<td>1</td>
</tr>
<tr>
<td>MACH</td>
<td>Available machines</td>
<td>M101,...</td>
<td>4</td>
</tr>
<tr>
<td>COST</td>
<td>Tool cost per part</td>
<td>xx.xx</td>
<td>5</td>
</tr>
</tbody>
</table>

Table 5.9  An example of INSERT.DAT file

<table>
<thead>
<tr>
<th>TID</th>
<th>SHAPE</th>
<th>TOL</th>
<th>TYPE</th>
<th>RAD</th>
<th>HAND</th>
<th>COST</th>
<th>MACH</th>
</tr>
</thead>
<tbody>
<tr>
<td>T101</td>
<td>0.001</td>
<td>J</td>
<td>A</td>
<td>3</td>
<td>L</td>
<td>2.30</td>
<td>M101,M102</td>
</tr>
<tr>
<td>T102</td>
<td>0.005</td>
<td>F</td>
<td>C</td>
<td>2</td>
<td>R</td>
<td>1.50</td>
<td>M103,M105</td>
</tr>
<tr>
<td>T103</td>
<td>0.001</td>
<td>D</td>
<td>E</td>
<td>4</td>
<td>L</td>
<td>2.50</td>
<td>M103,M104</td>
</tr>
</tbody>
</table>

The tool specifications about the milling cutters are stored in MILLS.DAT. There are many different milling cutters available for various applications. Some of the most common types of milling cutters which are plain (slab), face, and end mills, are selected to be stored in this file. The data structure of MILLS.DAT is shown in Table 5.10, and the data file is presented in Table 5.11.
Table 5.10  Data structure of MILLS.DAT file

<table>
<thead>
<tr>
<th>FIELD NAME</th>
<th>DESCRIPTION</th>
<th>FIELD SPACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>TID</td>
<td>Tool ID</td>
<td>5</td>
</tr>
<tr>
<td>MACH</td>
<td>m/c available for using this tool</td>
<td>7</td>
</tr>
<tr>
<td>TDIM</td>
<td>Diameter of tool</td>
<td>7</td>
</tr>
<tr>
<td>TOL</td>
<td>Tolerance on the size of the tool</td>
<td>7</td>
</tr>
<tr>
<td>COST</td>
<td>Cost per piece</td>
<td>4</td>
</tr>
</tbody>
</table>

Table 5.11  An example of cutters in MILLS.DAT file

<table>
<thead>
<tr>
<th>TID</th>
<th>MACH</th>
<th>TDIM</th>
<th>TOL</th>
<th>COST</th>
</tr>
</thead>
<tbody>
<tr>
<td>T2P1</td>
<td>M202</td>
<td>1.0000</td>
<td>0.0150</td>
<td>1.00</td>
</tr>
<tr>
<td>T2SI</td>
<td>M201</td>
<td>0.5000</td>
<td>0.0150</td>
<td>1.05</td>
</tr>
<tr>
<td>T2ST</td>
<td>M202</td>
<td>0.8750</td>
<td>0.0150</td>
<td>1.10</td>
</tr>
</tbody>
</table>

Tool selection for hole making process involves both data retrieval method and expert rules. Two data files are developed for hole making processes according to data from handbook [15]. Examples of those data are presented in Figure 5.12.

TWIDRILL.DAT and REAMER.DAT files are constructed to represent the relationships between drill sizes and corresponding tolerances of the tools. Hence, the tolerance of a twist drill or a reamer is retrieved according to its corresponding size of the hole.
Table 5.12 Tool accuracies

<table>
<thead>
<tr>
<th>Drill Size</th>
<th>Tool Tolerance on Diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum</td>
<td>Maximum</td>
</tr>
<tr>
<td>0.0059</td>
<td>0.013</td>
</tr>
<tr>
<td>0.0135</td>
<td>0.125</td>
</tr>
<tr>
<td>0.1285</td>
<td>0.250</td>
</tr>
<tr>
<td>0.2570</td>
<td>0.500</td>
</tr>
<tr>
<td>0.5156</td>
<td>1.000</td>
</tr>
<tr>
<td>1.0156</td>
<td>2.000</td>
</tr>
<tr>
<td>2.0125</td>
<td>3.500</td>
</tr>
</tbody>
</table>

(a) General purpose twist drills

<table>
<thead>
<tr>
<th>Reamer Size</th>
<th>Tool Tolerance on Diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum</td>
<td>Maximum</td>
</tr>
<tr>
<td>0.005</td>
<td>0.250</td>
</tr>
<tr>
<td>0.265</td>
<td>1.000</td>
</tr>
<tr>
<td>1.000</td>
<td>6.000</td>
</tr>
</tbody>
</table>

(b) Common reamers.

5.1.6 Machining Data File

Machining parameters are basically the cutting speed and the feed rate, as well as the depth of cut. There are several methods, such as data retrieval and empirical formulae [16], to determine efficient machining parameters. However, data retrieval method is selected since the empirical formulae are inaccurate. Actually, there are many tables about machining parameters that are available in handbooks. In order to avoid a huge size of memory storage, some graphical method has been developed by Schey [17]. Data from that graph can be converted into several tables.
for each combination of tool and workpiece materials. The
tables are stored in PARA.DAT and are formatted as shown in
Table 5.13.

<table>
<thead>
<tr>
<th>HARDNESS OF WORKPIECE (HB)</th>
<th>CUTTING SPEED ($V_s$, fpm)</th>
<th>FEED ($F_s$, in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>110</td>
<td>150</td>
<td>0.015</td>
</tr>
<tr>
<td>200</td>
<td>90</td>
<td>0.015</td>
</tr>
<tr>
<td>350</td>
<td>60</td>
<td>0.010</td>
</tr>
<tr>
<td>.</td>
<td>.</td>
<td>.</td>
</tr>
<tr>
<td>.</td>
<td>.</td>
<td>.</td>
</tr>
</tbody>
</table>

Given the type of workpiece and tool
materials as well as the specific hardness of workpiece
materials, cutting speed and feed can be found from the
tables. When a given hardness of workpiece is within the
ranges of hardness in the table, linear interpolation for
determining appropriate cutting speed and feed will be used.

5.1.7 Data File RADIUS.DAT for Nose Radius
relative to Surface Finish and Feed Rate

The nomograph for nose radius of inserts with
surface finish and feed rate is interpreted in RADIUS.DAT
file. Given a selected feed rate and a specified surface
finish on the surface, a corresponding nose radius is given
for verification of the selected indexable inserts for
turning. Table 5.14 illustrates a portion of the
RADIUS.DAT file.
Table 5.14  RADIUS.DAT file

<table>
<thead>
<tr>
<th>NOSE RADIUS (inch)</th>
<th>SURFACE FINISH min (µ in)</th>
<th>SURFACE FINISH max (µ in)</th>
<th>FEED RATE min (inch/rev)</th>
<th>FEED RATE max (inch/rev)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01563,</td>
<td>7.0,</td>
<td>17.9,</td>
<td>0.0020,</td>
<td>0.0029</td>
</tr>
<tr>
<td>0.01563,</td>
<td>18.0,</td>
<td>32.9,</td>
<td>0.0030,</td>
<td>0.0039</td>
</tr>
<tr>
<td>0.01563,</td>
<td>33.0,</td>
<td>53.9,</td>
<td>0.0040,</td>
<td>0.0049</td>
</tr>
<tr>
<td>0.01563,</td>
<td>54.0,</td>
<td>75.9,</td>
<td>0.0050,</td>
<td>0.0059</td>
</tr>
<tr>
<td>0.01563,</td>
<td>76.0,</td>
<td>104.9,</td>
<td>0.0060,</td>
<td>0.0069</td>
</tr>
<tr>
<td>0.01563,</td>
<td>105.0,</td>
<td>139.9,</td>
<td>0.0070,</td>
<td>0.0079</td>
</tr>
<tr>
<td>0.01563,</td>
<td>140.0,</td>
<td>174.9,</td>
<td>0.0080,</td>
<td>0.0089</td>
</tr>
<tr>
<td>0.01563,</td>
<td>175.0,</td>
<td>219.9,</td>
<td>0.0090,</td>
<td>0.0099</td>
</tr>
<tr>
<td>0.01563,</td>
<td>220.0,</td>
<td>539.9,</td>
<td>0.0100,</td>
<td>0.0149</td>
</tr>
<tr>
<td>0.01563,</td>
<td>540.0,</td>
<td>880.0,</td>
<td>0.0150,</td>
<td>0.0200</td>
</tr>
<tr>
<td>0.03125,</td>
<td>0.0,</td>
<td>7.9,</td>
<td>0.0023,</td>
<td>0.0029</td>
</tr>
<tr>
<td>0.03125,</td>
<td>8.0,</td>
<td>15.9,</td>
<td>0.0030,</td>
<td>0.0039</td>
</tr>
<tr>
<td>0.03125,</td>
<td>16.0,</td>
<td>25.9,</td>
<td>0.0040,</td>
<td>0.0049</td>
</tr>
<tr>
<td>0.03125,</td>
<td>26.0,</td>
<td>35.9,</td>
<td>0.0050,</td>
<td>0.0059</td>
</tr>
<tr>
<td>0.03125,</td>
<td>36.0,</td>
<td>48.9,</td>
<td>0.0060,</td>
<td>0.0069</td>
</tr>
<tr>
<td>0.03125,</td>
<td>49.0,</td>
<td>63.9,</td>
<td>0.0070,</td>
<td>0.0079</td>
</tr>
<tr>
<td>0.03125,</td>
<td>64.0,</td>
<td>79.9,</td>
<td>0.0080,</td>
<td>0.0089</td>
</tr>
<tr>
<td>0.03125,</td>
<td>80.0,</td>
<td>101.9,</td>
<td>0.0090,</td>
<td>0.0099</td>
</tr>
<tr>
<td>0.03125,</td>
<td>102.0,</td>
<td>259.9,</td>
<td>0.0100,</td>
<td>0.0149</td>
</tr>
<tr>
<td>0.03125,</td>
<td>260.0,</td>
<td>400.9,</td>
<td>0.0150,</td>
<td>0.0199</td>
</tr>
<tr>
<td>0.03125,</td>
<td>401.0,</td>
<td>569.9,</td>
<td>0.0200,</td>
<td>0.0249</td>
</tr>
<tr>
<td>0.03125,</td>
<td>570.0,</td>
<td>870.0,</td>
<td>0.0250,</td>
<td>0.0277</td>
</tr>
<tr>
<td>0.04688,</td>
<td>0.0,</td>
<td>2.9,</td>
<td>0.0028,</td>
<td>0.0029</td>
</tr>
<tr>
<td>0.04688,</td>
<td>3.0,</td>
<td>11.9,</td>
<td>0.0030,</td>
<td>0.0039</td>
</tr>
<tr>
<td>0.04688,</td>
<td>12.0,</td>
<td>17.9,</td>
<td>0.0040,</td>
<td>0.0049</td>
</tr>
<tr>
<td>0.04688,</td>
<td>18.0,</td>
<td>24.9,</td>
<td>0.0050,</td>
<td>0.0059</td>
</tr>
<tr>
<td>0.04688,</td>
<td>25.0,</td>
<td>32.9,</td>
<td>0.0060,</td>
<td>0.0069</td>
</tr>
</tbody>
</table>

5.1.8  Data File for Jigs and Fixtures

Holding devices, such as jigs and fixtures, in machining are important for maintaining the rigidity between the machine and the workpiece. Some jigs can hold a rotational part firmly without producing any significant effect on the roundness of the workpiece. However, some other jigs and fixtures may deform the geometric shape of the part after the machining processes due to the design of
the holding device. A holding device can be classified based on process type, available machines and its accuracy. An example of HOLDING.DAT file is presented in Figure 5.15.

<table>
<thead>
<tr>
<th>CODE</th>
<th>PROCESS</th>
<th>MACHINE AVAILABLE</th>
<th>ACCURACY</th>
<th>COST</th>
</tr>
</thead>
<tbody>
<tr>
<td>F001</td>
<td>TURNING</td>
<td>M101,M103,M110</td>
<td>0.0010</td>
<td>1.00</td>
</tr>
<tr>
<td>F002</td>
<td>MILLING</td>
<td>M202,M203,M207</td>
<td>0.0006</td>
<td>1.30</td>
</tr>
<tr>
<td>F003</td>
<td>TURNING</td>
<td>M102,M103,M104</td>
<td>0.0020</td>
<td>0.65</td>
</tr>
</tbody>
</table>

5.1.9 Database for the Statistical Data from SPC

The statistical data are the sample mean \( x \), and the variance \( S^2 \), or standard deviation \( S \). The database called SPC.DAT is created by means of the SPC package. The SPC monitors the on-line production outputs. The finished parts are being measured at certain time intervals. After a number of samples are collected, the SPC calculates the statistical data, for instance, sample mean, sample variance, natural tolerance limits and percentage of defectives, for that specific setting of production. A portion of SPC.DAT file is listed in Table 5.16.
<table>
<thead>
<tr>
<th>PROCESS NAME</th>
<th>M/C CODE</th>
<th>TOOL CODE</th>
<th>SPEED (rpm)</th>
<th>FEED (ipr)</th>
<th>DEPTH (in)</th>
<th>RATE OF CUT (in)</th>
<th>SIZE</th>
<th>UPPER LIMIT (in)</th>
<th>LOWER LIMIT (in)</th>
<th>% DEFECTIVES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drilling</td>
<td>M102</td>
<td>T3T2</td>
<td>2000</td>
<td>0.005</td>
<td>0.010</td>
<td>1.000</td>
<td>1.002</td>
<td>0.998</td>
<td>1.000</td>
<td>0.787</td>
</tr>
<tr>
<td>Reaming</td>
<td>M104</td>
<td>T3R1</td>
<td>1500</td>
<td>0.003</td>
<td>0.015</td>
<td>0.636</td>
<td>0.641</td>
<td>0.533</td>
<td>0.533</td>
<td>0.890</td>
</tr>
</tbody>
</table>

5.2. Qualifiers and Variables

All the data of the information in the expert system are represented by qualifiers and variables. Qualifiers represent the information qualitatively while variables represent the numerical values in any mathematical expression for external programs and expert rules. Choices, which are generated by the rules with qualifiers and variables, are the output of the expert system. The qualifiers and the choices are listed on Table 5.17. The variables are substituted by the corresponding value in the matched knowledge base. The variables and their units are listed in Table 5.18.
<table>
<thead>
<tr>
<th>QUALIFIER</th>
<th>CHOICES</th>
<th>CODE</th>
</tr>
</thead>
<tbody>
<tr>
<td>SHAPE IS</td>
<td>1) ROTATIONAL</td>
<td>(R)</td>
</tr>
<tr>
<td></td>
<td>2) PRISMATIC</td>
<td>(P)</td>
</tr>
<tr>
<td>COMPLEXITY IS</td>
<td>1) UNIFORM</td>
<td>(U)</td>
</tr>
<tr>
<td></td>
<td>2) CLOSED ONE END</td>
<td>(CL)</td>
</tr>
<tr>
<td></td>
<td>3) CHANGE AT THE END</td>
<td>(CE)</td>
</tr>
<tr>
<td></td>
<td>4) CHANGE AT CENTER</td>
<td>(CC)</td>
</tr>
<tr>
<td></td>
<td>5) THREAD</td>
<td>(T)</td>
</tr>
<tr>
<td>SURFACE IS</td>
<td>1) EXTERNAL</td>
<td>(E)</td>
</tr>
<tr>
<td></td>
<td>2) INTERNAL</td>
<td>(I)</td>
</tr>
<tr>
<td></td>
<td>3) CROSS-SECTION</td>
<td>(C)</td>
</tr>
</tbody>
</table>

a) Qualifiers for identifying features

<table>
<thead>
<tr>
<th>QUALIFIER</th>
<th>CHOICES</th>
</tr>
</thead>
<tbody>
<tr>
<td>FEATURE IS</td>
<td>1) HOLE</td>
</tr>
<tr>
<td></td>
<td>2) BLIND HOLE</td>
</tr>
<tr>
<td></td>
<td>3) TAPER</td>
</tr>
<tr>
<td></td>
<td>4) THREAD</td>
</tr>
<tr>
<td></td>
<td>5) STRAIGHT CYLINDRICAL SURFACE</td>
</tr>
<tr>
<td></td>
<td>6) FACE</td>
</tr>
<tr>
<td></td>
<td>7) COUNTERSINK</td>
</tr>
<tr>
<td></td>
<td>8) CHAMFER</td>
</tr>
<tr>
<td></td>
<td>9) PLANE</td>
</tr>
</tbody>
</table>

b) Qualifiers of FEATURE
Table 5.17  
Continued

<table>
<thead>
<tr>
<th>QUALIFIER</th>
<th>CHOICES</th>
</tr>
</thead>
<tbody>
<tr>
<td>CPROCESS IS</td>
<td>1) F_BORING</td>
</tr>
<tr>
<td>or  PROC(FEATURE)</td>
<td>2) R_BORING</td>
</tr>
<tr>
<td>or  PROC(GEO.TOL)</td>
<td>3) BROACHING</td>
</tr>
<tr>
<td></td>
<td>4) CHAMFERING</td>
</tr>
<tr>
<td></td>
<td>5) DRILLING</td>
</tr>
<tr>
<td></td>
<td>6) F_GRINDING</td>
</tr>
<tr>
<td></td>
<td>7) R_GRINDING</td>
</tr>
<tr>
<td></td>
<td>8) H_GRINDING</td>
</tr>
<tr>
<td></td>
<td>9) FACING</td>
</tr>
<tr>
<td></td>
<td>10) FLAME_CUTTING</td>
</tr>
<tr>
<td></td>
<td>11) HONIFG</td>
</tr>
<tr>
<td></td>
<td>12) MILLING</td>
</tr>
<tr>
<td></td>
<td>13) REAMING</td>
</tr>
<tr>
<td></td>
<td>14) TAPER_T</td>
</tr>
<tr>
<td></td>
<td>15) TAPER_B</td>
</tr>
<tr>
<td></td>
<td>16) THREADING</td>
</tr>
<tr>
<td></td>
<td>17) F_TURNING</td>
</tr>
<tr>
<td></td>
<td>18) R_TURNING</td>
</tr>
</tbody>
</table>

c) Qualifiers for process selection

<table>
<thead>
<tr>
<th>QUALIFIER</th>
<th>CHOICES</th>
</tr>
</thead>
<tbody>
<tr>
<td>HARDNESS IS</td>
<td>1) SOFT</td>
</tr>
<tr>
<td></td>
<td>2) MEDIUM</td>
</tr>
<tr>
<td></td>
<td>3) HARD</td>
</tr>
<tr>
<td>PRODUCTION RATE IS</td>
<td>1) LOW</td>
</tr>
<tr>
<td></td>
<td>2) MEDIUM</td>
</tr>
<tr>
<td></td>
<td>3) HIGH</td>
</tr>
<tr>
<td>PRODUCTION VOLUME IS</td>
<td>1) LOW</td>
</tr>
<tr>
<td></td>
<td>2) MEDIUM</td>
</tr>
<tr>
<td></td>
<td>3) HIGH</td>
</tr>
<tr>
<td>TOOL TYPE IS</td>
<td>1) CYLINDRICAL</td>
</tr>
<tr>
<td></td>
<td>2) FLAT</td>
</tr>
<tr>
<td></td>
<td>3) HOLE</td>
</tr>
<tr>
<td></td>
<td>4) OTHER</td>
</tr>
<tr>
<td>TCODE IS</td>
<td>1) TWIST</td>
</tr>
<tr>
<td></td>
<td>2) REAM</td>
</tr>
</tbody>
</table>

d) Qualifiers for tool selection
### CHOICES FOR PROCESSES

<table>
<thead>
<tr>
<th>PROCESS IS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1) F_BORING</td>
</tr>
<tr>
<td>2) R_BORING</td>
</tr>
<tr>
<td>3) BROACHING</td>
</tr>
<tr>
<td>4) CHAMFERING</td>
</tr>
<tr>
<td>5) DRILLING</td>
</tr>
<tr>
<td>6) F_GRINDING</td>
</tr>
<tr>
<td>7) R_GRINDING</td>
</tr>
<tr>
<td>8) H_GRINDING</td>
</tr>
<tr>
<td>9) FACING</td>
</tr>
<tr>
<td>10) FLAME_CUTTING</td>
</tr>
<tr>
<td>11) HONING</td>
</tr>
<tr>
<td>12) MILLING</td>
</tr>
<tr>
<td>13) REAMING</td>
</tr>
<tr>
<td>14) TAPER_T</td>
</tr>
<tr>
<td>15) TAPER_B</td>
</tr>
<tr>
<td>16) THREADING</td>
</tr>
<tr>
<td>17) F_TURNING</td>
</tr>
<tr>
<td>18) R_TURNING</td>
</tr>
</tbody>
</table>

