MODIFICATION OF A VERTICAL MACHINING CENTRE TO INCLUDE TURNING OPERATIONS:-

AN INVESTIGATION.

By

MWANZA SICHINGWENJE MOFFAT

A dissertation submitted to the University of Zambia in partial fulfillment of the requirements for the degree of Master of Engineering in Production Engineering and Management.

SCHOOL OF ENGINEERING
THE UNIVERSITY OF ZAMBIA

LUSAKA

2001

TABLE OF CONTENTS

DECLARATION

I, Mwanza S. Moffat, do hereby declare that this dissertation represents my own work and that, to the best of my knowledge it has not been previously submitted for the award of a degree at this or any other University.

Signed: Mwanza

Date: July 6th, 2001

APPROVAL

This thesis of Mwanza S. Moffat is approved as fulfilling partial requirements for the award of the Masters Degree in Production Engineering and Management by the University of Zambia.

Examiners	**Signature**	**Date**
1. DR. PATRICK K. NKANZA	[signature]	12-01-2001
External Examiner		
2. DR. CHARLES K. WAMUKWAMBA	[signature]	11th July 2001
Internal Examiner		
3. DR. L. SIAMINWE	[signature]	11-07-2001
Internal Examiner		

DEDICATION

To my daughters, ***Zangose*** and ***Thabo***.

ACKNOWLEDGEMENTS

I am most grateful to my supervisors Dr. Henry M. Mwenda and Dr. Levy Siaminwe for their invaluable guidance and encouragement throughout the study. I also wish to thank the Technology Development and Advisory Unit (TDAU) and Mechanical Engineering workshops staff, for their instrumental role in the fabrication of fixtures and execution of experiments. I am indebted to my sponsor, the University of Zambia for making my MEng. training a reality through their financial involvement.

Special thanks to Dr. Patrick Kona Nkanza and Dr. C. K. Wamukwamba for their special involvement in examining this work. Many thanks to the TDAU management and my wife Emelda for allowing me to proceed with my studies with full support and encouragement. And finally, I thank all those not mentioned who have helped in one way or another to the success of this study.

LIST OF ABBREVIATIONS AND SYMBOLS

AC........................	Alternating Current
ATC........................	Automatic Tool Changer
CNC........................	Computer Numerical Control
CRT........................	Cathode Ray Tube
DC........................	Direct Current
EIA........................	Electronic Industrial Association
EUT........................	Eindhoven University of Technology
ISO........................	International Standard Organization
MDI........................	Manual Data Input
MIT........................	Massachusetts Institute of Technology
NC........................	Numerical Control
TDAU......................	Technology Development &Advisory Unit
UNZA......................	University of Zambia
VMC........................	Vertical Machining Centre

Some Numerical Control ISO-codes

O............................	Programme number for program identification
N............................	Sequence number for line identification
G............................	Preparatory functions
X............................	X axis designation
Y............................	Y axis designation
Z............................	Z axis designation
R............................	Radius designation
F............................	Feedrate designation
S............................	Spindle speed designation
H............................	Tool length designation
D............................	Tool radius designation
T............................	Tool designation
M............................	Miscellaneous function

Symbols used in calculations

F_c = compressive force on the shear plane

F_s = shear force on the shear plane

d = depth of cut (mm)

f = feed (mm rev^{-1})

V = cutting speed (m min^{-1})

T = torque (N m)

D = Drill diameter (mm)

N = rotational speed (rev min^{-1})

W = width of cut (mm)

f_m = machine table feed (mm min^{-1})

k_l = machining factor for lathe work (N mm^{-2})

k_d = machining factor for drilling work (J mm^{-3})

k_m = machining factor for milling work (J mm^{-3})