### CHOICES FOR TOOLS

<table>
<thead>
<tr>
<th>TOOL IS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1) GRINDING WHEEL</td>
</tr>
<tr>
<td>2) BROACHER</td>
</tr>
<tr>
<td>3) MILLS</td>
</tr>
<tr>
<td>4) STRAIGHT-SHANK TWIST DRILL</td>
</tr>
<tr>
<td>5) TAPER-SHANK TWIST DRILL</td>
</tr>
<tr>
<td>6) HIGH-SHANK TWIST DRILL</td>
</tr>
<tr>
<td>7) LOW-HELIX TWIST DRILL</td>
</tr>
<tr>
<td>8) CORE DRILL</td>
</tr>
<tr>
<td>9) HAND REAMER</td>
</tr>
<tr>
<td>10) STUB-SCREW REAMER</td>
</tr>
<tr>
<td>11) HSS TOOL</td>
</tr>
<tr>
<td>12) BRAZED CARBIDE TOOL</td>
</tr>
<tr>
<td>13) INDEXABLE TOOL</td>
</tr>
<tr>
<td>14) TAPS</td>
</tr>
<tr>
<td>15) OPENING DIE HEADS</td>
</tr>
</tbody>
</table>

e) Choices of the expert system
Table 5.18  List of variables in the expert system

<table>
<thead>
<tr>
<th>VARIABLE DESCRIPTION</th>
<th>UNIT</th>
<th>VARIABLE NAME</th>
<th>FIELD SIZE</th>
<th>RANGE OF VARIABLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter</td>
<td>inch</td>
<td>DIMEN</td>
<td>7</td>
<td>0.0001 - 30.0000</td>
</tr>
<tr>
<td>Length of Surface</td>
<td>inch</td>
<td>LEN</td>
<td>7</td>
<td>0.0100 - 25.0000</td>
</tr>
<tr>
<td>Tolerance</td>
<td>inch</td>
<td>TOL</td>
<td>7</td>
<td>0.0001 - 0.5000</td>
</tr>
<tr>
<td>Concentricity</td>
<td>inch</td>
<td>CONC</td>
<td>7</td>
<td>0.00005 - 0.5000</td>
</tr>
<tr>
<td>Roundness</td>
<td>inch</td>
<td>ROUND</td>
<td>7</td>
<td>0.00005 - 0.5000</td>
</tr>
<tr>
<td>Straightness</td>
<td>inch</td>
<td>ST</td>
<td>7</td>
<td>0.00005 - 0.5000</td>
</tr>
<tr>
<td>Cylindricity</td>
<td>inch</td>
<td>CY</td>
<td>7</td>
<td>0.00005 - 0.5000</td>
</tr>
<tr>
<td>Surface Roughness</td>
<td>μ in</td>
<td>SR</td>
<td>7</td>
<td>1 - 2000</td>
</tr>
<tr>
<td>Cutting Speed</td>
<td>fpm</td>
<td>SPEED</td>
<td>6</td>
<td>1.0000 - 1000.0</td>
</tr>
<tr>
<td>Feed</td>
<td>inch</td>
<td>FEED</td>
<td>6</td>
<td>10.00 - 200.00</td>
</tr>
<tr>
<td>Depth of Cut</td>
<td>inch</td>
<td>DOC</td>
<td>6</td>
<td>0.0001 - 0.5000</td>
</tr>
<tr>
<td>Operation Index</td>
<td></td>
<td>INDEX</td>
<td>4</td>
<td>0.0000 - 1.0000</td>
</tr>
<tr>
<td>Machine ID</td>
<td></td>
<td>MACHINE</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>Tool ID</td>
<td></td>
<td>TOOL</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>Type of Cutting Tool</td>
<td></td>
<td>TOOL TYPE</td>
<td>12</td>
<td></td>
</tr>
<tr>
<td>Drilling Tool Type</td>
<td></td>
<td>TCODE</td>
<td>5</td>
<td></td>
</tr>
<tr>
<td>Holding Device ID</td>
<td></td>
<td>HOLDING</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>Total Production Cost</td>
<td></td>
<td>TCOST</td>
<td>6</td>
<td></td>
</tr>
</tbody>
</table>

5.3  Algorithms and External Programs

5.3.1  Algorithms to Extract Data and Verify Dimensional and Geometric Tolerances

TOLEXV (Tolerance Extraction and Verification) module is developed for the extraction of the dimensional tolerance data from CAD database and for the verification of the dimensional and the geometrical tolerances in a part drawing. The extraction algorithm retrieves and tabulates the data from a CAD database. The verification algorithm checks for the appropriate values of tolerances which have been put on the drawing and then generates tolerance limits for the user, in order to avoid arbitrary tolerancing.

49
Tolerance verification is implemented by means of two computer programs. The first program, TOLEXV.BAS, has been developed to extract dimensions and dimensional tolerances on the part drawing and to verify the dimensional tolerances. Then, the program continues to assist the user in assigning geometric tolerances. TOLEXV.BAS generates the verified values of tolerances. The listing of TOLEXV.BAS is shown in Appendix A. The second program, CAGT.LSP, continues to draw the verified dimensional and geometric tolerances for the part drawing. First, the part geometry is generated in the AutoCAD, then the user loads the CAGT.LSP program which generates new dimensional tolerances that have been modified. Afterwards, CAGT.LSP generates the geometric tolerances on the drawing, and saves the drawing in a new file. The listing of CAGT.LSP is shown in Appendix B. The system diagram for the TOLEXV is shown in Figure 5.2. TOLEXV module consists of the following three sub-modules.

5.3.1.1 Data Retrieval Module: This module has been developed to extract dimensions and tolerances from the DXF file by a sequential method. A search loop records all data about dimensions until all entities have been extracted. Then, the user will be asked to input the surface roughness of each entity, if the surface quality is required. Then, each surface with the above information is stored in DIMTOL.DAT file.
Figure 5.2  System diagram for tolerance extraction and verification (TOLEXV) module
5.3.1.2 Dimensional Tolerance Verification Module: A tolerance stack up occurs when accepted tolerances on individual dimensions combine in such a way as to create an unacceptable variation in an overall dimension. When extreme permissible tolerances combine, the condition is called a limit stack. Given an example of a cube which is machined to 1.000±0.005 inch, if two cubes are stacked, it is desired that their combined height does not exceed 2.000±0.005 inch. It is clear that the total tolerances ± 0.005 inch cannot be achieved all the time. The module verifies three cases in tolerancing. The first two cases are related to the principle of tolerance stackups. The last case is related to the assembly with other parts.

Case 1 - Tolerance Stackups of Some Surfaces of Rotational Part with Common-Point Dimensioning Method: Sometimes when part designers use the Common Point Dimensioning method, they do not assign dimension to one of the surfaces. However, the implication of the tolerance is hard to be seen directly. Some calculations may be involved to obtain the value of tolerance for the surface. If the surface of that segment is functionally critical, then the tolerance value will be assigned on the drawing. Even though the user does not specify tolerances on non-critical surfaces, the machine operator still needs to know the limit of the size of those surfaces. The relation between total dimensional tolerance and surfaces is illustrated by the example shown in Figure 5.3.
The dimensional tolerances of the surface C is calculated as follows:

plus tolerance of surface C : \( c = a + b \)
minus tolerance of surface C : \( -c = -(a + b) \)

The algorithm checks with the x coordinates of the extension lines in the dimension of each surface. Referring to Figure 5.3, if the x coordinates of surface A and surface B are the same, then it implies that these two surfaces are measured with common point dimensioning method. The program will extract the dimensions and the tolerances of those surfaces to calculate the dimension and tolerance values of surface C. The coordinates of the extension line of surface C are also defined by one of the extension line of surfaces A and B. The information of that new entity will be appended into the DIMTOL.DAT file.
Case 2 - Tolerance Stackups of the Total Length of the Rotational Part: According to the chain dimensioning method, the overall tolerance of the total length of the rotational part is the sum of all tolerances in all surfaces.

\[ \text{OVERALL TOL.} = T(1) + T(2) + \ldots + T(n) \]

where the part has \( n \) surfaces.

Since bilateral tolerancing method is applied, overall plus tolerance and overall minus tolerances are equal but in opposite sign. The program first calculates the overall tolerance which is going to be compared with the assigned tolerance of the total dimension of the part. If the overall tolerance is greater than the assigned tolerance, then the tolerances of some surfaces have to be tightened up in order to fulfill the design requirement of the part. If the overall tolerance is less than the assigned tolerance, then the tolerances of the individual surfaces have to be reduced until the overall tolerance does not exceed the assigned tolerance. The flow charts of the algorithms for Case 1 and 2 is presented in Figure 5.4.
Figure 5.4 Flow Chart of dimensional tolerances verification in Case 1 and 2
Case 3 - Verification for Dimensional Tolerance of Diameter with the Input Dimension and its Tolerance of the Matching Part: All dimensional tolerances for the diameters of a rotational part are verified in order to avoid mismatching of this part with its matching one. The program scans through the DIMTOL.DAT file for searching the rotational surfaces. The designer has to answer a query whether the fit is clearance or interference between a surface, such as a shaft, and another surface from its matching part, such as a hole. Before the calculation for verification starts, the dimensions of both parts D(s) & D(h), tolerances of both parts T(s) & T(h), and the minimum clearance between the two parts have to be retrieved. The flow chart of this verification is illustrated on Figure 5.5, and the logic of the program is described as follows. First, if the clearance is specified, then minimum (MINCL) clearance will be specified by the user. If maximum size of the shaft with the minimum clearance has already exceeded the smallest size of the hole, then the tolerance on the shaft diameter should be reduced. If interference is required, then the maximum size of the shaft should not be smaller than the minimum size of the hole; otherwise, increase in shaft tolerance is allowed. The program gives either the upper or the lower limits of the tolerances for the user to adjust into a logical value of tolerance.
Figure 5.5 Flow chart of dimensional tolerance verification in Case 3
5.3.1.3 Computer Assisted Geometric Tolerancing Module: It assumes that no geometric tolerance has been assigned on the part drawing. The system will assist the user to assign geometric tolerances on the drawing according to the standard rules from CSA standard. The user will be advised of the maximum and minimum values of the geometric tolerances. According to the design requirement, values of the geometric tolerances can be adjusted to smaller values. Therefore, the system suggests but does not assign the geometric tolerances for the user. The flow chart of the geometric tolerance verification is shown in Figure 5.6. The program will first identify a feature with two surfaces. For example, the diameter of a circular surface and the length of a surface normal to the circular surface will form a cylinder as a shaft. Then the user has to decide whether this feature is going to assemble with another part. If assembly is required, the user needs to input the minimum diameter of the matching part for the feature. If assembly is not required, then the program assumes that there is no need to assign geometric tolerance for that feature. Similarly, the other surface is analyzed. Then, the user will be asked a specific query for each type of geometric tolerances in each circular surface.

In the proposed expert system, the geometric tolerances are applied to all cross section surfaces. There are four types of geometric tolerances being considered in this module:
a) Straightness and Circularity

Figure 5.6 Flow chart of Computer-Assisted Geometric Tolerancing module (CAGT)
b) Cylindricity and Concentricity

Figure 5.6  Continued
STRAIGHTNESS is a condition of a generator of a surface or of a centre line. The system calculates the aspect ratio (L/D) which indicates the relation between the length and the diameter of a rotational surface. If the value is greater than three, it implies that more attention has to be paid to the straightness of the feature. So, the straightness is always applied to a long shaft. According to the CSA B78.2-86 standard [19], maximum straightness can be calculated by the following equation:

\[
\text{MAXIMUM STRAIGHTNESS} = \text{TOLERANCE RANGE} - \left( \frac{\text{MINIMUM HOLE SIZE} - \text{DIAMETER}}{\text{D(S)+T(S)}} \right) \quad (1)
\]

Straightness tolerance also depends on the length of the rotational part. Some minimum values of straightness are given by Reshetor [20]. If the surface is longer than 1.67 inches or its diameter is bigger than 6.3 inches, then the minimum required straightness tolerance has to be 0.00055 inch. If the rotational part is longer than 23.6 inches, then minimum straightness will be 0.0014 in.

CIRCULARITY (ROUNDNESS) refers to a condition of the surface of a circular part or feature wherein the periphery of a plane cross section is equidistant from the common centre [19]. If the cylindrical surface is used to fit into the hole or collar instead of having clearance between the two surfaces, then roundness is applied to limit the cross section surface into a circular shape. The query: "Should
the shaft fit in a hole" will be asked. The maximum circularity tolerance is usually just one half of the tolerance range or is the tolerance value stated as \( T(s) \).

**Cylindricity** is a condition of the surface of revolution of a right circular in which all points on the surface are equidistant from a common axis. A cylindricity tolerance specifies a tolerance zone bounded by two concentric cylinders within which the surface lies [19]. If circularity tolerance is considered for a single surface, the program does not evaluate the cylindricity tolerance, vice versa. If a shaft has to be rotated inside a motor with certain clearance like a cylindrical boundary away from the edge of hole, then the diameter of the shaft should not be greater than the minimum diameter of the hole or collar. The maximum cylindricity tolerance is calculated by the following equation:

\[
\text{MAXIMUM CYLINDRICITY} = \frac{1}{2} \left( \frac{\text{MINIMUM HOLESIZE} - \text{DIAMETER}}{\text{MAX. SHAFT}} \right) - \left( \frac{\text{D(h)} - \text{T(h)}}{\text{D(s)} + \text{T(s)}} \right)
\]

Based on the general average requirements of accuracy for workpiece on machine tool [20], if the shaft diameter is greater than 1.67 inches or the length of the shaft is greater than 8 times of the diameter, then the minimum cylindricity tolerance has to be 0.0006 in.
CONCENTRICITY controls the permissible eccentricity between two or more surfaces of revolution that are intended to be coaxial. If the shaft is rotating, it has to be concentric with the center line of all the components. The maximum and minimum concentricity tolerance limits are derived from the following equations:

\[
\text{MAXIMUM CONCENTRICITY} = \begin{bmatrix}
\text{MAXIMUM} & \text{MIN. DIAMETER} \\
\text{HOLE SIZE - ON THE SHAFT} & D(h)+T(h) \\
\text{Diameter Tolerance (MAXCO)} & D(s)-T(s)
\end{bmatrix}
\]

\[
\text{MAXIMUM CONCENTRICITY} = \begin{bmatrix}
\text{MINIMUM} & \text{MAX. DIAMETER} \\
\text{HOLE SIZE - ON THE SHAFT} & D(h)-T(h) \\
\text{Diameter Tolerance (MAXCO)} & D(s)+T(s)
\end{bmatrix}
\]

If the concentricity is required, then the program calculates the maximum and the minimum geometric tolerance values.

So, the user can assign the geometric tolerance value within that tolerance range. After all surfaces have been examined by this module, a list of the surfaces which require geometric tolerances is stored in DIMTOL.DAT. The geometric tolerance values, such as CONC, ROUND, ST, and CY, are also added to the DIMTOL.DAT file.

A second part of the module is the CAGT.LSP program which is the post-processor of the TOLEXV.BAS program. CAGT.LSP can run the commands of the CAD system. The program first retrieves dimensions and their tolerances in the DIMTOL.DAT file which is generated from the TOLEXV.BAS program. The program draws the changed
dimensional tolerances. Second, the program creates all the BLOCKs for drawing the feature control symbols of the geometric tolerances. Then, verified geometric tolerances are printed on the features according to the standard format. After a complete drawing is shown on the screen, the new drawing files can be saved.

5.3.2 External Program for Verification of Surface Roughness based on Dimensional Tolerances (SURF-TOL.BAS)

A surface-finish notation defines the minimum surface quality of a workpiece. It is designated by the maximum allowable surface roughness which is usually expressed as an arithmetic average in micro-inches. The surface finish actually is another input other than tolerances from the user. For each type of the specified surface finishes, relative values of dimensional tolerance and surface roughness $R_a$ are implied as shown in Table 5.19. Therefore, Doyle et al. [13] and Machinery's Handbook [15] mention that the surface roughness $R_a$ can be expressed in the following equation:

$$\text{Tolerance Range} \geq 8 R_a$$

(5)
Table 5.19  Relative values of surface finish with usual tolerances

<table>
<thead>
<tr>
<th>COMMON NAME FOR FINISH</th>
<th>SURFACE FINISH $R_a$ (μ in)</th>
<th>USUAL TOLERANCE (inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mirror</td>
<td>4</td>
<td>0.0002</td>
</tr>
<tr>
<td>Polished</td>
<td>8</td>
<td>0.0005</td>
</tr>
<tr>
<td>Ground</td>
<td>16</td>
<td>0.0010</td>
</tr>
<tr>
<td>Smooth</td>
<td>32</td>
<td>0.0020</td>
</tr>
<tr>
<td>Fine</td>
<td>63</td>
<td>0.0030</td>
</tr>
<tr>
<td>Semifine</td>
<td>125</td>
<td>0.0040</td>
</tr>
<tr>
<td>Medium</td>
<td>250</td>
<td>0.0070</td>
</tr>
<tr>
<td>Semirough</td>
<td>500</td>
<td>0.0130</td>
</tr>
<tr>
<td>Rough</td>
<td>1000</td>
<td>0.0250</td>
</tr>
</tbody>
</table>

The proposed expert system first verifies the surface finish of each surface before surface roughness is applied for process selection. If the required surface quality is low, i.e. the surface roughness is high, then the tolerance implied from surface finish may be much bigger than the assigned dimensional tolerance. However, according to the design requirement for function and/or assembly, design tolerance may not be easily changed. The way to adjust the deviation between tolerance and surface finish is to improve the surface quality of the part. In order to improve surface quality, decrease in the surface roughness $R_a$ to a value of 1/8 of the tolerance range will compensate for the deviation. On the other hand, if surface quality is high, the small value of surface roughness will not be able to affect the design tolerance in any sense of assembly or functionality.
An external program called SURF-TOL.BAS retrieves surface finish and the tolerance from DIMTOL.DAT file. Since the dimensional tolerance is presented as bilateral tolerance, the tolerance range is twice the tolerance value given in the tolerance information file (DIMTOL.DAT). The program simply assigns the surface roughness into 1/8 of the tolerance range when the original surface roughness is greater than 1/8 of the tolerance range.

5.3.3 External Program PROCESS.BAS for Process Selection Based on Dimensional Tolerance and Surface Finish

There is an overlap among different processes within certain ranges and surface roughness. According to the information of each surface from DIMTOL.DAT file, its dimensional tolerance and surface roughness values will match with several processes. Probability for each process is assigned for comparing the preferences between different processes.

The external program PROCESS.BAS has been developed to handle this matching task for selecting machining processes. The program retrieves the PROCESS.DAT files. This data file consists of a number of machining processes with both upper and lower limits of dimensional tolerances and surface roughness. Any process that is able to produce a part with the input tolerance and surface finish values will be assigned with a probability of at least 5 out of 10. If the input value stays within the
upper and lower limits, the total accession of each possible process will be calculated. Given two different machining processes, the process with smaller accession implies that the process is relatively suitable to achieve the input conditions and is relatively more economical. Therefore, the smaller the accession for a process, the higher the probability for selecting that process.

All the feasible processes are rated from 5 to 9 out of 10. The output of this algorithm will be sent back to the expert system. For example, if a surface has a dimensional tolerance of 0.004 inch, the algorithm generates several processes such as finish bore, finish turning, milling and drilling. Regardless of the feature of the entity, drilling operation is ranked into the highest probability while the low possible operation is finish bore. Comparing with the frame of machining operation knowledge in Turbo-CAPP, once any entity has the dimensional tolerance which is greater than 0.0003 inch and other conditions are satisfied, then those processes will be selected.

5.3.4 External Program for Generating COMBINED PROCESS (JOIN-T&S.BAS)

According to the outputs of PROCESS.BAS, different processes may be recommended based on two different criteria. First, the output from dimensional tolerance is combined with that from surface roughness. Both outputs are formed into two groups of data and are sent into the JOIN-T&S.BAS program. A process is selected
based on both tolerance and surface finish, because both technological characteristics of dimensional tolerance and surface roughness are equally important. The probability of the combined process is by averaging the two probabilities from both characteristics. The combined processes with probabilities are identified as COM-PROCESS.

Second, processes from geometric tolerance rules are also combined with COM-PROCESS. The recommended process(es) based on geometric tolerance is/are appended into the list of COM-PROCESS. If a process recommended from geometric tolerance is also selected in COM-PROCESS, then the probability of preference of that process will be calculated by the equation:

\[ Pr.(\text{COM-PRO.}) = Pr.(\text{COM-PRO.}) + 0.5 \times Pr.(\text{Geo.Tol.}) \] (6)

Finally, the JOIN-T&S.BAS program sends the list of the CPROCESS back to the expert system by PROCESS.OUT file. A flow chart for the logic in PROCESS.BAS and JOIN-T&S.BAS programs is presented on Figure 5.7.
Figure 5.7 Flow chart for logic in PROCESS.BAS and JOIN-T&S programs
5.3.5 External Program for Parting or Finishing
the End of the Part based on Raw Material
Size (FINISH.BAS)

This program is used when all surfaces have
been identified as features. The program first searches
for the two ends of the rotational part, then it decides
whether the two ends require facing operation or cutoff
operation. If two features are faces and the distance
between them is the longest, then the program will retrieve
the following data for calculation.

<table>
<thead>
<tr>
<th>Description</th>
<th>Variable</th>
</tr>
</thead>
<tbody>
<tr>
<td>surface roughness of both faces</td>
<td>SR(1), SR(2)</td>
</tr>
<tr>
<td>total length of part</td>
<td>PLEN</td>
</tr>
<tr>
<td>tolerance of total length</td>
<td>PTOL</td>
</tr>
<tr>
<td>total length of raw material</td>
<td>RLEN</td>
</tr>
<tr>
<td>tolerance of the length of raw material</td>
<td>RTOL</td>
</tr>
</tbody>
</table>

Usually raw materials are cut off within
certain allowances. The sizes of the raw materials are
usually at least one-eighth inch bigger than the required
size of a part. If the allowance is too big, there is no
need for the facing operation. So, an algorithm for the
calculation of the amount of stock suggests whether a cut
off process is required for the end of the part.

The flow chart of the program is shown in
Figure 5.8. First of all, the size of the raw material is
verified based on the design requirements. The design
requirements are the dimensional tolerance of the part
length and the surface finishes on both end of the part.
Figure 5.8    Flow chart for FINISH.BAS program
The minimum size of the raw material (RMS) is simply calculated by the equation:

$$RMS = DIMENSION\ (RM) - MINUS\ TOLERANCE\ (RM)$$  \hspace{1cm} (7)

The maximum possible size (PS) of the finished part is defined by the equation:

$$PS = DIMENSION\ (PART) + PLUS\ TOLERANCE\ (PART)$$  \hspace{1cm} (8)

Based on machining experience, a rough cut in facing usually takes out 0.016 inch while a finish cut will only take out 0.004 inch from the part length. Assuming only one single finish cut can produce a finished surface at one end of the part, the allowance length for finishing the part will depend on the surface requirement on both ends of the part. The allowable length for facing or cutoff is calculated by the equation:

$$ALLOWANCE = [RMS] - [PS] - N \times [FINISH] \geq 0$$  \hspace{1cm} (9)

where N is equal to 1 or 2, which represents the number of end(s) of the part that requires facing operation. The variable FINISH is 0.004 in. If the allowance is less than zero, then a warning for requiring a larger size of the raw materials is displayed. If the user overlooks the warning message, a large quantity of scrap may result.

The second part of the program will continue if ALLOWANCE is a positive value. The program will decide whether to cut off the part or to finish the ends of the part by facing. It is practically assumed that a rough cut will take off 0.016 inch. So, if the ALLOWANCE is 4 times larger than a rough cut, then a cut off process is required. Moreover, if the surface roughness $R_a$ is less
than 63 \( \mu \) inches, then a finish cut is required. Finally, the selected process is appended into the PROCESS.OUT and machines are going to be selected. All the listings of the above external programs for process selection are presented in Appendix C.

5.3.6 Machine Selection Algorithms

Two external programs LATHE.BAS and MCCENTER.BAS are developed for selecting machines from machine data file LATHE.DAT and MCCENTER.DAT, respectively. The logic of the programs is similar to that of the PROCESS.DAT file. Both programs retrieve the machine specifications. Each machine in the database will be examined by matching the values from surface information with the machine specifications. Several conditions in the following have to be satisfied:

1. Length of raw material \( \leq \) Maximum turn length, MTL(m)
2. Diameter of the surface \( \leq \) Maximum turn diameter, MTD(m)
3. Dimensional tolerance of the surface \( \geq \) Machine Accuracy

If a machine can satisfy the above conditions, then this machine will be selected and a probability of preference will be assigned based on the difference between the dimensional tolerance of surface and the machine accuracy. The smaller the difference, the higher the probability will be assigned to that machine. The probability will be greater than 5 out of 10 and no
greater than 9 out of 10. The program finally generates an output file OPERATE.DAT which consists of the information of selected processes and machines for each machining surface. The flow charts for both external programs are shown on Figure 5.9 and the listings of the programs are located in Appendix D.

a) LATHE.BAS program for selection of lathes

Figure 5.9 Flow chart for machine selection algorithms
b) MCCENTER.BAS program for selection of machining centers

Figure 5.9  Continued
5.3.7 Tool Selection Algorithm

Tool selection module is required to extract the tool library for matching available tools with the selected machines and tool accuracy (tolerance). TOOL.BAS is a program to select a list of turning and milling tools from INSERT.DAT and MILLS.DAT, respectively. The tool tolerances have to be smaller than the allowable variation of the size (tolerance) of the surface. Not only the tool tolerance is considered, the compatibility of available machines are also considered. Probability of each selected tool will be calculated for the user to consider the best choice of tools. A tool, with an accuracy which is just slightly smaller than the required tolerance may be ranked as the first choice, because it is unnecessary to pay for tools that cost more but generate the same surface finish. So, the difference between tolerance required and the tool accuracy is calculated for each potential tool. Similar to the PROCESS.BAS selection program, the smaller the difference, the higher the probability for the selected tool.

Moreover, Twidrill.DAT and Reamer.DAT files are also extracted for checking the corresponding tool accuracies based on the drill size for hole making. If a twist drill or a reamer is accurate enough to drill a hole, then the tool ID of the selected type of twist drills and reamers, cost of tools and probabilities from knowledge base rules are extracted, and are appended to the OPERATE.DAT file. The flow chart of TOOL.BAS is shown in Figure 5.10
and the listing of TOOL.BAS is presented in Appendix E.

Figure 5.10  Flow chart of tool selection algorithm [TOOL.BAS]
5.3.8 Machining Parameter Selection

Cutting speeds and feeds in machining vary with the type of machining processes. The speed and feed of different machining processes are multiplied by the factors $Z_s$ and $Z_f$, respectively to give the actual value.

PARA.BAS program first determines those factors and the depth of cut by a series of rules. Workpiece materials, such as steel, free-machining steel, and cast iron and the tool materials, such as high speed steel and carbide will be chosen by the user. According to the required machining process, the program will retrieve those factors and calculate the cutting speed $[\text{SPEED}]$ in feet per minute (fpm) and the feed rate $[\text{FEED}]$ in inch per revolution (ipr).

\[
[SPEED] = V = V_s Z_s \quad (10)
\]

\[
[\text{FEED}] = F = f_f Z_f \quad (11)
\]

In addition, depth of cut $[\text{DOC}]$ for the corresponding machining process is assigned by PARA.BAS program.

Second, PARA.BAS program calculates the following factors given by Schey [17] and verifies the selected machine and tool in each operation:
Required spindle speed (rpm) for machining the surface can be calculated by the equation:

\[
\text{SPINDLE SPEED} = \frac{\text{CUTTING SPEED}}{\text{DIAMETER} \times \pi} \times 12 \quad \text{(rev/inch)} \quad (12)
\]

Cutting speed is the speed of a tool moving on the part surface.

Required Feedrate in machining is calculated by

\[
\text{FEED RATE} = \frac{\text{FEED} \times \text{SPINDLE}}{\text{FEEDRATE}} \quad (13)
\]

Required Power is calculated by

\[
\text{POWER (HP)} = \frac{E_1 \times h^{-0.3} \times \text{FEED} \times \text{SPEED} \times \text{DOC}}{\text{eff}} \quad (14)
\]

where
- \( E_1 \) = Specific Heat of Workpiece Material
- \( h' \) = chip thickness (in)
- \( \text{FEED} \) = Feed rate (in)
- \( \text{SPEED} \) = Cutting Speed on the Surface (fpm)
- \( \text{DOC} \) = Depth of cut (in)
- \( \text{eff} \) = Efficiency of the Machine

Required Cutting Force on machining is calculated by

\[
\text{Cutting Force} = \frac{\text{Power Required (HP)}}{\text{Cutting Speed (SPEED)}} \quad (15)
\]

The selected feed rate in turning and the nose radius on the tool have the greatest impact on the surface finish [11]. Increasing the nose radius or reducing the feed rate improves the surface finish.
The Generated Surface Roughness by indexable inserts for turning is calculated by

\[
\text{Generated Surface Roughness} = \frac{(\text{Feed})^2}{8 \times \text{Nose Radius}}
\]

(16)

For each operation in the OPERATE.DAT file, the program checks the above factors with the selected machine and tool specifications, such as, maximum and minimum limits on spindle speed, the limits of feed rate, cutting force and power, and nose radii of selected tools. A machine is only accepted if the following conditions are met:

a) \([\text{MINSP}] \leq [\text{SPINDLE}] \leq [\text{MAXSP}]\)

b) \([\text{POWER}] \leq [\text{MPOWER}]\)

c) \([\text{FORCE}] \leq [\text{MFORCE}]\)

d) \([\text{FEEDRATE}] \leq [\text{MFEED}]\)

A tool is being rejected when the generated surface roughness (GSR) is larger than specified surface finish from design.

If either a machine or a tool of the operation is rejected, the operation is also rejected from the list in OPERATE.DAT. Finally, when some operations are erased from the data file, a new OPERATE.DAT file will be created. The flow chart of PARA.BAS program is illustrated in Figure 5.11. The listing of the program is shown in Appendix F.
Figure 5.11. Flow chart of machining parameter selection and verification of machines and tools based on selected machining parameters.
5.3.9 Holding Device Selection

Having gone through all the verification procedures, a jig or fixture will be selected. According to the type of processes and the required geometric tolerances on the machining surface, as well as the selected machine, several types of jigs and fixtures can be found in the HOLDING.DAT file. The ID code of the selected jig or fixture with its unit cost will be retrieved and appended to the OPERATx.DAT file to form a complete operation. The program for retrieving holding device information is presented in Appendix G.

5.3.10 Process Capability Analysis

Statistical techniques can be helpful throughout the production process. Analysis of process variability relative to product requirements or specifications assists development and manufacturing in eliminating or greatly reducing this variability. This general activity is called Process-Capability Analysis.

Process Capability is a measure of the uniformity of output. It is customary to take the 6-sigma spread in the distribution of the product quality characteristic as a measure of process capability. When the quality characteristic has a normal distribution with mean \( \mu \) and standard deviation \( \sigma \), the upper and lower natural tolerance limits can be calculated as \( (\mu + 3\sigma) \) and \( (\mu - 3\sigma) \), respectively. That is,

\[
\text{UNTIL} = \mu + 3\sigma
\]
\[
\text{LNTL} = \mu - 3\sigma
\]
For a normal distribution, the natural tolerance limits include 99.73% of the output, and only 0.27% of the process output will fall outside the natural tolerance limits. The SPC package collects the data from an on-line production, takes the measurements of a number of samples, and calculates the sample mean $\bar{x}$ and its variance $s^2$. Four possible cases related to the natural tolerance limits (UNTL and LNTL) and the specification limits (USL and LSL) are generated.

Case 1: $\text{UNTL} < \text{USL}$ and $\text{LNTL} > \text{LSL}$

The operation produces the parts within the design tolerance ranges. There is only 0.27% chance of production that falls beyond the natural tolerance limits.

Case 2: $\text{UNTL} > \text{USL}$ and $\text{LNTL} < \text{LSL}$

The operation will produce a surface with the natural tolerance ranges which are larger than the specification limits. This operation does not seem to be as efficient as the one in Case 1.

Case 3: $\text{UNTL} > \text{USL}$ and $\text{LNTL} > \text{LSL}$ or $\text{UNTL} < \text{USL}$ and $\text{LNTL} < \text{LSL}$

The operation is similar to Case 2, except only one of the natural tolerance limits exceeds the specification limits. However, this operation is still not acceptable.
Case 4: \( \text{UNTL} < \text{USL and LNTL} \gg \text{LSL} \)

The natural tolerance limits are much smaller than the specification limits. It means the operation is too precise for producing the required specification limits. Economically, this operation is less preferred than the one in Case 1.

Alternatively, process capability can be expressed as a percentage of defectives \( (P_{\text{total}}) \). \text{SPC.BAS} is developed to generate the process-capability analysis by retrieving the statistical data from \text{SPC.DAT}. The program assigns the probability of preference to each alternative operation. The overall probability of reference can be calculated according to two factors:

(i) difference between \text{UNTL} and \text{USL}, and difference between \text{LNTL} and \text{LSL};

(ii) values of \( P_{\text{total}} \).

In order to represent both factors evenly in probability, each factor will have a maximum of 50% of the proportion. As for the difference between natural tolerance limits and specification limits, the probability of preference is formulated as

\[
1 - \left[ \frac{\text{USL} - \text{UNTL}}{\text{USL} - \text{LSL}} \right] \quad \text{which is equal to a value between 0 and 1.}
\]

The larger the difference, the smaller probability of preference will be given.
For the value of $P_{\text{total}}$, the higher the $P_{\text{total}}$ value, the lower the probability of preference will be. The overall probability of preference can be called an operation index which is calculated by the equation:

\[
\text{OPERATION INDEX} = \frac{1}{1 - \frac{(USL-UNTL)+(LNTL-LSL)}{USL-LSL}} + \frac{1}{2 \cdot (1-P_{\text{total}})} \quad (17)
\]

It is assumed that the minimum acceptable operation index is 0.7 or 70%. Those operations with unacceptable operation index will be eliminated from OPERATE.DAT file. Then, the file is updated for the next procedures. The flow chart of SPC.BAS program is shown in Figure 5.12, and the listing of the program is shown on Appendix H.

### 5.3.11 Alternative Process Plan Generation (COMBINE.BAS)

After the number of alternative operations has been reduced as much as possible from the OPERATE.DAT, process plans without operation sequence are generated by COMBINE.BAS. The program generates the combination of operations. For instance, if a machining surface has more than one alternative operation, then a preliminary process plan will show different operations for producing that surface. The total number of preliminary process plans is the product of the number of alternative operations for each operation in the surface. The alternative process plans without operation sequence are stored in the PREPLAN.DAT file for further analysis. Appendix I shows the listing of
the COMBINE.BAS program.

Figure 5.12 Flow chart for process-capability analysis (SPC.BAS)
5.3.12 Operation Sequence Generation
(SEQUENCE.BAS)

The SEQUENCE.BAS program retrieves the preliminary process plan from PREPLAN.DAT. The program first assigns a datum surface as the first surface to be machined in an operation sequence. Then, the rest of the machining surfaces are sorted according to their dimensional tolerances. If one surface is finer than the other, it will be machined later. Some conditions are considered in the operation sequence. For instance, a hole has to be drilled before threads are made by tapping, or milling has to be done on a surface before any other operations are applied on the surface. So, the program relocates the operations in the right logical sequence and writes the output in PLAN.DAT file for cost evaluation. The SEQUENCE.BAS program is listed in Appendix J.

5.3.13 Cost Estimation Program (COST.BAS)

Production cost of a process plan includes setup cost, machining cost, tool cost and the cost of holding device. These costs are mainly the direct cost that is allocated to the part and is proportional to the batch size and the time spent in production.

Setup cost \((SC)\) depends on the loading and unloading time \(t_L\). The setup is performed either by workers or by robot arm, a cost factor \(R_L\) which represents the hourly rate of the setup cost. Since a machine is set up for a certain batch size \(B\) of a part, setup cost is
defined as:

\[ SC = \frac{t_L R_t}{B} \]  \hspace{1cm} (18)

Machining cost (MC) is the sum of the labor cost and the cost of electricity for machining a part. The labor cost is the cutting time per part \( t_C \) which is charged at a rate of labor \( R_L \) (\$/hour). The cutting time \( t_C \) is simply the length of cut (CL) divided by the feed rate (FEED) as shown in the following equation:

\[ t_C = \frac{CL}{[\text{FEED}]} \]  \hspace{1cm} (19)

The utility cost is the product of electricity cost \( R_e \), the power required (POWER), and the cutting time \( t_C \). Therefore,

\[ MC = t_C \left[ R_L + R_e \text{POWER} \right] \]  \hspace{1cm} (20)

Tool cost (TC) is calculated as follows:

\[ TC = \left[ \frac{t_{ch} R_t + C_t}{N_t} \right] \]  \hspace{1cm} (21)

where \( t_{ch} \) is the tool change time, \( C_t \) is the tool cost and \( N_t \) is the number of parts produced by the tool.

Cost of holding device (HC) is estimated in the unit per piece and is given in the holding device data file HOLDING.DAT. Therefore, total cost for each operation is equal to

\[ \text{TCOST} = SC + MC + TC + HC \]  \hspace{1cm} (21)
Finally, the sum of the total cost of all the operations is the estimated production cost for the process plan. The COST.BAS program retrieves data from different data files or from user input for the calculation of production cost as shown in Table 5.20.