LIST OF FIGURES

LIST OF APPENDICES

ABSTRACT

There has been an accelerated use of Computer Numerical Control machine tools by developed countries over the past decade resulting in manufacturing companies becoming more competitive on the world market. However, the initial cost of these machines is prohibitive to many companies in the third world countries such as Zambia. Therefore, methods of adapting these machines to do much more than initially designed for would make the investment much more attractive. Various CNC machine builders have developed latest designs of combined turning and milling centres. This has been made possible through construction of more than three axes of rotation. None of the three-axis machining facilities convert a machining centre to a turning centre. It was the goal of this research to investigate the possibility of modifying a 3-axes CNC Vertical Machining Centre (VMC) to include CNC turning operations. This report outlines the research activity in the Department of Mechanical Engineering - School of Engineering at University of Zambia, exploring the methodology for widening the capability of a VMC to operate as a Turning Centre as well. The main objective was to add a modification to the existing Supermax 65A Vertical Machine Centre, so that turning operations could be performed within the designed parameters and programming software.

During the research, fixtures for tool and workpiece holding were designed and produced. These were designed in line with the requirement for turning operations, original machine tool designed features and parameters. The use of Mastercam Mill software (Direct Numerical Control) and manual programming (Manual Data Input) were demonstrated as options for carrying out numerical control turning operations. Three samples of typical turned profiles were produced using designed fixtures. An estimation of force and power consumed during these operations was carried out. Although the study had certain limitations as regards the size and weight of turned jobs, it enhanced the number of part profiles that could be produced on the CNC Supermax 65A. The results proved that CNC machine tools could be improved for adaptability and productivity at minimum cost to a company.

SUMMARY

The main aim of this project was to investigate the possibility of modifying a 3-axes CNC Vertical Machining Centre (VMC) to include CNC turning operations. In order to achieve the stated aim, the following were identified as the main objectives:

i) To investigate the possibility of modifying the 16-tool CNC Vertical Machining Centre (Supermax 65A) with Fanuc Control system, in the Mechanical Workshop of the Department of Mechanical Engineering - School of Engineering at the University of Zambia, to include CNC turning operations.
ii) To design, construct and test the prototype mechanism for turning operations.
iii) To investigate the use of Mastercam Mill software to prepare CNC programs for turning operations.

In view of the sophistication and advanced construction of most CNC machine tools and the fact that very few of such machines are found in Zambia, time was spent to study those that were available at Boart Longyear and the University of Zambia. Information on general CNC machine tool designs, construction and related works on widening their capabilities was also gathered from selected manufacturers of CNC machine tools.

The overall research was divided into various sections beginning with a general overview on the development of CNC technology. In subsequent sections, a detailed study of design features of the Supermax 65A was conducted as a prerequisite to proposed design modifications and selection of the most suitable modification. Following a successful design, production and testing of the most suitable modification, part programming options (Manual Programming and Direct Numerical Control) for carrying out numerical control turning operations were considered. Three samples of typical turned profiles were produced using designed fixtures. An estimation of force and power consumed during these operations was carried out. Due to the observed changes in chip formation when turning chamfer and radius sections, a consideration for tool selection was conducted. It was recommended that reasons for these changes could be investigated further.

The study has been able to increase the number of part profiles that could be produced on the CNC Supermax 65A. However, it had certain limitations as regard the size and weight of turned jobs that could be done. This was due to the fact that its original construction as a 3-axes vertical machining centre allows the spindle to move up and down. Nevertheless, the research results proved that CNC machine tools could be improved for adaptability and productivity at minimum cost to a company.

CHAPTER ONE

INTRODUCTION

1.1 BACKGROUND

The global market-place is increasingly being characterized by short product life cycles, increased product variety and extensive customization. To retain and increase market share in such a competitive environment, firms have to be more flexible in their operations and satisfy different market segments. The ability of ***Computer Numerical Control (CNC)*** machine tools to accomplish the stated goals is now well appreciated by any manufacturing firm competing on the global market.

The most important reasons for working with CNC-machine tools include the fact that [**1**]:

i) With the high production speed of the CNC machine tool, it is possible to make more products with fewer man-hours. This means the process costs are reduced and the products can be made cheaper.

ii) With conventional machine tools, mostly only 20 to 30 percent of the available machine hours are used for machining of the final product. The rest of the time is used for reading the part print, positioning, gearing, tool-changing, clamping the product, etc. Using CNC machine tools increases actual productivity time to over 80 percent.

iii) There is less human involvement in the actual machining process when using CNC machine tools as compared to conventional machining, with a result that products produced are within required quality standards as indicated in quality curves in Figure 1. Given the same level of minimum acceptable level of quality, it is evident that the number of final products rejected is more in conventional machining and almost none in CNC machining.

iv) There is a *Shorter run-time* as a result of integration of different jobs, hence allowing production of more complex products in one fixation.

1.2 PROBLEM DEFINITION

Various Computer Numerical Control (CNC) machine builders have developed latest designs of combined turning and milling centres. This is aimed at increasing the capabilities of single machine tools to do more processes within the same work-station. This has been made possible through construction of more than three axes of rotation and machining. However, none of the existing three-axis machining facilities are able to do both milling and turning operations. This was also confirmed by two of the CNC machine builders (Spinner and Hankook) who were contacted during the course of this work. This particular research was proposed to specifically investigate the possibility of modifying a 3-axes CNC Vertical Machining Centre (VMC) to include CNC turning operations.