Table 5.20 Data location of the COST.BAS program

<table>
<thead>
<tr>
<th>Data Entry from</th>
<th>Data Retrieved</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIMTOL.DAT</td>
<td>Raw material diameter, RDIA</td>
</tr>
<tr>
<td></td>
<td>Diameter of part, DIA</td>
</tr>
<tr>
<td></td>
<td>Length of part, L</td>
</tr>
<tr>
<td></td>
<td>Width of prismatic part, W</td>
</tr>
<tr>
<td></td>
<td>Thickness of prismatic part, T</td>
</tr>
<tr>
<td>TOOL.DAT</td>
<td>Tool Change Cost, $t_{ch}$</td>
</tr>
<tr>
<td></td>
<td>Tool Cost, $C_t$</td>
</tr>
<tr>
<td></td>
<td>Tool Life, $T\text{LIFE}$</td>
</tr>
<tr>
<td>HOLDING.DAT</td>
<td>Loading and unloading time, $t_L$</td>
</tr>
<tr>
<td></td>
<td>Holding Device cost per piece, $HC$</td>
</tr>
<tr>
<td>PARA.DAT</td>
<td>Cutting speed, $SPEED$</td>
</tr>
<tr>
<td></td>
<td>Depth of cut, $DOC$</td>
</tr>
<tr>
<td></td>
<td>Feed, $FEED$</td>
</tr>
<tr>
<td>USER INPUT</td>
<td>Labour wage, $R_L$</td>
</tr>
<tr>
<td></td>
<td>Electricity rate, $R_e$</td>
</tr>
<tr>
<td></td>
<td>Hourly rate for setup and tool change, $R_t$</td>
</tr>
</tbody>
</table>

Using the above data, the COST.BAS program calculates the production cost for each proposed process plan. The calculated total cost for each process plan is presented on the EXSYS program as shown in Table 5.21. The program listing of COST.BAS is shown in Appendix K.
Table 5.21  The output of the COST.BAS program

<table>
<thead>
<tr>
<th>SEQ. NO.</th>
<th>SETUP COST</th>
<th>MACHINING COST</th>
<th>TOOL COST</th>
<th>FIXTURE COST</th>
<th>TOTAL COST/part</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.45</td>
<td>33.34</td>
<td>23.32</td>
<td>12.23</td>
<td>71.34</td>
</tr>
<tr>
<td>2</td>
<td>2.12</td>
<td>34.23</td>
<td>12.34</td>
<td>10.00</td>
<td>58.69</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL</td>
<td>15.32</td>
<td>130.22</td>
<td>102.20</td>
<td>89.20</td>
<td>336.94</td>
</tr>
</tbody>
</table>

Total cost of process plan #1 = $ 336.94/part

Considering the components of the total cost, such as setup cost, machining cost, tool cost and holding device cost for each operation, the user can select an alternative process plan based on the production constraints, like machining cost, tool cost, and holding device cost.
5.4 Knowledge Base Rules

The knowledge base which production rules generates the process plan. The rules are categorized as follows:

1. Set 1 rules (1-31) for selecting the type of processes based on geometric tolerances, features.
2. Set 2 rules (32-51) for selecting cutting tools and running external programs to generate alternative process plans.

5.4.1 Rules for Process Selection based on Geometric Tolerances

Geometric tolerances about an entity are recorded in the DIMTOL.DAT file. In order to select a more efficient process to produce the part, the user should also consider the geometric tolerances such as when the part is machined by precision boring and cylindrical grinding. Precision or finish boring is affected by either concentricity of the workpiece or the roundness of the bore, or both. For example,

Rule 1:

\[
\text{IF } [\text{CONCENTRICITY}] \geq 0.0002
\]

\[
\text{THEN } [\text{PROC(GEO.TOL)}] = \text{F_BORING}
\]

and \[
[\text{CALL PROCESS}] \text{ IS GIVEN THE VALUE [CPROCESS]}
\]

Notes:
The values of tolerances can only be obtained when all the factors involved are reasonably favorable to the operation. Improving these values with temperature control, special spindles and other precision adaptions may need.

Reference:
Grinding operation can be applied to produce a fine surface. Cylindrical grinding is especially performed for external or internal surfaces of a round workpiece. Roundness (circularity) and straightness for a rotational workpiece can be the criteria for selecting grinding to produce a surface. According to rules 3 and 4, finish or precision grinding is more suitable for either straightness or roundness which is less than 75 micro-inches. Otherwise, regular or rough grinding is already appropriate. For instance,

Rule 3:

IF \[ \text{ROUNDNESS} \geq 0.00075 \]

THEN \[ \text{PROC(GEO.TOL) IS R.GRINDING and F.GRINDING} \]
and \[ \text{[CALL PROCESS] IS GIVEN THE VALUE [PROCESS]} \]

Note:
External program PROCESS is going to generate combined processes for the surface

Reference:

5.4.2 Rules for Process Selection based on Feature

Feature is a general term applied to a physical portion of a part and may include one or more surface(s) such as holes, pins, screw threads, faces or slots [18]. Some features require one or more particular machining processes to manufacture. All the combinations of qualifiers form the premises of a set of nine rules. The actions for the premises are the appropriate features and the machining processes. The rules are applied after the

92
processes for all the machining surfaces are retrieved from PROCESS.OUT file. The features and machining processes available in the rules are listed in the Table 5.22.

<table>
<thead>
<tr>
<th>Feature</th>
<th>Possible Machining Processes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1) THROUGH HOLE</td>
<td>Rough or Finish Boring, Honing Drill, Reaming, Broaching, Rough or Finish Grinding</td>
</tr>
<tr>
<td>2) BLIND HOLE</td>
<td>Rough or Finish Boring, Rough or finish Grinding, reaming</td>
</tr>
<tr>
<td>3) TAPER</td>
<td>Taper boring or turning</td>
</tr>
<tr>
<td>4) THREAD</td>
<td>Single-point threading, thread chasing</td>
</tr>
<tr>
<td>5) CYLINDRICAL SURFACE</td>
<td>Turning, Rough or Finish Grinding</td>
</tr>
<tr>
<td>6) PLANE</td>
<td>Milling</td>
</tr>
<tr>
<td>7) FACE</td>
<td>Facing, Parting</td>
</tr>
<tr>
<td>8) COUNTERSINK</td>
<td>Drilling</td>
</tr>
<tr>
<td>9) CHAMFER</td>
<td>Chamfering</td>
</tr>
</tbody>
</table>

A rule for process selection based on feature is shown as follows:

Rule 5:

IF SHAPE OF THE PART FEATURE IS ROTATIONAL and COMPLEXITY OF THE PART IS UNIFORM and SURFACE IS INTERNAL

THEN FEATURE IS HOE and PROCESS BASED ON FEATURE IS F BORING and R BORING and F BORING and BROACHING and DRILLING and HONING and REAMING

Reference:
5.4.3 Rules for Process Selection based on Dimension

Given an entity in a drawing that has been defined by its feature, tolerances and surface finish, a list of possible machining processes can be obtained. However, without considering the dimension of the entity, it may not be possible to overcome some special cases. For instance, according to the size of the commonly used tools in drilling, a very large hole can only be machined by boring instead of drilling. Also, a large cylindrical part with low surface quality can also be recommended for casting rather than any machining operation.

There are some rules which consider the dimensions of the part in process selection after the processes have been evaluated based on feature, tolerance and surface finish.

Rule 20:

IF FEATURE IS HOLE
and [DIAMETER] < 0.005
and [TOLERANCE] < 0.0005

THEN PROCESS IS HONING - Probability=8/10

Note:

Honing is more accurate than broaching or reaming to drill a small hole with close tolerance.

Reference:
Rule 21:

IF FEATURE IS TAPER or CYLINDRICAL SURFACE
and [diameter] > 20
and [tolerance] > 0.005
and [surface roughness] > 16

THEN PROCESS IS CASTING Probability=8/10
and Machine, tool, machining parameters and cost
are not determined by the expert system.
and STOP

Note: Due to a large size and low surface quality
is required for the finished part, casting is
more productive and effective.

Reference:
Schey, "Introduction to Manufacturing Processes,"

5.4.4 Rules for Combining [PROCESS] and
[PROC(FEATURE)]

Processes recommended based on feature should
dominate the final result in process selection module. It
is because the geometry of the workpiece will restrict
certain types of processes that can be chosen. Several
rules are designed for improving the effectiveness of the
machining processes in a process plan. The following rules
demonstrate the effect of the COM-PROCESS on the choice of
machining process based on feature. The COM-PROCESSes are
generated from tolerance and surface roughness requirement.

If there is only one possible machining
process generated based on feature, for instance, threading,
countersink or chamfering, then the COM-PROCESS [PROCESS]
could not change the decision from feature [PROC(FEATURE)].
So, there is no rule to combine PROCESS and PROC(FEATURE)
for those processes.
Rule 25:

IF FEATURE IS CYLINDRICAL SURFACE
and [PROCESS] = R_Boring
and PROC(FEATURE) IS R_Boring
and PROC(FEATURE) IS F_Boring

THEN PROCESS IS R_Boring - Probability = 9/10
and [Call LATHE] IS GIVEN THE VALUE OF [MACHINE]
and [TOOL_TYPE] IS GIVEN THE VALUE CYLINDRICAL

Note: Based on the surface requirement, a rough cut is already enough. Then, machines are selected when a machining process has been defined.

5.4.5 Rules for Tool Selection

The rules for tool selection in the expert system are called Set 2 Rules. The rules first assign some special tools for grinding and broaching. Then, some rules are going to notify the TOOL.BAS program to select the specified tool type. The rules then classify four types of tools with the given process type. The tool ID is directly retrieved as [TOOL]. Some other rules for tool selection in hole making and threading give appropriate tools after the TOOL.BAS program has been checked the tool accuracies with the TWIST.DAT and REAMER.DAT files. The qualifier [TOOL] is retrieved from the OPERATE.DAT. If tool accuracy is not suitable for the process, the [TOOL] is a blank space. Otherwise, [TOOL] is defined as either TWIST or REAM. Then, the set 2 rules continue to assign the type of twist drills or reaming tools based on the criteria given from the handbook [12]. After the tools are defined, the following actions are initiated to run external programs to
select machining parameters and holding devices, and generate alternative process plans for the EXSYS. Examples of the set 2 rules for selection of drilling tools and turning tools are listed as follows:

Rule 34:

IF

[TOOL_TYPE] = HOLE
and PROCESS IS DRILLING - Probability=7/10

THEN

[TCODE] IS GIVEN THE VALUE TWIST
and [CALL TOOL] IS GIVEN THE VALUE OF [TOOL]

Note:
When drilling tool is needed, TOOL program checks the tool accuracy with the size of the drilled hole.

Rule 38:

IF

[TOOL] = TWIST
and [TCODE] = TWIST
and WORKPIECE MATERIAL IS HARD

THEN

TOOL IS STRAIGHT-SHANK - Probability=7/10
and [CALL PARA] IS GIVEN THE VALUES OF [SPEED], [FEED], [DOC]
and [CALL HOLDING] IS GIVEN THE VALUE [HOLDING]
and [CALL SPC] IS GIVEN THE VALUE OF [INDEX]
and [CALL COMBINE] IS GIVEN THE VALUE OF [TCOST]
and STOP

Note:
Straight-Shank is more suitable for hard materials while taper-shank is a general purpose tool.

Reference:
Tool and Manufacturing Engineers Handbooks, 4th ed., V.1, SME, 1983, pp.9-16 to 9-17.

Rule 46:

IF

[TOOL_TYPE] = CYLINDRICAL

THEN

[Call TOOL] IS GIVEN THE VALUE OF [TOOL]
and [CALL PARA] IS GIVEN THE VALUES OF [SPEED], [FEED], [DOC]
and [CALL HOLDING] IS GIVEN THE VALUE [HOLDING]
and [CALL SPC] IS GIVEN THE VALUE OF [INDEX]
and [CALL COMBINE] IS GIVEN THE VALUE OF [TCOST]
and STOP
5.5 Inference Engine

The inference engine is the mechanism that guides the program through its knowledge base, and links the above sets of rules effectively. The link is directed from the Set 1 to Set 2 rules with the algorithms and external programs. A complete flow chart about the proposed expert system for process planning is shown on Figure 5.13. The inference engine starts with TOLEXV module to extract and verify the dimensional and geometrical tolerance for each machining surface, and then runs the CAGT program to draw all the new tolerances into the CAD drawing.

Second, set 1 rules select a process based on dimensional and geometric tolerances, and surface finish for each machining surface in DIMTOL.DAT. The PROCESS.BAS, SURF-TOL.BAS and JOIN-T&S.BAS programs are applied to generate a list of potential processes, that will be stored as the COM-PROCESS, with their probabilities of preference. Some processes that are generated based on the features of part, are combined with COM-PROCESS. So, the outputs are a number of processes for each machining surface. The rules continue to call LATHE.BAS and MCCENTER.BAS programs to select machines and assign their probabilities.

Third, set 2 rules assign alternative tools for each record in OPERATE.DAT file. The TOOL.BAS program is applied with the required tool type and machining process. Tool ID for each selected tool is sent into the knowledge base. Then, machining parameters are selected by PARA.BAS
Figure 5.13  System flow chart of the proposed expert system
program based on process, material of workpiece and tool. Selected machines and tools in each operation in OPERATE.DAT are verified by machining parameters. Some operations with machine and tool are rejected from the OPERATE.DAT file. Afterwards, the holding devices for the remaining operations are selected by HOLDING.BAS program.

Fourth, process-capability analysis is performed by the SPC.BAS program to calculate the operation index for each remaining operation. If the operation index is less than 7 out of 10, the operation is removed from the OPERATE.DAT file, because, in a practical sense, the selected operation does not perform as well as the handbooks mention. Remaining operations in the file are organized in different preliminary process plans. Operations in each preliminary process plan are finally sequenced as different alternative process plans. Then, the total cost required for each process plan is calculated by the COST.BAS program.

The EXSYS program only presents alternative process plans for the user, no decision making intelligence is available. So, the user can make a decision on whether to accept one of the alternatives or to change the input conditions of machine(s), tool(s) and holding device(s). If any modification in the input conditions is required, then the expert system generates a new process plan so the user can compare with the previous one.
CHAPTER VI

APPLICATION OF THE EXPERT SYSTEM TO THE CASE STUDY

A case study illustrating the expert system described above was derived from the information provided by a manufacturing company. A rotational part is chosen as a sample part for process planning.

6.1 Description of the Sample Part

Figure 6.1 shows the geometry of the sample part. It is a shaft-like part with a threaded end which is mated with an external member in the assembly. Basic geometric features on the external surface are chamfers, fillets, neck and thread. The length of the raw material is the total length of the finished part, and the diameter of the raw material is equal to the biggest diameter of the part. The material of the rotational part and the matching part is aluminium and no geometric tolerance is given from the drawing.

6.2 Tolerance Extraction and Verification

The expert system first runs the CAD system as an external program to input the part drawing. A PART.DXF file which is shown in Appendix L, is created by using the command DXFOUT. Then, the TOLEXV module starts to extract on each surface from the DXF file. Eight surfaces with their dimensions and tolerances are retrieved, and two
Figure 6.1. Geometry of the Sample Port
cylindrical surfaces are assigned with surface roughness during the extraction.

According to the Case 1: common point dimensioning method in dimensional tolerance verification, three more surfaces with dimensions and tolerances are defined. After the verification of the dimensional tolerance is done, new tolerance data for each surface with dimensions are stored in DIMTOL.DAT as shown in Table 6.1.

<table>
<thead>
<tr>
<th>SURFACE NO.</th>
<th>DIMENSION</th>
<th>TOLERANCE</th>
<th>SURFACE ROUGHNESS</th>
<th>DATUM</th>
<th>GEOMETRIC TOLERANCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.1225</td>
<td>0.0005</td>
<td></td>
<td>A</td>
<td>CY = 0.001</td>
</tr>
<tr>
<td>2</td>
<td>0.7485</td>
<td>0.0005</td>
<td></td>
<td>B</td>
<td>CO = 0.0005</td>
</tr>
<tr>
<td>3</td>
<td>0.1250</td>
<td>----------</td>
<td></td>
<td>N</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1.1250</td>
<td>0.0030</td>
<td>80</td>
<td>N</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.1875</td>
<td>----------</td>
<td></td>
<td>N</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2.2125</td>
<td>0.0040</td>
<td></td>
<td>N</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>2.8750</td>
<td>0.0050</td>
<td></td>
<td>N</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.1250</td>
<td>----------</td>
<td></td>
<td>N</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>1.0000</td>
<td>0.0030</td>
<td></td>
<td>N</td>
<td>CY = 0.001</td>
</tr>
<tr>
<td>10</td>
<td>1.5625</td>
<td>0.0070</td>
<td>60</td>
<td>N</td>
<td>CO = 0.0005</td>
</tr>
<tr>
<td>11</td>
<td>0.6250</td>
<td>0.0090</td>
<td></td>
<td>N</td>
<td></td>
</tr>
</tbody>
</table>

CAGT module assigns the maximum cylindricity and the maximum concentricity for each of the two major cylindrical surfaces 4 and 10. The updated drawing is generated by the CAGT.LSP program and is presented in Figure 6.2. Surface finishes of the surface 4 and 10 are verified by the SUR-TOL.BAS program. However, the assigned surface
finish is already larger than 1/8 of the tolerance range, so, surface finish does not need to be changed.

6.3 Process, Machine, Tool and Machining Parameters Selection

The expert system runs the PROCESS.BAS program to generate processes based on dimensional tolerance and surface finish of each machining surface. The machining surfaces are assumed to be retrieved by any feature recognition program. So, actually there are only 7 surfaces that require machining operation in order to produce the finished part, because the longitude surface with the biggest diameter in the part comes from the raw material. Therefore, surfaces 4, 6, 7 and 9 are eliminated before processes are selected. Using the set 1 rules for assigning machining processes based on features, each machining surface has only one alternative process.

Then, LATHE.BAS is applied to select machines for each process. Since most of the machines are precise enough to be used, only machines with probability greater than 7 out of 10 are accepted in order to eliminate a large number of alternative operations at the end. The expert system continues to select tools by running the rules and the TOOL.BAS program. There are more than 25 alternative operations generated. When the PARA.BAS program is applied to select machining parameters, and to verify the machine and tool specification, only 15 alternatives are left as shown in Table 6.2. Different jigs are also assigned to
each operation before the process-capability analysis is performed. SPC.BAS program calculates the operation indices for those 15 alternative operations and removes 6 alternative operations. The remaining alternative operations are presented in Table 6.3.

Table 6.2 Verified operations generated from PARA.BAS

<table>
<thead>
<tr>
<th>OPER. No.</th>
<th>Surf. Proc. No.</th>
<th>Pr(p) Mach.</th>
<th>Pr(m)</th>
<th>Tool</th>
<th>SPEED (fpm)</th>
<th>FEED (ipr)</th>
<th>DOC (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>F</td>
<td>0.9</td>
<td>M113</td>
<td>0.79</td>
<td>T101</td>
<td>330</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>F</td>
<td>0.9</td>
<td>M113</td>
<td>0.79</td>
<td>T102</td>
<td>330</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>F</td>
<td>0.9</td>
<td>M113</td>
<td>1.00</td>
<td>T101</td>
<td>330</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>F</td>
<td>0.9</td>
<td>M113</td>
<td>1.00</td>
<td>T103</td>
<td>330</td>
</tr>
<tr>
<td>5</td>
<td>3</td>
<td>C</td>
<td>0.9</td>
<td>M113</td>
<td>1.00</td>
<td>T103</td>
<td>290</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>C</td>
<td>0.9</td>
<td>M113</td>
<td>1.00</td>
<td>T110</td>
<td>290</td>
</tr>
<tr>
<td>7</td>
<td>5</td>
<td>TT</td>
<td>0.9</td>
<td>M113</td>
<td>1.00</td>
<td>T106</td>
<td>350</td>
</tr>
<tr>
<td>8</td>
<td>8</td>
<td>FT</td>
<td>0.9</td>
<td>M113</td>
<td>0.65</td>
<td>T104</td>
<td>360</td>
</tr>
<tr>
<td>9</td>
<td>8</td>
<td>FT</td>
<td>0.9</td>
<td>M113</td>
<td>0.65</td>
<td>T105</td>
<td>360</td>
</tr>
<tr>
<td>10</td>
<td>8</td>
<td>FT</td>
<td>0.9</td>
<td>M114</td>
<td>0.63</td>
<td>T104</td>
<td>360</td>
</tr>
<tr>
<td>11</td>
<td>8</td>
<td>FT</td>
<td>0.9</td>
<td>M115</td>
<td>0.63</td>
<td>T104</td>
<td>360</td>
</tr>
<tr>
<td>12</td>
<td>10</td>
<td>FG</td>
<td>0.9</td>
<td>M113</td>
<td>0.79</td>
<td>T109</td>
<td>335</td>
</tr>
<tr>
<td>13</td>
<td>10</td>
<td>FG</td>
<td>0.9</td>
<td>M113</td>
<td>0.79</td>
<td>T110</td>
<td>335</td>
</tr>
<tr>
<td>14</td>
<td>11</td>
<td>T</td>
<td>0.9</td>
<td>M113</td>
<td>0.79</td>
<td>T111</td>
<td>150</td>
</tr>
<tr>
<td>15</td>
<td>11</td>
<td>RT</td>
<td>0.9</td>
<td>M113</td>
<td>0.81</td>
<td>T106</td>
<td>300</td>
</tr>
</tbody>
</table>

F = FACING  C = CHAMFERING  T = THREADING
RT = ROUGH TURNING  FT = FINISH TURNING
FG = ROUGH GRINDING  TT = TAPER TURNING

Referring to Table 6.3, surface 8 can be produced in a process with two different operation 9 and 10. Therefore, two process plans (PREPLAN) with different operations are generated by the COMBINE.BAS program as shown in Table 6.4.
Table 6.3  Remaining alternative operations after process capability analysis

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>F</td>
<td>0.9</td>
<td>M113 0.79</td>
<td>101 330</td>
<td>0.010</td>
<td>0.004</td>
<td>F11 0.76</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>F</td>
<td>0.9</td>
<td>M113 1.00</td>
<td>103 330</td>
<td>0.010</td>
<td>0.004</td>
<td>F13 0.81</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>C</td>
<td>0.9</td>
<td>M113 1.00</td>
<td>110 290</td>
<td>0.005</td>
<td>0.004</td>
<td>F12 0.74</td>
</tr>
<tr>
<td>7</td>
<td>5</td>
<td>TT</td>
<td>0.9</td>
<td>M113 1.00</td>
<td>106 350</td>
<td>0.010</td>
<td>0.005</td>
<td>F13 0.83</td>
</tr>
<tr>
<td>9</td>
<td>8</td>
<td>FT</td>
<td>0.9</td>
<td>M113 0.65</td>
<td>105 360</td>
<td>0.008</td>
<td>0.005</td>
<td>F11 0.81</td>
</tr>
<tr>
<td>10</td>
<td>8</td>
<td>FT</td>
<td>0.9</td>
<td>M114 0.63</td>
<td>104 360</td>
<td>0.008</td>
<td>0.005</td>
<td>F11 0.83</td>
</tr>
<tr>
<td>12</td>
<td>10</td>
<td>FG</td>
<td>0.9</td>
<td>M113 0.79</td>
<td>109 335</td>
<td>0.001</td>
<td>0.003</td>
<td>F16 0.80</td>
</tr>
<tr>
<td>14</td>
<td>11</td>
<td>T</td>
<td>0.9</td>
<td>M113 0.79</td>
<td>111 150</td>
<td></td>
<td></td>
<td>F11 0.73</td>
</tr>
<tr>
<td>15</td>
<td>11</td>
<td>RT</td>
<td>0.9</td>
<td>M113 0.81</td>
<td>106 300</td>
<td>0.008</td>
<td>0.016</td>
<td>F12 0.74</td>
</tr>
</tbody>
</table>

Table 6.4  List of operations in both PREPLANS

<table>
<thead>
<tr>
<th>COMBINATIONS OF PREPLANS</th>
</tr>
</thead>
<tbody>
<tr>
<td>SURFACE NUMBER</td>
</tr>
<tr>
<td>----------------</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>3</td>
</tr>
<tr>
<td>5</td>
</tr>
<tr>
<td>8</td>
</tr>
<tr>
<td>10</td>
</tr>
<tr>
<td>11</td>
</tr>
<tr>
<td>15</td>
</tr>
</tbody>
</table>

The sequences of operations in both PREPLANS are generated by SEQUENCE.BAS program. There are a large number of alternative sequences for 8 operations, but, in fact, only two alternatives exist because there are some precedence relationships among the operations as follows:
(i) Operation 1 (Facing) for surface 1 has to be done before all the other surfaces because surface 1 is the datum surface.

(ii) Operation 2 (Chamfering) for surface 3 should be done after the facing operation.

(iii) Since the dimensional tolerance for surface 11 is small, this surface should be done last.

(iv) Before the thread in surface 11 is machined, surface 8 has to been machined first to generate a runout for the thread.

(v) Turning operation for surface 11 also needs to be done prior to the thread.

(vi) Operation 4 (Facing) in surface 2 needs to be done before the surface 11 is machined.

Therefore, operations 1 and 6 are grouped as the first set of operations in the sequence, while operations 4, 15, 9 and 14 or operations 4, 15, 10 and 14 are the last set of operations in the sequence. If only operations 7 and 12 remain in between two sets of operations, then, two alternative process plans can be generated from each PREPLAN.

Finally, alternative process plans are generated, and the production cost per part for each process plan is estimated for comparison. The cost implies that if operation 7 is performed before operation 12, then the cost is relatively higher than the others. Moreover, operation 10, which has a higher operation index, requires low cost of production. User can consider more details in each alternative process plan, which is shown on screen as Table 6.6, before a decision is made. For instance, referring to Table 6.5, the cost for alternative B(?) is the slightly
lower than that of alternative A(2) because of the tool cost of the tool T105 is much higher than that of the tool T106 for the finish cut on surface 8. However, alternative A(2) only involves one machine (M113) to produce the whole part. If the part will be produced in a machining cell and tool T105 is available, then alternative A(2) will be more effective for only using one machine to produce the part.

Table 6.5 Summary of alternative process plan

<table>
<thead>
<tr>
<th>SEQUENCES OF OPERATIONS IN EACH PREPLAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALTERNATIVE PROCESS PLAN</td>
</tr>
<tr>
<td>OPERATION NUMBER</td>
</tr>
<tr>
<td>------------------</td>
</tr>
<tr>
<td>10</td>
</tr>
<tr>
<td>20</td>
</tr>
<tr>
<td>30</td>
</tr>
<tr>
<td>40</td>
</tr>
<tr>
<td>50</td>
</tr>
<tr>
<td>60</td>
</tr>
<tr>
<td>70</td>
</tr>
<tr>
<td>80</td>
</tr>
<tr>
<td>TOTAL COST</td>
</tr>
</tbody>
</table>
### Table 6.6 Process Plan for each alternative process plan

<table>
<thead>
<tr>
<th>OPER. SURF. NO.</th>
<th>PROCESS</th>
<th>Mach. Tool</th>
<th>SPEED (fpm)</th>
<th>FEED (inch)</th>
<th>DOC (in)</th>
<th>JIG</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>Facing</td>
<td>M113 T101</td>
<td>330</td>
<td>0.010</td>
<td>0.004</td>
<td>F11</td>
</tr>
<tr>
<td>20</td>
<td>Chamfering</td>
<td>M113 T110</td>
<td>290</td>
<td>0.005</td>
<td>0.004</td>
<td>F12</td>
</tr>
<tr>
<td>30</td>
<td>Taper Turn</td>
<td>M113 T106</td>
<td>350</td>
<td>0.010</td>
<td>0.005</td>
<td>F13</td>
</tr>
<tr>
<td>40</td>
<td>Finish Turn</td>
<td>M113 T104</td>
<td>360</td>
<td>0.008</td>
<td>0.005</td>
<td>F11</td>
</tr>
<tr>
<td>50</td>
<td>Facing</td>
<td>M113 T103</td>
<td>330</td>
<td>0.010</td>
<td>0.004</td>
<td>F13</td>
</tr>
<tr>
<td>60</td>
<td>Rough Turn</td>
<td>M113 T106</td>
<td>300</td>
<td>0.008</td>
<td>0.016</td>
<td>F12</td>
</tr>
<tr>
<td>70</td>
<td>Finish Turn</td>
<td>M113 T105</td>
<td>360</td>
<td>0.008</td>
<td>0.005</td>
<td>F11</td>
</tr>
<tr>
<td>80</td>
<td>Threading</td>
<td>M113 T111</td>
<td>150</td>
<td>-----</td>
<td>-----</td>
<td>F11</td>
</tr>
</tbody>
</table>

Total production cost per part = $50.45

### Process Plan A(2)

<table>
<thead>
<tr>
<th>OPER. SURF. NO.</th>
<th>PROCESS</th>
<th>Mach. Tool</th>
<th>SPEED (fpm)</th>
<th>FEED (inch)</th>
<th>DOC (in)</th>
<th>JIG</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>Facing</td>
<td>M113 T101</td>
<td>330</td>
<td>0.010</td>
<td>0.004</td>
<td>F11</td>
</tr>
<tr>
<td>20</td>
<td>Chamfering</td>
<td>M113 T110</td>
<td>290</td>
<td>0.005</td>
<td>0.004</td>
<td>F12</td>
</tr>
<tr>
<td>30</td>
<td>Finish Turn</td>
<td>M113 T104</td>
<td>360</td>
<td>0.008</td>
<td>0.005</td>
<td>F11</td>
</tr>
<tr>
<td>40</td>
<td>Taper Turn</td>
<td>M113 T106</td>
<td>350</td>
<td>0.010</td>
<td>0.004</td>
<td>F13</td>
</tr>
<tr>
<td>50</td>
<td>Facing</td>
<td>M113 T103</td>
<td>330</td>
<td>0.010</td>
<td>0.004</td>
<td>F13</td>
</tr>
<tr>
<td>60</td>
<td>Rough Turn</td>
<td>M113 T106</td>
<td>300</td>
<td>0.008</td>
<td>0.016</td>
<td>F12</td>
</tr>
<tr>
<td>70</td>
<td>Finish Turn</td>
<td>M113 T105</td>
<td>360</td>
<td>0.008</td>
<td>0.005</td>
<td>F11</td>
</tr>
<tr>
<td>80</td>
<td>Threading</td>
<td>M113 T111</td>
<td>150</td>
<td>-----</td>
<td>-----</td>
<td>F11</td>
</tr>
</tbody>
</table>

Total production cost per part = $42.90
Table 6.6  Continued

<table>
<thead>
<tr>
<th>OPER. NO.</th>
<th>SURF. NO.</th>
<th>PROCESS</th>
<th>Mach. Tool</th>
<th>SPEED (fpm)</th>
<th>FEED (inch)</th>
<th>DOC JIG</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1</td>
<td>Facing</td>
<td>M113 T101</td>
<td>330</td>
<td>0.010</td>
<td>0.004 F11</td>
</tr>
<tr>
<td>20</td>
<td>3</td>
<td>Chamfering</td>
<td>M113 T110</td>
<td>290</td>
<td>0.005</td>
<td>0.004 F12</td>
</tr>
<tr>
<td>30</td>
<td>5</td>
<td>Taper Turn</td>
<td>M113 T106</td>
<td>350</td>
<td>0.010</td>
<td>0.005 F13</td>
</tr>
<tr>
<td>40</td>
<td>10</td>
<td>Finish Turn</td>
<td>M113 T104</td>
<td>360</td>
<td>0.008</td>
<td>0.005 F11</td>
</tr>
<tr>
<td>50</td>
<td>2</td>
<td>Facing</td>
<td>M113 T103</td>
<td>330</td>
<td>0.010</td>
<td>0.004 F13</td>
</tr>
<tr>
<td>60</td>
<td>11</td>
<td>Rough Turn</td>
<td>M113 T106</td>
<td>300</td>
<td>0.008</td>
<td>0.016 F12</td>
</tr>
<tr>
<td>70</td>
<td>8</td>
<td>Finish Turn</td>
<td>M114 T104</td>
<td>360</td>
<td>0.008</td>
<td>0.005 F11</td>
</tr>
<tr>
<td>80</td>
<td>11</td>
<td>Threading</td>
<td>M113 T111</td>
<td>150</td>
<td>-----</td>
<td>------- F11</td>
</tr>
</tbody>
</table>

Total production cost per part = $47.67

<table>
<thead>
<tr>
<th>OPER. NO.</th>
<th>SURF. NO.</th>
<th>PROCESS</th>
<th>Mach. Tool</th>
<th>SPEED (fpm)</th>
<th>FEED (inch)</th>
<th>DOC JIG</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1</td>
<td>Facing</td>
<td>M113 T101</td>
<td>330</td>
<td>0.010</td>
<td>0.004 F11</td>
</tr>
<tr>
<td>20</td>
<td>3</td>
<td>Chamfering</td>
<td>M113 T110</td>
<td>290</td>
<td>0.005</td>
<td>0.004 F12</td>
</tr>
<tr>
<td>30</td>
<td>10</td>
<td>Finish Turn</td>
<td>M113 T104</td>
<td>360</td>
<td>0.008</td>
<td>0.005 F11</td>
</tr>
<tr>
<td>40</td>
<td>5</td>
<td>Taper Turn</td>
<td>M113 T106</td>
<td>350</td>
<td>0.010</td>
<td>0.005 F16</td>
</tr>
<tr>
<td>50</td>
<td>2</td>
<td>Facing</td>
<td>M113 T103</td>
<td>330</td>
<td>0.010</td>
<td>0.004 F13</td>
</tr>
<tr>
<td>60</td>
<td>11</td>
<td>Rough Turn</td>
<td>M113 T106</td>
<td>300</td>
<td>0.008</td>
<td>0.016 F12</td>
</tr>
<tr>
<td>70</td>
<td>8</td>
<td>Finish Turn</td>
<td>M114 T104</td>
<td>360</td>
<td>0.008</td>
<td>0.005 F11</td>
</tr>
<tr>
<td>80</td>
<td>11</td>
<td>Threading</td>
<td>M113 T111</td>
<td>150</td>
<td>-----</td>
<td>------- F11</td>
</tr>
</tbody>
</table>

Total production cost per part= $40.85
 CHAPTER VII

CONCLUSIONS

An expert system approach in process planning has been presented in this study. The expert system integrates the combined effects of process selection, machine selection, tool selection, machining parameters, and tolerance requirements in developing more efficient alternative process plans. The knowledge base in the expert system serves as the intelligence to interpret and relate different databases, algorithms and production rules. The resulting data structure can be accessed by existing functions, such as NC programming, bill of materials and plant layout. The main advantages of the proposed expert system over existing ones are its capability:

1) to verify the input values of tolerances by the user or from an AUTOCAD drawing;

2) to check tolerance values against standards (CSA-B78.2-86);

3) to generate four types of geometric tolerances: straightness, circularity (roundness), cylindricity and concentricity;

4) to generate a limited number of operations for the statistical process control analysis;

5) to generate alternative process plans in which each operation has an operation index;
6) to calculate the cost for each process.

The proposed expert system should lead to the development of a systematic means for automatic CAD/CAPP integration from given CAD drawing, machine and tool databases.

In order to implement the system into industry, several actions have to be taken by the company. First, a CAD system should be available for generating part drawing and DXF file for extraction and tolerance verification. Second, the database files in industry should adopt the proposed format described in the thesis. Third, process capability information should be extracted from existing SPC packages in the company.

Although the proposed expert system has shown many improvements compared to existing ones and has expanded the horizon in dealing with the tolerance verification problem, further studies needed to be investigated, particularly with respect to the following:

1) implementation of the expert system in industry to test the practicality of the system;

2) development of an interface program to integrate CAD and CAPP through a more general neutral file, such as IGES.

3) expanding the number of rules and the size of the databases.
REFERENCES


APPENDICES
APPENDIX A

LISTING OF TOLEKV.BAS PROGRAM
10  DIM  IN$(5500),DIMINDEX$(100),DIMEN$(100),STATE$(30),
    STATE(30),XI(30),X2(30),Y1(30),Y2(30),D(70),T(70),NEWD(70),
    NEWT(70),NEWX1(33),NEWX2(30),NEWY1(30),NEWY2(30),C10$(30),
    C20$(33),C13$(30),C23$(30)
20  DIM  NEWXN(30),NEWXN(30),X3(30),Y3(30),XM(30),YM(30)
30  DIM  OSS$(30),DIFF1(30),DIFF2(30),DIFF(30),YRELATIVE(30),
    RELD(30),ST(30),CI(30),CY(33),STS(30),C1$(30),CY$(30),MAXST(30),
    MHS(30),MAXCI(30),MAXCY(30),MAXCO(30)
40  CLS
50  INPUT "PLEASE ENTER THE DECIMAL PLACES OF DIMENSIONS AND
    TOLERANCES -> ",DEC
60  DES = DEC + 2
70  PRINT ""
80  PRINT "ENTITY DIMENSION TOLERANCE STATE  X1    Y1"
90  PRINT "X2    Y2 LOCATION"
100 OPEN "partDIM.DXF" FOR INPUT AS #1
110       I = 1 : J = 1
120       IF EOF(1) THEN CLOSE : GOTO 260
130       LINE INPUT #1,IN$(J)
140       IF IN$(J)="DIMENSION" THEN GOTO 160
150       J = J + 1
160       GOTO 110
170       LINE INPUT #1,DIMINDEX$(I)
180       IF DIMINDEX$(I) = " 2" THEN INPUT #1,DUM$(I),DUM$,
190       DUM$(I),DUM$(I),C12$(I) : GOTO 160
200       IF DIMINDEX$(I) = " 1" THEN INPUT #1,DIMEN$(I),DUM$,
210       DUM$(I),DUM$(I),C13$(I) ELSE IF DIMINDEX$(I) = " 0" THEN GOTO 190
220       ELSE GOTO 160
230       IF LEFT$(DIMEN$(I),3) = "%C" OR LEFT$(DIMEN$(I),3)="%C"
240       THEN GOSUB 1400 ELSE GOSUB 1500
250       PRINT USING "###.##    #.##    ;I,D(I),T(I);"
260       PRINT TAB(30);STATE$(I);
270       PRINT USING " #.##    ;X1(I),Y1(I),X2(I),Y2(I);"
280       PRINT TAB(67);OSS$(I)
290       I = I + 1
300  GOTO 110
310  EN = I - 1
320  GOSUB 340
330  GOSUB 600
340  GOSUB 780
350  GOSUB 2070
360  GOSUB 1600
370  GOSUB 950
380  END
390  SUBROUTINE FOR VERIFICATION OF MISSING TOLERANCES IN THE
400  PART DRAWING
410  I = 0
420  FOR  M = 1 TO EN
430      IF STATE$(M) = "V" THEN 580
440      NEWI = I + EN
450      FOR  N = 1 TO NEWI
460         IF M >= N THEN 570
470         IF STATE$(M) = "V" THEN 570
480         IF X1(M) = X1(N) THEN GOSUB 1300 : GOTO