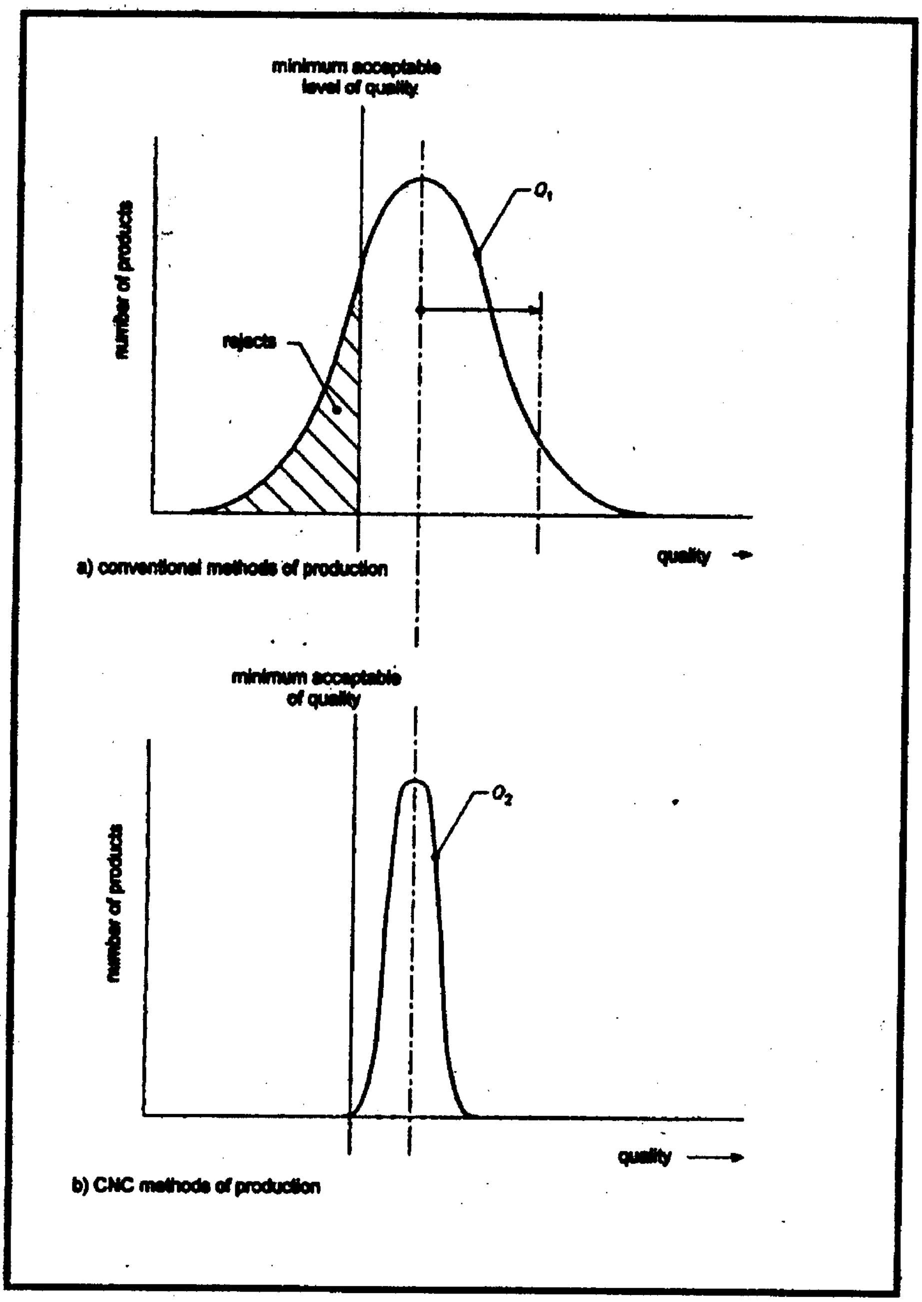


Fig. 1 Quality curve

1.3 JUSTIFICATION

It is generally agreed that the rapid expansion of the Japanese economy from the late 1950s through the 1960s was powered by a vigorous investment of private industry in new plant, quality controls and modern equipment. Investment for modernization made Japanese industries more competitive on the world market, created new products and brought Japanese enterprises the benefit of mass production and improved productivity per worker. Dramatic increases in private-sector capital investment and growth in export sales also helped to bring the Japanese economy out of the long tunnel of recession during the 1980s. This, among other practices resulted in real growth to climb to 4.5 percent in fiscal 1984 (April 1984 - March 1985) and 4.8 percent in fiscal 1985 **[2]**.

Many countries of South-East Asia have followed the Japanese example and have become competitors on the World market. Within our sub-region, South Africa has been able to dominate the market in engineering products, mainly because of modernizing their manufacturing industry [**3**]. The use of CNC machine tools has enabled firms to make their production systems more competitive in flexibility and costs. They have further enhanced their strengths of product quality and customer service [**4**].

There are a number of these machines in use in Zambia at present. Boart Longyear has CNC machine tools. These are:

i) Turning Centre for manufacturing specific mining spare parts and rock drill tools.

ii) Five axes Drilling Machine for drilling holes in all bit blanks and all precision milling operations.

iii) Helical Spring Coiler for producing hot coilings of both left and right handed helical steel springs.