380 ELSE 470
430 IF D(M) > D(N) THEN 440 ELSE 460
440 NEWX1(I) = X2(N) : NEWX2(I) = X2(M) : NEWY1(I) = Y2(N) :
NEWY2(I) = Y2(M)
450 GOTO 570
460 NEWX1(I) = X2(M) : NEWX2(I) = X2(N) : NEWY1(I) = Y2(M) :
NEWY2(I) = Y2(N)
470 IF X2(M) = X2(N) THEN GOSUB 1300 : GOTO
480 ELSE 520
480 IF D(M) > D(N) THEN GOTO 490 ELSE
GOTO 510
490 NEWX1(I) = X1(M) : NEWX2(I) = X1(N) : NEWY1(I) = Y1(N) :
NEWY2(I) = Y1(N)
500 GOTO 510
510 NEWX1(I) = X1(N) : NEWX2(I) = X1(M) : NEWY1(I) = Y1(M) :
NEWY2(I) = Y1(M)
520 IF X1(M) = X2(N) THEN GOSUB 1350 : GOTO
530 ELSE GOTO 550
530 NEWX1(I) = X1(N) : NEWX2(I) = X2(M) : NEWY1(I) = Y1(N) +
540 GOTO 570
550 IF X2(M) = X1(N) THEN GOSUB 1350 : GOTO
560 ELSE GOTO 570
560 NEWX1(I) = X1(M) : NEWX2(I) = X2(N) : NEWY1(I) = Y1(M) +
570 NEXT N
580 NEXT M
590 RETURN
600 'SUBROUTINE FOR APPENDING NEW ENTITIES IN THE DATA BASE
610 PRINT "" PRINT "NEW"
630 PRINT "ENTITY DIMENSION TOLERANCE STATE X1 Y1 X2 Y2 LOCATION"
640 FOR J = 1 TO I
650 IF NEWX1(J) = NEWX2(J) THEN STATE$(J+EN)="V" ELSE
STATE$(J+EN)="H"
660 IF STATE$(J+EN) = "V" THEN 760
670 D(J+EN) = NEWD(J) : T(J+EN) = NEWT(J)
680 X1(J+EN) = NEWX1(J) ; X2(J+EN) = NEWX2(J)
690 Y1(I+EN) = NEWY1(J) ; Y2(I+EN) = NEWY2(I)
700 OSS(J+EN) = "BD"
710 XM(J+EN) = .5 * (NEWX1(J) + NEWX2(J)) : YM(J+EN) =
NEWY1(J)
715 NEWXM(J) = XM(J+EN) : NEWYM(J) = YM(J+EN)
720 PRINT USING "####.##" ; J+EN,NEWD(J),NEWT(J);
730 PRINT TAB(30);STATE$(J+EN);
740 IF D(M) > D(N) THEN GOTO 490 ELSE
750 PRINT USING "####.##" ; NEWX1(J),NEWY1(J),NEWX2(J),NEWY2(J);
760 PRINT TAB(67);OSS$(J+EN)
770 NEXT J
770 RETURN
780 'SUBROUTINE FOR SEARCHING RELATIVE ENTITY FOR EACH DIAMETER OF
790 'THE ROTATIONAL PART.
800 FOR K = 1 TO NEWI
810 IF STATE$(K) = "H" THEN 930
820 SDIFF = 100
830 FOR L = 1 TO NEWI
840 IF STATE$(L) = "V" THEN 920
850 IF X3(K) > X1(L) AND X3(K) < X2(L) THEN 920 ELSE IF X3(K) < X1(L) AND X3(K) > X2(L) THEN 920 ELSE
860 IF OSS(K) = "LD" THEN DIFF1(L) = X1(L) - X3(K) ELSE DIFF2(L) = X2(L) - X3(K) ELSE DIFF1(L) = X3(K) - X1(L) ELSE DIFF2(L) = X3(K) - X2(L)
870 IF DIFF1(L) <= 0 THEN DIFF1(L) = 100
880 IF DIFF2(L) <= 0 THEN DIFF2(L) = 100
890 IF DIFF1(L) > DIFF2(L) THEN DIFF(L) = DIFF2(L) ELSE DIFF(L) = DIFF1(L)
900 IF SDIFF < DIFF(L) THEN GOTO 920 ELSE SDIFF = DIFF(L)
910 RELD(K) = D(L)
920 NEXT L
930 NEXT K
940 RETURN
950 'SUBROUTINE FOR OUTPUT FILE
960 OPEN "TEST.DAT" FOR OUTPUT AS #2
970 WRITE #2, DEC
980 FOR J = 1 TO I
990 WRITE #2, J
1000 WRITE #2, NEWD(J)
1010 WRITE #2, NEWT(J)
1020 WRITE #2, NEWX1(J)
1030 WRITE #2, NEWY1(J)
1040 WRITE #2, NEWX2(J)
1050 WRITE #2, NEWY2(J)
1060 WRITE #2, NEWXM(J)
1070 WRITE #2, NEWYM(J)
1080 NEXT J
1090 WRITE #2, NEW
1100 FOR J = 1 TO NEWI
1110 WRITE #2, J
1120 WRITE #2, D(J)
1130 WRITE #2, T(J)
1140 WRITE #2, STATE(J)
1150 WRITE #2, X1(J)
1160 WRITE #2, Y1(J)
1170 WRITE #2, X2(J)
1180 WRITE #2, Y2(J)
1190 WRITE #2, XM(J)
1200 WRITE #2, YM(J)
1210 WRITE #2, YRELATIVE(J)
1220 WRITE #2, RELD(J)
1230 WRITE #2, ST(J)
1240 WRITE #2, CI(J)
1250 WRITE #2, CY(J)
1255 WRITE #2, CO(J)
1260 NEXT J
1280 CLOSE #2
1290 RETURN
1300 'CALCULATE THE DIFFERENCE FOR NEW DIMENSION AND
TOLERANCE
1310 I = I + 1
1320 NEWD(I) = ABS(D(M) - D(N))
1330 NEWT(I) = ABS(T(M) - T(N))
1340 RETURN 'CALCULATE THE SUM FOR NEW DIMENSION AND TOLERANCE
1360 I = I + 1
1370 NEWD(I) = ABS(D(M) + D(N))
1380 NEWT(I) = ABS(T(M) + T(N))
1390 RETURN
1400 'SUBROUTINE FOR RETRIEVING DATA OF DIAMETERS FOR ROTATIONAL PART
1410 STATES(I) = "V" : STATE(I) = 0
1420 D(I) = VAL(MIDS(DIMEN$(I),4,DES))
1430 T(I) = VAL(RIGHTS(DIMEN$(I),DES))
1440 X1(I) = VAL(C10S$(I)) : Y1(I) = VAL(C20S$(I))
1450 X2(I) = VAL(C10S$(I)) : Y2(I) = VAL(C23S$(I))
1460 X3(I) = VAL(C13S$(I))
1470 XM(I) = X1(I) : YM(I) = .5 * (Y1(I) + Y2(I))
1480 IF X3(I) > X1(I) THEN OS$(I) = "LD" ELSE OS$(I) = "RD"
1490 RETURN 'SUBROUTINE FOR RETRIEVING DATA OF LENGTHS FOR ROTATIONAL PART
1510 STATE$(I) = "H" : STATE(I) = 1
1520 D(I) = VAL(LEFT$(DIMEN$(I),DES))
1530 T(I) = VAL(RIGHT$(DIMEN$(I),DES))
1540 X1(I) = VAL(C13S$(I)) : Y1(I) = VAL(C20S$(I))
1550 X2(I) = VAL(C10S$(I)) : Y2(I) = VAL(C20S$(I))
1560 X3(I) = VAL(C23S$(I))
1570 XM(I) = .5 * (X1(I) + X2(I)) : YM(I) = Y1(I)
1580 IF Y3(I) > Y1(I) THEN OS$(I) = "TD" ELSE OS$(I) = "BD"
1590 RETURN 'SUBROUTINE FOR VERTIFICATION OF GEOMETRIC TOLERANCE
1610 FOR J = 1 TO NEWI
1620 IF STATE$(J) = "H" THEN 1740
1630 RATIO = RELD(J)/D(J)
1640 CLS: LOCATE 3,15
1650 PRINT "ENTITY DIAMETER SIZE TOLERANCE"
1660 LOCATE 4,18: PRINT USING "##.##";J,D(J),T(J)
1670 LOCATE 6,15: PRINT "LENGTH OF THE CIRCULAR SURFACE = "
1680 LOCATE 6,51: PRINT USING "##.##";RELD(J)
1690 LOCATE 6,57: PRINT "INCH"
1700 GOSUB 2300: IF ANS$ = "N" OR ANS$ = "n" THEN GOTO 1735
1710 GOSUB 1790
1720 GOSUB 1920
1730 GOSUB 2020
SUBROUTINE FOR STRAIGHTNESS
1800 IF RATIO > 3 THEN ST(J) = 1 ELSE ST(J) = 0
1810 IF ST(J) = 0 THEN 1910
1820 MAXST(J) = 2 * T(J) + (MHS(J) - (D(J) + T(J)))
1830 IF MAXST(J) > 0 THEN 1870
1840 LOCATE 9,15 : PRINT "DIAMETER OF THE HOLE IS TOO SMALL, TRY A LARGER ONE!"
1850 GOSUB 2070 : LOCATE 23,1 : PRINT "" : LOCATE 9,1 : PRINT ""
1880 LOCATE 8,15 : PRINT "MAXIMUM STRAIGHTNESS TOLERANCE REQUIRED IS ";
1885 PRINT USING "##.##" ; MAXST(J)
1890 LOCATE 9,15 : INPUT "Do you want to change the straightness tolerance (y/n) "; ANS$
1900 IF ANS$ = "Y" OR ANS$ = "y" THEN LOCATE 10,15 : INPUT "NEW STRAIGHTNESS TOLERANCE = "; ST(J) ELSE ST(J) = MAXST(J)
1910 RETURN

SUBROUTINE FOR CIRCULARITY
1920 LOCATE 12,15 : INPUT "Should this shaft fit in a hole (y/n) "; ANS$
1940 IF ANS$ = "Y" OR ANS$ = "y" THEN MAXCI(J) = T(J) :
GOTO 1950 ELSE CI(J) = 0 : GOSUB 1970 : GOTO 1960
1950 LOCATE 13,15 : PRINT "MAXIMUM ROUNDNESS TOLERANCE REQUIRED IS ";
1951 PRINT USING "##.##" ; MAXCI(J)
1952 LOCATE 14,15 : INPUT "Do you want to change the circularity (roundness) tolerance (y/n) "; ANS$
1953 IF ANS$ = "Y" OR ANS$ = "y" THEN LOCATE 15,15 : INPUT "NEW ROUNDNESS TOLERANCE = "; CI(J) ELSE CI(J) = MAXCI(J)
1960 RETURN

SUBROUTINE FOR CYLINDRICITY
1970 LOCATE 16,15 : INPUT "Does this segment need to be rotated in the hole (y/n) "; ANS$
1990 IF ANS$ = "Y" OR ANS$ = "y" THEN MAXCY(J) = .5 * (MHS(J) - (D(J) + T(J))) : GOTO 1995 ELSE CY(J) = 0 : GOTO 2010
1995 IF MAXCY(J) <= 0 THEN GOTO 1840
2000 LOCATE 17,15 : PRINT "MAXIMUM CYLINDRICITY TOLERANCE REQUIRED IS ";
2001 PRINT USING "##.##" ; MAXCY(J)
2002 LOCATE 18,15 : INPUT "Do you want to change the cylindricity tolerance (y/n) "; ANS$
2003 IF ANS$ = "Y" OR ANS$ = "y" THEN LOCATE 19,15 : INPUT "NEW CYLINDRICITY TOLERANCE = "; CY(J) ELSE CY(J) = MAXCY(J)
2010 RETURN

SUBROUTINE FOR CONCENTRICITY
2020 IF RATIO < 3 THEN CO(J) = 0 : GOTO 2060
2030 LOCATE 20,15 : INPUT "Should it be accurately rotated
about the centre line (y/n) ";ANS$
2040     IF ANS$="Y" OR ANS$="y" THEN MAXCO(J) = MHS(J) -
         (D(J)+T(J)) : GOTO 2045 ELSE CO(J) = 0 : GOTO 2060
2045     IF MAXCO(J) <= 0 THEN GOTO 1840
2050     LOCATE 21,15 : PRINT "MAXIMUM CONCENTRICITY TOLERANCE
         REQUIRED IS ";
2051     PRINT USING "##.##";MAXCO(J)
2052     LOCATE 22,15 : INPUT "Do you want to change the
         concentricity tolerance (y/n) ";ANS$
2053     IF ANS$ = "Y" OR ANS$ = "y" THEN LOCATE 23,15 : INPUT
         "NEW CONCENTRICITY TOLERANCE = ";CO(J) ELSE CO(J) = MAXCO(J)
2060     RETURN
2070     'SUBROUTINE TO PAUSE THE PROGRAM
2080     LOCATE 24,20
2090     PRINT "<< PRESS ANY KEY TO CONTINUE >>"
2100     A$ = INKEY$
2110     IF A$ = "" THEN 2100
2120     RETURN
2130     'SUBROUTINE FOR GENERAL REPORT OF GEOMETRIC TOLERANCES
2140     CLS : LOCATE 2,55 : PRINT "TYPE OF"
2150     LOCATE 3,10
2160     PRINT "ENTITY DIAMETER SIZE TOLERANCE GEOMETRIC
         TOLERANCE"
2170     FOR J = 1 TO NEWI
2180     IF STATE$(J) = "H" THEN 2280
2190     IF ST(J) <> 0 THEN ST$(J) =
         "STRAIGHTNESS"
2200     IF CI(J) <> 0 THEN CI$(J) = "CIRCULARITY"
2210     IF CY(J) <> 0 THEN CY$(J) =
         "CYLINDRICITY"
2215     IF CO(J) <> 0 THEN CO$(J) =
         "CONCENTRICITY"
2220     PRINT "-------------------------------------------------
         -----------"
2230     PRINT TAB(12);
2240     PRINT USING "##.##";J,D(J),T(J);
2240     PRINT USING "##.##";J,D(J),T(J);
2250     PRINT TAB(51);ST$(J)
2260     PRINT TAB(51);CI$(J)
2270     PRINT TAB(51);CY$(J)
2280     PRINT TAB(51);CO$(J)
2280     NEXT J
2290     RETURN
2300     'SUBROUTINE FOR RETRIEVING INFORMATION OF ASSEMBLY
2310     LOCATE 7,15 : INPUT "Will this surface assembly with
         another part (y/n) ";ANS$
2320     IF ANS$ = "N" OR ANS$ = "n" THEN GOTO 2340
2330     LOCATE 8,15 : INPUT "What is the minimum allowable
         diameter (INCH) of the hole ";MHS(J)
2340     RETURN
2415     PRINT TAB(51);CO$(J)
APPENDIX B

LISTING OF "CAGT. LSP" PROGRAM
(defun dxf (code elist)
    (cdr (assoc code elist))
)

(defun getnth (n c)
    (nth n (entget c))
)

(defun getnext (c)
    (cond
        ((= (dxf 0 (entget c)) "SEQEND") nil)
        (t (append (list (cdr (getnth : c)))
            (getnext (entnext c))))
    )
)

(defun mainget (a)
    (setq c (entnext (dxf -1 a)))
    (getnext c)
)

(defun line_cal (v1 v2 choic)
    (setq atry (getangle (v1 choic)))
    (setq aact (getangle (v1 v2)))
    (cond ((= atry aact) (list 'line v1 v2))
        (t nil)
    )
)

(defun apoly_select (lst dlpt)
    (cond ((= lst nil) nil)
        (t (aline (car lst) dlpt)
            (apoly_select (cdr lst) dlpt)
        )
    )
)

(defun apolyline (lst)
    (setq dlpt 0)
    (apoly_select (cdr lst) dlpt)
)

(defun setv2 (lst first)
    (cond
        ((= (cdr lst) nil) (dxf 10 first))
        (t (dxf 10 (cadr lst)))
    )
)

(defun poly_line (lst first choic)
    (setq v1 (dxf 10 (car lst)))
    (setq v2 (setv2 lst first))
(line_cal v1 v2 choic)
)

(defun poly_arc (lst first)
  ;)

(defun line_or_arc(lst first choic)
  (cond
    ((= (dfx 42 (car lst)) 0) (poly_line lst first choic))
    (t nil)
  )

(defun getinfo(lst first flag choic)
  (cond ((= lst nil) nil)
        ((and (= (cdr lst) nil) (= flag 0)) nil)
        (t (append (list (line_or_arc lst first choic))
                   (getinfo (cdr lst) first flag choic)))))

(defun choice(lst first flag)
  (setq anys
    (getstring "\nAny segment needed to be measured (y/n) ?")
  )
  (cond ( (= anys "y")
    (setq choic
      (getpoint "\nSelect a segment to be measured ?")
    )
    (append (list (getinfo lst first flag choic))
            (choice lst first flag)
    )
    )
  )
  (t nil)
)

(defun xmax(lst xm)
  (cond ((= (car lst) (last lst))
         (list xm lst)
    )
    (t
      (setq x1 (car (car lst)))
      (setq x2 (car (cdr lst)))
      (if (< xm (max x1 x2))
        (setq xm (max x1 x2))
      )
      (xmax (cdr lst) xm)
    )
)

(defun xmin(lst xm)
  (cond ((= (car lst) (last lst))
         (list xm lst)
    )
    (t
      (setq x1 (car (car lst)))
      (setq txm (min x1 xm n))
      (setq xm txm)
    )
)

(defun ymax (lst ym)
  (cond ((= (car lst) (last lst))
         (list ym lst)
       )
        (t (setq y1 (cadr (car lst)))
           (setq y2 (cadr (cadr lst)))
           (if (< ym (max y1 y2))
               (setq ym (max y1 y2))
           )
        )
    )
    (ymax (cdr lst) ym)
  )
)

(defun ymin (lst ymn)
  (cond ((= (car lst) (last lst))
         (list ymn lst)
       )
        (t (setq y1 (cadr (car lst)))
           (setq y2 (cadr (cadr lst)))
           (if (> ymn (min y1 y2))
               (setq ymn (min y1 y2))
           )
        )
    )
    (ymin (cdr lst) ymn)
  )
)

(defun sortxm (lst xxm) ; sort points on the max x-axis
  (cond ((= (car lst) (last lst)) sortlst)
        (t (if (= xxm (car (car lst)))
            (setq sortlst (append (list (car lst))
                                   (sortxm (cdr lst) xxm)
                                 ))
            (sortxm (cdr lst) xxm)
        )
  )
)

(defun sortxn (lst xxn) ; sort points on the min x-axis
  (cond ((= (car lst) (last lst)) sortlst)
        (t (if (= xxn (car (car lst)))
            (setq sortlst (append (list (car lst))
                                   (sortxn (cdr lst) xxn)
                                 ))
            (sortxn (cdr lst) xxn)
        )
  )
)
(defun sortyn(lst yyn) ; sort points on the min y-axis
  (cond ((eq (car lst) (last lst)) sortlst)
    (t (if (= yyn (cadr (car lst)))
       (setq sortlst (append (list (car lst))
                             (sortyn (cdr lst) yyn))
       (sortyn (cdr lst) yyn))
  )
)
)

(defun polylinef(code)
  (prompt "\n RECOGNIZING THE FEATURE ... ")
  (setq flag (dxf 70 code))
  (setq lst (map 'get code))
  (setq first (car lst))
  (setq xmn 0) (setq ymn 0)
  (setq xmn 30) (setq ymn 30)
  (setq xxm (car (xmax lst xmn)))
  (setq yym (car (ymax lst ymn)))
  (setq xxn (car (xmin lst xxm)))
  (setq yyn (car (ymin lst yyn)))
  (setq sortlst nil)
  (setq l1p (list xxn (car (ymin(sortxn lst xxn) 30))))
  (setq sortlst nil)
  (setq l1p (list xxm (car (ymin(sortxm lst xxm) 30))))
  (setq sortlst nil)
  (setq u1p (list xxn (car (ymin(sortxn lst xxn) 0 ))))
  (setq sortlst nil)
  (setq u1p (list xxm (car (ymin(sortxm lst xxm) 0 ))))
  (setq sortlst nil)
  (setq u1p (list (car (xmin(sortyn lst yyn) 30))) yyn))
  (list 'polyline (list l1p u1p) (list lrp urp)
    (list lmp lrp) (list (list xxn yyn) lmp))
)

(defun circlef(a)
  (prompt "\n This is a circle, RECOGNIZING .")
  (setq rad (dxf 40 code))
  (setq cp (dxf 10 code))
  (list 'circle rad cp a)
)

(defun acircle(lst)
  (setq rad (nth 1 lst))
  (setq cp (nth 2 lst))
  (setq a (nth 3 lst))
(command "diam" a ")

(defun arcf()
  (prompt "\n This is an arc. ")
  (setq sang (dxf 50 code))
  (setq eang (dxf 51 code))
  (setq acp (dxf 10 code))
  (setq arad (dxf 40 code))
  (list 'arc sang eang acp arad)
)

(defun aarc(lst)
  (setq sang (nth 1 lst))
  (setq eang (nth 2 lst))
  (setq acp (nth 3 lst))
  (setq arad (nth 4 lst))
)

(defun linef()
  (prompt "\n This is a line. ")
  (setq stp (dxf 10 code))
  (setq endp (dxf 11 code))
  (setq X1 (car stp))
  (setq X2 (car endp))
  (setq Y1 (cdr stp))
  (setq Y2 (cdr endp))
  (setq dist (Sqrt (+ (* (- X1 X2) (- X1 X2)) (* (-
    Y1 Y2) (- Y1 Y2)))))
)

(defun box(sym value datum loc)
  (command "units" "2" "4" "" "" "")
  (setq ul loc)
  (setq ur (list (+ 2 (car loc)) (cadr loc)))
  (setq lr (list (car ur) (- (cadr ur) 0.5)))
  (setq ll (list (- (car lr) 2) (cadr lr)))
  (setq um1 (list (+ 0.5 (car loc)) (cadr loc)))
  (setq um2 (list (+ 1.0 (cadr loc)) (cadr loc)))
  (setq lm1 (list (car um1) (- (cadr um1) 0.5)))
  (setq lm2 (list (car um2) (- (cadr um2) 0.5)))
  (command "pline" ul ur ll "c")
  (command "line" um1 lm1 ")
  (command "line" um2 lm2 ")
  (setq cp (list (+ 0.25 (car ul)) (- (cadr ul) 0.25)))
  (cond ((= sym "c")
    (command "circle" cp 0.2)
    (setq cpta (list (car cp) (- (cadr cp) 0.225)))
    (command "text" "c" cpta 0.45 "" "))
  )
  ((= sym "s")
    (setq pl1 (list (+ 0.05 (car ul)) (- (cadr ul) 0.25)))
    (setq pl2 (list (+ 0.4 (car pl1)) (cadr pl1)))
    (command "line" pl1 pl2 "")
  )
(= sym "conc")
(command "circle" cp 0.2)
(command "circle" cp 0.1)

(= sym "p")
(setq pl (list (+ 0.25 (car ul)) (- (cadr ul) 0.05)))
(setq p2 (list (car pl) (- (cadr pl) 0.4)))
(setq p3 (list (- (car p2) 0.2) (cadr p2)))
(setq p4 (list (+ (car p3) 0.4) (cadr p3)))
(command "line" p1 p2 "")
(command "line" p3 p4 "")

(setq cpa (list (+ 1.75 (car ul)) (- (cadr ul) 0.25)))
(if (= sym "c") (command "circle" cpa 0.2))
(setq ccppa (list (car cpa) (- (cadr cpa) 0.10)))
(command "text" "c" ccppa 0.2 "" datum)
(setq tx (list (+ 1 (car ul)) (- (cadr ul) 0.25)))
(command "units" "2" "2" "" "" "" "")
(setq txp (list (car tx) (- (cadr tx) 0.15)))
(command "text" "c" txp 0.18 "" value)
(redraw)

(defun a-line (lst dlpt)
  (command "units" "2" "2" "" "" "")
  (command "dim")
  (setq vl (nth 0 lst))
  (setq v2 (nth 1 lst))
  (setq dist (distance vl v2))
    (setq aaa (angle vl v2))
    (setq ang1 (+ aaa 1.570796))
  (if (= (car vl) xxm) (setq sign -1) (setq sign 1))
  (if (= (cadr vl) yym) (setq sign -1) (setq sign 1))
  (setq extpt (* sign 0.2))
  (setq pt1 (polar vl ang1 extpt))
  (setq pt2 (polar v2 ang1 extpt))
    (setq dis (* sign 0.7))
  (setq dlpt (polar vl ang1 dis))
  (cond ((= (car vl) (car v2))
    (command "vertical" pt1 pt2 dlpt "")
  )
    ((= (cadr vl) (cadr v2))
    (command "horizontal" pt1 pt2 dlpt "")
  )
    (t (command "aligned" pt1 pt2 dlpt "")
  )
  (setq asktol (getstring "$ Do you need
tolerancing ? (y/n)" ))
  (cond ((= asktol "y")
    (setq tolp (getreal "$ Enter the +ve tolerance ? ")
    (setq tolh (getreal "$ Enter the -ve tolerance ? ")
    (command "dimtol" "on"
(command "dimtp" tolp)
(command "dimtm" toln)
(setq toltype (getstring "\n Limit Dimensioning(1)
or +/- tolerancing(? ?))")
(if (= toltype "l") (command "dimlim" "on")
(command "dimlim" "off"))
(command "exit") (command "erase" "l" "")
(aline lst dlpt)
)
(t (command "dimtol" "off")
(command "dimlim" "off")
(command "exit")
)
)
(cond ((and (= (car v1) (car v2)) (= (car v1) xxm))
(setq sym "c")
(setq loc (polar pt2 0.75 1))
(command "line" pt2 loc "))
(if (or (< tolp 0.3) (< toln 0.3)) (setq value "0.015") (setq value "0.025")
(setq datum "M")
(box sym value datum loc)
)
((= v1 lmp)
(setq sym "conc")
(setq loc (polar pt2 -0.7 1))
(command "line" pt2 loc ")
(if (or (< tolp 0.4) (< toln 0.4)) (setq value "0.005") (setq value "0.01")
(setq datum 'A")
(box sym value datum loc)
)
((and (= (car v1) (car v2)) (= (car v1) xxn))
(setq sym "s")
(setq loc (polar pt1 4 1))
(command "line" pt1 loc ")
(if (or (< tolp 0.3) (< toln 0.3)) (setq value "0.015") (setq value "0.025")
(setq datum ""
(box sym value datum loc)
)
)
((= v2 lmp)
(setq sym "p")
(setq loc (polar pt2 4 1))
(command "line" pt2 loc ")
(setq dpt1 (list (car lmp) (+ (cadr urp) 0.2)))
(setq dpt2 (list (car dpt1) (+ (cadr dpt1) 1)))
(command "line" dpt1 dpt2 ")
(setq p1 (list (car dpt1) (+ (cadr dpt1) 0.3)))
(setq p2 (list (car dpt1) (+ (cadr p1) 0.4)))
(setq p3 (list (+ (car dpt1) 0.3) (+ (cadr p1) 0.2)))
(command "line" p1 p3 p2 ")
(command "line" p3 (polar p3 0 0.2) ")
(setq b1 (list (+ (car p1) 0.5) (cadr p1)))
(setq b2 (list (+ (car b1) 0.4) (cadr b1)))
(setq b3 (list (car b2) (+ (cadr b2) 0.4)))
(setq b4 (list (- (car b3) 0.4) (cadr b3)))
(command "line" b1 b2 b3 b4 "c")
(setq b5 (list (+ (car b1) 0.2) (+ (cadr b1) 0.05)))
(command "text" "c" b5 0.3 "" "A")
(if (or (< tolp 0.4) (< toln 0.4)) (setq value "0.002") (setq value "0.005"))
(setq datum "A")
(box sym value datum loc)
)
(command "exit")
(command "units" "2" "4" "" "" "")

(defun select()
  (setq c "CIRCLE") ; set the variable
  (setq d "POLYLINE")
  (setq arc "ARC")
  (setq line "LINE")
  (setq a (entsel)) ; give the name to entity
  (setq code (entget (car a))) ; the type of data
  (cond
    ((= c e) (circlef a))
    ((= d e) (polylinet code))
      ((= arc e) (arcf ))
      ((= line e) (linef ))
    (t (prompt "\n No objects is selected") nil)
  ))

(defun query()
  (setq ans
    (getstring "\n Do you want to select an object? (y/n) ")
  (cond ((= ans "y") (append (list (select)) (query)))
    ((= ans "n") nil))
)

(defun doselect(lst)
  (cond ((= (car lst) 'circle) (acircle lst))
    ((= (car lst) 'arc) (aarc lst))
    ((= (car lst) 'line) (aline lst))
    ((= (car lst) 'polyline) (apolyline lst) )
  )
)

(defun do(lst)
  (cond ((= (car lst) nil) nil)
    (t (doselect (car lst)) (do (cdr lst))))
)

(defun c:TOL()
  (prompt "\n << Please Don't Use BLOCK LETTER. >>") )
(setq lst (query))
(do lst)
(redraw)
(PROMPT "\n  << AUTOMATIC DIMENSIONING AND TOLERANCING IS FINISHED >>
")
}
APPENDIX C

LISTING OF PROCESS.BAS, JOIN-T&S.BAS, FINISH.BAS PROGRAMS
'PROCESS.BAS program selects machining processes based on dimensional tolerances and surface roughness of the machining surfaces. 'Using output file TPROC.DAT and RPROC.DAT to store selected processes 'for JOIN-T&S.bas program to read.

CLS
INPUT "ENTER THE DIMENSIONAL TOLERANCE (+/- in) ? ", T
INPUT "ENTER THE SURFACE ROUGHNESS (f in) ? ", R
N = 0
OPEN "A:PROCESS.DAT" FOR INPUT AS #1
TMAXV = 0 : RMAXV = 0
IF EOF(1) THEN CLOSE : GOTO 180
N = N + 1
INPUT #1, PROCS(N), TMIN(N), TMAX(N), RMIN(N), RMAX(N)
IF T <= TMAX(N) AND T >= TMIN(N) THEN GOSUB 480
IF R <= RMAX(N) AND R >= RMIN(N) THEN GOSUB 510
GOTO 120
180 PRINT
PRINT "PROCESS BASED ON DIMENSIONAL TOLERANCE"
PRINT " PROCESS PROBABILITY"
PRINT "TPROC.DAT" FOR OUTPUT AS #2
FOR M = 1 TO N
    IF TMAXV = 0 OR TVALUE(M) = 0 THEN 310
    PR(M) = (100 - 40 * (TVALUE(M)/TMAXV))/100
    WRITE #2, PROCS(M), PR(M)
    PRINT PROCS(M); TAB(20);
    PRINT USING "##"; PR(M)*10;
    PRINT TAB(25);"/ 10"
NEXT M
CLOSE #2
PRINT : PRINT
PRINT "PROCESS BASED ON SURFACE ROUGHNESS"
PRINT " RPROC.DAT" FOR OUTPUT AS #3
FOR M = 1 TO N
    IF RMAXV = 0 OR RVALUE(M) = 0 THEN 450
    PR(M) = (100 - 40 * (RVALUE(M)/RMAXV))/100
    WRITE #3, PROCS(M), PR(M)
    PRINT PROCS(M); TAB(20);
    PRINT USING "##"; PR(M)*10;
    PRINT TAB(25);"/ 10"
NEXT M
CLOSE #3
END
TVALUE(N) = T - TMIN(N)
IF TVALUE(N) > TMAXV THEN TMAXV = TVALUE(N)
RETURN
RVALUE(N) = R - RMIN(N)
IF RVALUE(N) > RMAXV THEN RMAXV = RVALUE(N)
RETURN

'JOIN-T&SBAS program combines the selected processes which are based
'on dimensional tolerance and surface finish.
'Program also reads the selected processes which are based on geometric
tolerances from the knowledge base rule through
EXPERT4.DAT file.
'ASSUMING NUMBER OF SELECTED PROCESSES IS LESS THAN 10
OPEN "TPROC.DAT" FOR INPUT AS #1
N = 1
IF EOF (1) THEN CLOSE #1 : GOTO 120
INPUT #1,TP$(N),TPR(N)
N = N + 1
GOTO 80
OPEN "RPROC.DAT" FOR INPUT AS #2
M = 1
IF EOF (2) THEN CLOSE #2 : GOTO 170
M = M + 1
GOTO 140
K = 0
FOR I = 1 TO N-1
FOR J = 1 TO M-1
IF TP$(I) <> RP$(I) THEN 240
K = K + 1
CP$(K) = TP$(I)
CPR(K) = .5 * (TP$(I) + RPR$(J))
240 NEXT J
NEXT I
OPEN "EXPERT4.OUT" FPR INPUT AS #3
L = 1
IF EOF(3) THEN CLOSE #3 : GOTO 370
INPUT #3,GTP$(L),GTPR(L)
FOR JJ = 1 TO K
IF CP$(JJ) <> GTP$(L) THEN 340
CPR(JJ) = CPR(JJ) + .5 * GTPR(L)
330 IF CPR(JJ) > 1 THEN CPR(JJ) = 1
340 NEXT JJ
L = L + 1
GOTO 280
OPEN "PROCESS.OUT" FOR OUTPUT AS #4
FOR M = 1 TO K
WRITE #4,CP$(M),CPR(M)
NEXT M
CLOSE
END
APPENDIX D

LISTING OF LATHE.BAS AND MCCENTER.BAS PROGRAMS
'LATHE.BAS program selects machines from LATHE.DAT file for processes.
Selection is based on size of the part and its dimensional tolerances.
Selected machines are listed in the OPERATE.DAT file.

DIM CODE$(20), MNAMES$(20), MTD(20), MTL(20), MAXSP(20),
MINSP(20), F(20), XACC(20), YACC(20), MP(20), DIFF(20), PROB(20)

INPUT "ENTER THE SELECTED PROCESS FROM EXPERT SYSTEM :
", P$

INPUT "ENTER THE CORRESPONDING PROBABILITY : ", PR

INPUT "ENTER THE SURFACE NUMBER : ", SN

INPUT "ENTER THE DIMENSIONAL TOLERANCE OF SURFACE : ", DT

INPUT "ENTER THE DIAMETER OF RAW MATERIAL : ", RDIA

INPUT "ENTER THE LENGTH OF RAW MATERIAL : ", RLEN

OPEN "A: LATHE.DAT" FOR INPUT AS \\#1

IF P$ = "DRILLING" OR P$ = "drilling" THEN GOTO 140

IF P$ <> "REAMING" OR P$ <> "reaming" THEN GOTO 150

INPUT "ENTER THE SIZE OF THE HOLE : ", DIMEN

MAXDFF = 0 : I = 1

IF EOF(1) THEN CLOSE : GOTO 230

INPUT \\#1, CODE$(I), MNAMES$(I), MTD(I), MTL(I), MAXSP(I),
MINSP(I), F(I), XACC(I), YACC(I), MP(I)

IF MTD(I) < RDIA OR MTL(I) < RLEN THEN GOTO 220

IF DT < XACC(I) AND DT < YACC(I) THEN GOTO 220

DIFF(I) = ((DT - XACC(I)) + (DT - YACC(I)))/2

IF DIFF(I) > MAXDFF THEN MAXDFF = DIFF(I)

I = I + 1 : GOTO 160

N = I - 1

OPEN "A: OPERATE.DAT" FOR APPEND AS \\#2

PRINT

PRINT "SURFACE MACHINE PROCESS PROCESS MACHINE MACHINE"

PRINT " NO. CHOICE PROB. CODE"

PRINT "PROB."

FOR M = 1 TO N

IF DIFF(M) = 0 OK MAXDFF = 0 THEN 390

J = J + 1

PROB(J) = (10 - 4 * (DIFF(M)/MAXDFF))/10

WHITE

PRINT USING " #  " ; SN; J;

PRINT TAB(20); P$; TAB(35);

PRINT USING " #.## " ; PR;

PRINT TAB(45); CODE$(M); TAB(55);

PRINT USING " #.## " ; PROB(J)

NEXT M

IF J = 0 THEN PRINT : PRINT "WARNING ! There is no suitable machine. You can add more machine on the LATHE data file and run this program again. OR you can try other alternative process."

END
'MCCENTER.BAS program selects machine from MCCENTER.DAT data file. 'Selection is based on size of the part and its dimensional tolerances.

'Selected machines are listed in the OPERATE.DAT file.

DIM CODE$(20), MNAME$(20), MAXX(20), MAXY(20), MAXZ(20),
MAXSP(20), MINS(20), F(20), ACC(20), POWER(20), DIFF(20), PROB(20)

INPUT "ENTER THE SELECTED PROCESS FROM EXPERT SYSTEM :
", PS

INPUT "ENTER THE CORRESPONDING PROBABILITY : ", PR

INPUT "ENTER THE SURFACE NUMBER : ", SN

INPUT "ENTER THE DIMENSIONAL TOLERANCE OF SURFACE : ", DT

INPUT "ENTER THE HEIGHT OF RAW MATERIAL : ", H

INPUT "ENTER THE WIDTH OF RAW MATERIAL : ", W

INPUT "ENTER THE LENGTH OF RAW MATERIAL : ", L

IF PS = "DRILLING" OR PS = "drilling" THEN GOTO 130

ELSE

INPUT "ENTER THE SIZE OF THE HOLE : "; DIMEN

OPEN "A:MCCENTER.DAT" FOR INPUT AS #1

MAXDFF = 0 : I = 1

IF EOF(1) THEN CLOSE : GOTO 250

INPUT #1, CODE$(I), MNAME$(I), MAXX(I), MAXY(I), MAXZ(I),
MAXSP(I), MINS(1), F(I), ACC(I), POWER(I)

IF MAXX(I) < W OR MAXX(I) < L THEN GOTO 240

IF MAXY(I) < W OR MAXY(I) < L THEN GOTO 240

IF MAXZ(I) < W OR MAXZ(I) < L THEN GOTO 240

IF DT < ACC(I) THEN GOTO 240

DIFF(I) = DT - ACC(I)

IF DIFF(I) > MAXDFF THEN MAXDFF = DIFF(I)

I = I + 1 : GOTO 160

N = I - 1

OPEN "A:OPERATE.DAT" FOR APPEND AS #2

PRINT

PRINT "SURFACE MACHINE PROCESS PROCESS MACHINE"

PRINT " NO. CHOICE PROB. CODE"

PRINT

FOR M = 1 TO N

IF DIFF(M) = 0 OR MAXDFF = 0 THEN 410

J = J + 1

PROB(J) = (10 - 4 * (DIFF(M) / MAXDFF)) / 10

WRITE #2, SN, DIMEN, DT, J, PS, PR, CODE$(M), PROB(J),

POWER(M), MINS(M), MAXSP(M), F(M)

PRINT USING " #    #    "; SN; J;

PRINT TAB(20); PS; TAB(35);

PRINT USING " #    "; PR;

PRINT TAB(45); CODE$(M); TAB(55);

PRINT USING " #    "; PROB(J)

NEXT M

IF J = 0 THEN PRINT : PRINT "WARNING! There is no suitable machine. You can add more machine on the LATHE data file and run this program again. OR you can try other alternative process."

430 END
APPENDIX E

LISTING OF TOOL.BAS PROGRAM
INPUT #3, TIDS(N), MACHS(N), TDIM(N), TOL(N), COST(N),
N = N + 1 : GOTO 420
NT = N - 1
GOSUB 860
RETURN

' WHEN "H", CHOOSE DRILLING TOOLS FROM
INPUT "ENTER THE TOOL CODE, [TCODE = TWIST or REAM] :" , TC$
IF TC$ = "TWIST" THEN GOSUB 710 : GOTO 550
IF TC$ <> "REAM" THEN PRINT " PLEASE RE-ENTER THE TOOL
CODE " : GOTO 490
OPEN "A:REAMER.DAT" FOR INPUT AS #5
JJ = 1

' "JJ" IS THE NUMBER OF DRILL SIZE RANGES
IF EOF(5) THEN CLOSE #5 : GOTO 650
INPUT #5, MINS(JJ), MAXS(JJ), TD(JJ)
IF DIMEN(J) > MAXS(JJ) OR DIMEN(J) < MINS(JJ) THEN GOTO 640
OPEN "A:EXPERT2.OUT" FOR INPUT AS #8
KK = 1
IF EOF(8) THEN CLOSE #8 : K(J) = KK - 1 :
GOTO 640

INPUT #8, TOOL$(J,KK), TCOST(J,KK)
TOOLTOL(J,KK) = TD(JJ)
KK = KK + 1 : GOTO 600
JJ = JJ + 1 : GOTO 550
RETURN

' WHEN "O", CHOOSE THREADING TOOLS
OPEN "A:EXPERT3.OUT" FOR INPUT AS #9
K(J) = 1
INPUT #9, TOOL$(I,1), TCOST(I,1)
RETURN

' SUBROUTINE TO SELECT A TWISTDRILLS AND CHECK ITS TOLERANCE
OPEN "A:TWIDRILL.DAT" FOR INPUT AS #4
JJ = 1

' "JJ" IS THE NUMBER OF DRILL SIZE RANGES
IF EOF(4) THEN CLOSE #4 : GOTO 850
INPUT #4, MINS(JJ), MAXS(JJ), TD(JJ)
IF DIMEN(J) > MAXS(JJ) OR DIMEN(J) < MINS(JJ) THEN 840
OPEN "A:EXPERT1.OUT" FOR INPUT AS #7
KK = 1
IF EOF(7) THEN CLOSE #7 : K(J) = KK - 1 :
GOTO 840

INPUT #7, TOOL$(J,KK), TCOST(J,KK)
TOOLTOL(J,KK) = TD(JJ)
KK = KK + 1 : GOTO 800
JJ = JJ + 1 : GOTO 750
RETURN

' SUBROUTINE FOR CHECKING TOOL TOLERANCES OF EACH "L" TOOL
FOR L = 1 TO NT
IF TOL(L) < DT(J) THEN GOSUB 920
K(J) = KK
NEXT L
RETURN

' SUBROUTINE FOR CHECKING EACH "LL" AVAILABLE MACHINE AND
ASSIGN TOOL "L"
930 'AND WILL SELECT "KK" TOOLS FOR EACH OPERATION "I"
940 KK = 0
950 SI = (LEN(MACH$(L) + 1)/5
960 FOR LL = 1 TO SI
970 P = 5 * (I-1) + 1
980 EACH$(LL) = MID$(MACH$(L), P, 4)
990 IF CODE$(J) <> EACH$(LL) THEN 1040
1000 KK = KK + 1
1010 TOOL$(J, KK) = TIDS$(L)
1020 TOOLTOL$(J, KK) = TOL$(L)
1030 TCOST$(J, KK) = COST$(L)
1040 NEXT LL
1050 RETURN
APPENDIX F

LISTING OF PARA.BAS PROGRAM
I = 1
OPEN "A: OPERATEN.DAT" FOR INPUT AS #1
IF EOF(1) THEN CLOSE #1 : GOTO 60
INPUT #1, IN, SN(I), DIMEN(I), DT(I), J(I), P$(I), CODES(I),
PROB(I), POWER(I), MINSP(I), MAXSP(I), F(I), TOOL$(I), TCOST(I)
I = I + 1 : GOTO 30
M = I - 1
PRINT
INPUT "ENTER THE HARDNESS OF THE WORKPIECE (HB) : "; HB
PRINT
PRINT "WORKPIECE MATERIALS ARE STEEL (S), FREE-MACHINING STEEL (F), OR CAST IRON (C)"
INPUT "ENTER THE WORKPIECE MATERIAL : "; WPMS
OPEN "A: PARA.DAT" FOR INPUT AS #2 : N = 1
IF EOF(2) THEN CLOSE #2 : GOTO 160
INPUT #2, HARD(N), SHS(N), SHF(N), SCS(N), SCF(N), FHS(N),
FHF(N), FCS(N), FCF(N), CHS(N), CHF(N), CCS(N), CCF(N)
N = N + 1 : GOTO 130
FOR J = 1 TO M
PRINT
PRINT "TOOL MATERIALS ARE CARBIDE (C) OR HIGH SPEED STEEL (H)"
PRINT "ENTER THE TOOL MATERIAL : "; TM$"
APPENDIX G

LISTING OF HOLDING.BAS PROGRAM
10  'HOLDING.BAS PROGRAM
20  OPEN "OPERATE.DAT" FOR INPUT AS #1
30      N = 1
40  IF EOF(1) THEN CLOSE #1 : GOTO 80
50  INPUT ON(N),SN(N),P$(N),PPR(N),MS(N),MPR(N),T$(N),S(N),
    F(N),DOC(N)
60      N = N + 1
70  GOTO 40
80  FOR    L = 1 TO N-1
90  OPEN "DIMTOL.DAT" FOR INPUT AS #3
100     J = 1
110  IF EOF(3) THEN CLOSE #3 : GOTO 160
120  INPUT SURF(J),DI(J),TOL(J)
130  IF SURF(J) <> SN(N) THEN GOTO 150
140      STOL(N) = TOL(J) : GOTO 170
150     J = J + 1 : GOTO 110
160  OPEN "HOLDING.DAT" FOR INPUT AS #2
170     M = 1
180  IF EOF(2) THEN CLOSE #2 : GOTO 300
190  INPUT HNS(N),PROC$(M),MC$(M),ACC(M),HC(M)
200  IF P$(L) <> PROC$(M) THEN 240
210  IF M$(L) <> MC$(M) THEN 240
220  IF TOL(L) <> ACC(M) THEN 240
230  GOTO 260
240     M = M + 1 : GOTO 180
250     H(L) = HN(M)
260  KILL "OPERATE.DAT"
270  OPEN "OPERATE.DAT" FOR APPEND AS #4
280  WRITE OP(L),SN(L),P$(L),M$(L),T$(L),S(L),F(L),DOC(L),F(L),
    H(L)
290  GOTO 180
300  NEXT L
310  CLOSE #4
320  END
APPENDIX H

LISTING OF SPC.BAS PROGRAM
'SPC.BAS PROGRAM
OPEN "OPERATE.DAT" FOR INPUT AS #1
N = 1
IF EOF(1) THEN CLOSE #1 : GOTO 80
INPUT ON(N),SN(N),PS(N),PPR(N),MS(N),MPR(N),TS(N),S(N),
F(N),DOC(N),HS(N)
N = N + 1
GOTO 40
FOR L = 1 TO N-1
OPEN "DIMTOL.DAT" FOR INPUT AS #3
J = 1
IF EOF(3) THEN CLOSE #3 : GOTO 170
INPUT SURF(J),DI(J),TOL(J)
IF SURF(J) <> SN(N) THEN GOTO 150
STOL(N) = TOL(J) : GOTO 160
J = J + 1 : GOTO 110
INPUT "ENTER THE CUTTOFF LEVEL FOR OPERATION INDEX : ";CF
OPEN "SPC.DAT" FOR INPUT AS #2
M = 1
IF EOF(2) THEN CLOSE #2 : GOTO 400
INPUT PROCS(M),MC$M,TOOL$(N),SP(M),FR(M),D(M),DIMEN(M),
UNTL(M),LNTL(M),PD(M)
IF PS(L) <> PROC$(M) THEN 290
IF MS(L) <> MCS(M) THEN 290
IF TS(L) <> TOOL$(M) THEN 290
IF S(L) <> SP(M) THEN 290
IF F(L) <> FR(M) THEN 290
IF DOC(L) <> D(M) THEN 290
SUNTL = UNTL(M) : SLNTL = LNTL(M) : DEF = PD(M)
GOTO 300
M = M + 1 : GOTO 190
USL = DI(J) + TOL(J)
LSL = DI(J) - TOL(J)
IF USL > SUNTL OR LSL < SLNTL THEN GOTO 390
K = K + 1
O1 = .5*(1-((USL-SUNTL)+((SLNTL-LSL))/(USL-LSL))
+ .5*(1-DEF)
IF CF < O1 THEN GOTO 390
KILL "OPERATE.DAT"
OPEN "OPERATE.DAT" FOR APPEND AS #4
WRITE OP(L),SN(L),PS(L),MS(L),TS(L),S(L),F(L),DOC(L),O1
GOTO 190
NEXT L
CLOSE #4
END
APPENDIX I

LISTING OF COMBINE.BAS PROGRAM
'COMBINE.BAS program for alternative process plan

OPEN "operate.dat" FOR INPUT AS #1
N = 1
IF EOF(1) = 0 THEN CLOSE #1 : GOTO 120
INPUT OP(N),SN(N)
N = N + 1
GOTO 40
FOR II = 1 TO M
NN = 0
FOR JJ = 1 TO M
IF II = JJ THEN GOTO 150
IF SN(II) <> SN(JJ) THEN GOTO 150
NN = NN + 1
OP(II,NN) = OP(II)
NEXT JJ
NEXT II
FOR K = 1 TO NN
IF K = NN-1 THEN PRINT USING " ##";OP(II,K) : GOTO 300
PRINT USING " ##";OP(II,K);
NEXT K
NEXT II
END
APPENDIX J

LISTING OF SEQUENCE.BAS PROGRAM
'SEQUENCE.BAS program allocates the operations in a logical sequence.
'Sequence is based on datum surfaces, dimensional tolerances, and the
'related operations on a surface.

DIM E(20), T(20), P$(20), D$(20), D$P(20), D$E(20), D$T(20), D$P$(20)
I = 1
OPEN "a:DIMTOL.DAT" FOR INPUT AS #1
INPUT #1, E(I), T(I), P$(I), D$(I)
IF D$(I) = "N" THEN GOTO 150
LI = DI + 1
DD$(DI) = D$(I)
DE(DI) = E(I)
DT(DI) = T(I)
D$P$(DI) = P$(I)
I = I - 1
IF EOF(1) THEN CLOSE ELSE I = I + 1 : GOTO 70
M = I
FOR N = 1 TO M-1
FOR K = 1 TO M-N
IF T(K) > T(K+1) THEN GOTO 290
TEMP = T(K)
TEMPE = E(K)
TEMPP$ = P$(K)
T(K) = T(K+1)
E(K) = E(K+1)
P$(K) = P$(K+1)
T(K+1) = TEMPE
E(K+1) = TEMPE
P$(K+1) = TEMPP$
NEXT K
NEXT N
PRINT
PRINT "INPUT OPERATIONS RANKED BY DIMENSIONAL TOLERANCE"
PRINT "-----------------------------------------------"
PRINT "SEQUENCE SURFACE NO. PROCESS TOLERANCE"
PRINT IF II > DI THEN 480
II = II + 1
IF DD$(II) < DD$(II+1) THEN GOTO 440
PRINT II+1, DE(II+1), D$P$(II+1), TAB(46);
PRINT USING "##.##", DT(II+1)
PRINT II, DE(II), D$P$(II), TAB(46);
PRINT USING "##.##", DT(II)
GOTO 480
PRINT II, DE(II), D$P$(II), TAB(46);
PRINT USING "##.##", DT(II)
PRINT II+1, DE(II+1), D$P$(II+1), TAB(46);
PRINT USING "##.##", DT(II+1)
FOR I = 1 TO M
PRINT I+DI, E(I), P$(I), TAB(46);
PRINT USING "#.###";T(I)
NEXT I
FOR J = 1 TO M
IF P$(J) = "THREADING" OR P$(J) = "COUNTERSINK"
THEN GOSUB 810
IF P$(J) = "BORING" THEN GOSUB 850
IF P$(J) = "CHAMFER" THEN GOSUB 890
IF P$(J) = "FORMING" THEN GOSUB 930
NEXT J
PRINT
PRINT "OPERATION SEQUENCE ADJUSTED BASED ON RELATED PROCESS"
PRINT
PRINT "SEQUENCE SURFACE NO. PROCESS TOLERANCE"
IF II > DI THEN 480
II = II + 1
IF DDS(II) < DDS(II+1) THEN GOTO 720
PRINT II+1,DE(II+1),DPS(II+1),TAB(46);
PRINT USING "#.###";DT(II+1)
PRINT II,DE(II),DPS(II),TAB(46);
PRINT USING "#.###";DT(II)
GOTO 760
PRINT II,DE(II),DPS(II),TAB(46);
PRINT USING "#.###";DT(II)
PRINT II+1,DE(II+1),DPS(II+1),TAB(46);
PRINT USING "#.###";DT(II+1)
FOR N = 1 TO M
PRINT N+DI,E(N),P$(N),TAB(46);
PRINT USING "#.###";T(N)
NEXT N
END
FOR K = 1 TO M
IF P$(K) = "DRILLING" AND E(K) = E(J) THEN GOSUB 970
NEXT K
RETURN
FOR K = 1 TO M
IF P$(K) = "DRILLING" AND E(K) = E(J) THEN GOSUB 970
NEXT K
RETURN
FOR K = 1 TO M
IF P$(K) = "TURNING" AND E(K) = E(J) THEN GOSUB 970
NEXT K
RETURN
FOR K = 1 TO M
IF P$(K) = "MILLING" AND E(K) = E(J) THEN GOSUB 970
NEXT K
RETURN
IF K < J THEN GOTO 1040
TP$ = P$(J)
TT = T(J)
P$(J) = P$(K)
T(J) = T(K)
1020 $P_S(K) = TP_S$
1030 $T(K) = TT$
1040 RETURN
APPENDIX K

LISTING OF COST.BAS PROGRAM
'COST.BAS program estimates the total production cost of
a proposed
' process plan. The output is returned into EXSYS
program for decision
'making.
DIM TCH(20),TLIFE(20),CT(20),SPEED(20),F(20),DOC(20),
POWER(20),TL(20),HC(20),PS(20),L(20),RD(20),D(20),W(20),H(20),
VR(20),SC(20),CL(20),CTIME(20),TC(20),MC(20),TCOST(20)
PI = 3.1465
CLS
INPUT "Hourly rate for labour is [$/hours] ",RT
INPUT "Hourly rate for electricity [$/kwh] ",RE
PRINT : INPUT "Do you want to enter the data from data file? (Y/N) ";ANS$;
100 IF ANS$ = "N" OR ANS$ = "n" THEN GOTO 180
110 PRINT : PRINT "<< Please wait
120 OPEN "A:\PLAN.DAT" FOR INPUT AS #1
130 INPUT #1,N
140 FOR II = 1 TO N
150 I = II
160 INPUT #1,TCH(I),TLIFE(I),CT(I),SPEED(I),F(I),DOC(I),
POWER(I),TL(I),HC(I),PS(I)
170 GOTO 480
180 PRINT
190 PRINT " INPUT DATA FOR EACH OPERATION OF THE PROCESS PLAN"
200 PRINT
210 INPUT "THE NUMBER OF OPERATIONS IN THE PROCESS PLAN IS ",N
220 PRINT
230 PRINT
240 FOR I = 1 TO N
250 PRINT " Please enter the information for OPERATION ";I
260 PRINT
270 PRINT " INPUT FOR TOOLING INFORMATION"
280 PRINT
290 PRINT "Time for changing the tool from TOOL LIBRARY is
[min] ",TCH(I)
300 INPUT "Tool life for the tool from TOOL LIBRARY is
[hours] ",TLIFE(I)
310 INPUT "Cost of the tool [$] ",CT(I)
320 PRINT
330 PRINT " INPUT FOR MACHINING PARAMETERS"
340 PRINT
350 INPUT "Cutting speed is [fpm] ",SPEED(I)
360 INPUT "Feed is [inch] ",F(I)
370 INPUT "Depth of Cut is [inch] ",DOC(I)
380 INPUT "Power required is [kw] ",POWER(I)
390 PRINT
400 PRINT " INPUT FOR HOLDING DEVICE"
410 PRINT
420 INPUT "Loading Time for a part from HOLDING.DAT is [mins]
",TL(I)
430 INPUT "Unit cost of the Jig or Fixture is [$/pc] ",HC(I)
440 PRINT
PRINT "INPUT FOR GEOMETRY"

INPUT "PROCESS TYPE IS [T=TURNING,D=DRILLING,M=MILLING]",PS(I)

IF PS(I) = "M" THEN 610
IF PS(I) = "D" THEN 560
PRINT : PRINT "The process is going to produce a cylindrical surface."
PRINT
INPUT "Length of the surface is [inch] ",L(I)
INPUT "Diameter of Raw material is [inch] ",RD(I)
VR(I) = L(I)*(RD(I)^2 - (RD(I) - 2)^2)*PI/4
GOTO 650
PRINT : PRINT "The process is going to drill a hole." : PRINT
INPUT "Diameter of the hole is [inch] ",D(I)
INPUT "Depth of the hole is [inch] ",L(I)
VR(I) = D(I)^2 * PI/4 * L(I)
GOTO 660
PRINT : PRINT "The process is going to mill the surface." : PRINT
INPUT "Length of the surface is [inch] ",L(I)
INPUT "Width of the surface is [inch] ",W(I)
INPUT "Height of the cavity is [inch] ",H(I)
VR(I) = L(I) * W(I) * H(I)
CALCULATION
SC(I) = TL(I) * RT / 60
CL(I) = VR(I) / (DOC(I) * F(I))
CTIME(I) = CL(I) / (SPEED(I) * 12)
TC(I) = CTIME(I) * ((TCH(I) * RT / 60) + CT(I)) / (TLIFE(I) / 60)
MC(I) = (CTIME(I) / 60) * (RT + (RE * POWER(I)))
TCOST(I) = SC(I) + TC(I) + MC(I) + HC(I)
CLS
IF ANS$ = "N" OR ANS$ = "n" THEN NEXT I ELSE NEXT II
SUMC = 0
CLS
PRINT "SEQ. SETUP TOOL MACHINING FIXTURE TOTAL"
PRINT " NO. COST COST COST COST"
PRINT
FOR N = 1 TO N
PRINT USING "# ####### ####### ####### #" ;N,SC(N),TC(N),MC(N),HC(N),TCOST(N)
SUMC = SUMC + TCOST(N)
NEXT N
PRINT : PRINT "TOTAL COST FOR THIS PROCESS PLAN IS";TAB(45);
PRINT USING "$####.##";SUMC
END
APPENDIX L

A PORTION OF DXF FILE OF THE SAMPLE PART IN THE CASE STUDY
DIMENSION
  8
  0
  2
&D0
  10
  1.5442
  20
  3.2707
  30
  0.0
  11
  1.5262
  21
  5.1507
  31
  0.0
  1
  %c1.225%p0.0005
  13
  3.0369
  23
  7.0308
  33
  0.0
  14
  3.0369
  24
  3.2707
  34
  0.0
  50
  90.0
  0
DIMENSION
  8
  0
  2
&D1
  10
  12.5594
  20
  3.6828
  30
  0.0
  11
  12.5414
  21
  5.1507
  31
  0.0
  1
  %c0.7485%p0.0005
  13
  11.427
<p>| | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>23</td>
<td>6.6187</td>
<td>33</td>
<td>0.0</td>
<td>14</td>
<td>11.427</td>
<td>24</td>
<td>3.6828</td>
</tr>
<tr>
<td>34</td>
<td>0.0</td>
<td>50</td>
<td>90.0</td>
<td>0</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**DIMENSION**

<p>| | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>0</td>
<td>2</td>
<td>*D2</td>
<td>10</td>
<td>3.1656</td>
<td>20</td>
<td>2.8587</td>
</tr>
<tr>
<td>30</td>
<td>0.0</td>
<td>11</td>
<td>3.7806</td>
<td>21</td>
<td>2.8767</td>
<td>31</td>
<td>0.0</td>
</tr>
<tr>
<td>1</td>
<td>0.125</td>
<td>13</td>
<td>2.8568</td>
<td>23</td>
<td>3.6313</td>
<td>33</td>
<td>0.0</td>
</tr>
<tr>
<td>14</td>
<td>3.1656</td>
<td>24</td>
<td>3.1677</td>
<td>34</td>
<td>0.0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

**DIMENSION**

<p>| | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>0</td>
<td>2</td>
<td>*D3</td>
<td>10</td>
<td>6.5886</td>
<td>20</td>
<td>2.4787</td>
</tr>
<tr>
<td>30</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
0.0
11
4.7227
21
2.4967
31
0.0
1
1.125%
0.003
13
2.8568
23
3.6313
33
0.0
14
6.5886
24
3.1935
34
0.0
0

DIMENSION
8
0
2
*D4
10
6.9489
20
2.4787
30
0.0
11
7.6539
21
2.4967
31
0.0
1
0.1875
13
6.5886
23
3.1935
33
0.0
14
6.9489
24
3.5798
34
0.0
0

DIMENSION
8
0
2
*D5
10
9.4453
20
1.9058
30
0.0
11
6.151
21
1.9238
31
0.0
1
2.125%p0.004
13
2.8568
23
2.5754
33
0.0
14
9.4453
24
3.5798
34
0.0
0
DIMENSION
8
0
2
*D6
10
11.4785
20
1.5258
30
0.0
11
7.1676
21
1.5438
31
0.0
1
2.875%p0.005
13
2.8568
23
2.5754
33
0.0
14
11.4785
24
3.8115
34
0.0
0
DIMENSION
8
0
2
*D7
10
9.7542
20
3.1677
30
0.0
11
10.3692
21
3.1857
31
0.0
1
0.125
13
9.4453
23
3.5798
33
0.0
14
9.7542
24
3.5798
VITA AUCTORIS

1967 Born in Hong Kong on the 23th of May.

1985 Completed high school education from Brebeuf College School, Willowdale. Ontario, Canada.

1989 Graduated from University of Windsor, Windsor, Ontario, with a B.A.Sc. Degree in Industrial Engineering.

1991 Currently a candidate for the M.A.Sc. Degree in Industrial Engineering at the University of Windsor, Ontario, Canada.