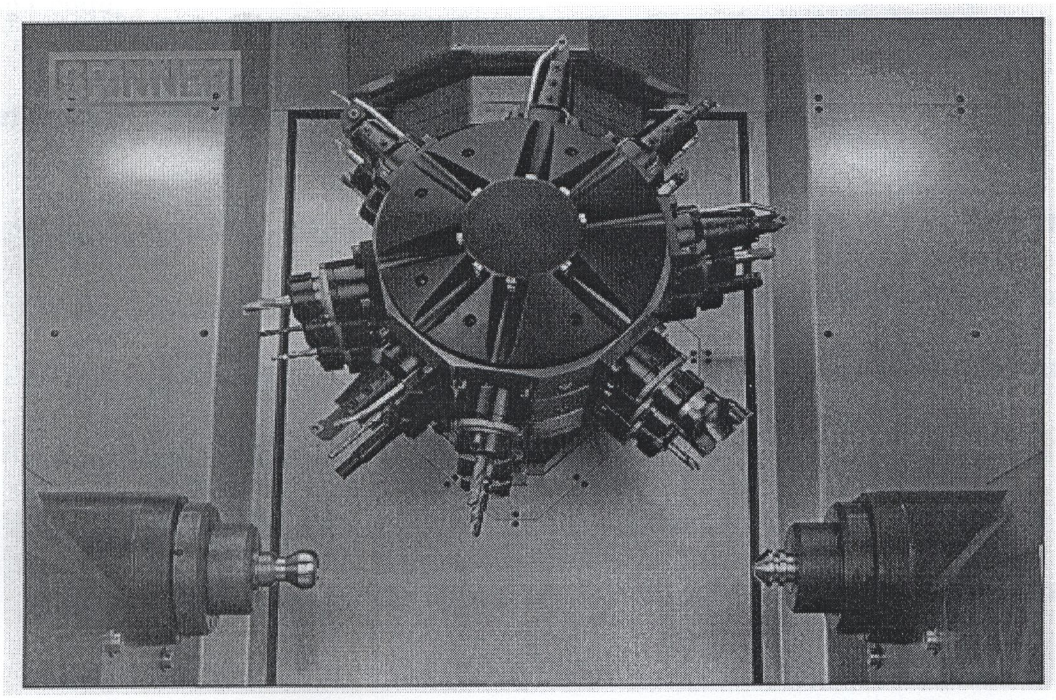
Due to their specific production processes and product lines, the company does not make any other turned parts, hence wanted to sub-contract the School of Engineering at the University of Zambia to have some of their turned parts (requiring high volumes and precision) made on the CNC machine tool. The School of Engineering recently acquired a CNC Vertical Machining Centre through its technical cooperation with Eindhoven University of Technology (EUT) in Holland.

Various contacts with CNC machine tool builders revealed latest designs of combined turning and milling centres. The latest Spinner Turning and Milling Center (TM) is a complete type of machine tool for complex machining of precision parts at the highest level of quality in small and medium quantities [**5**]. The machine brings together all machining capability of a 5-axes machining centre and a twin spindle turning cell with main and sub-spindle. Two identical spindles are located opposite to each other and moveable in the axes *Z1* and *Z2*, while clamping the workpiece during machining of first and second operations. In between the two spindles there is the 24-station turret movable in X and Y axes, and rotates in each angle around the B-axis. Complex turning and milling of all possible 3-dimensional geometries by using up to 5-axes interpolation is possible.

Figure 2 shows the general design concept of the Spinner TM costing $300,000, a cost prohibitive to most engineering companies in the third world countries such as Zambia.

It was therefore justifiable to extend the capabilities of existing single machine tools as a way forward for accelerating their use in third world countries such as Zambia.

Figure 2: SPINNER TURNING AND MILLING CENTRE [5]



1.4 RESEARCH OBJECTIVES

The aim of this research work was to add a modification to a Supermax 65A Vertical Machine Centre, so as to have a machine that would operate as a Vertical Machining and Turning Centre.

In order to achieve the stated aim, the following were identified as the main objectives:

i) To investigate the possibility of modifying the 16-tool CNC Vertical Machining Centre (Supermax 65A) with Fanuc Control system, in the Mechanical Workshop of the Department of Mechanical Engineering - School of Engineering at the University of Zambia, to include CNC turning operations.

ii) To design, construct and test the prototype mechanism for turning operations.

iii) To investigate the use of Mastercam Mill software to prepare CNC programs for turning operations.

1.5 METHODOLOGY

In view of the sophistication and advanced construction of most CNC machine tools and the fact that very few of such machines existed in Zambia, time was spent to study those that were available at Boart Longyear and UNZA. Information on general CNC machine tool designs, construction and related works on widening their capabilities was also gathered from selected manufacturers of CNC machine tools. These included Supermax of Taiwan, Hankook of South Korea, Monarch of USA, Spinner of Germany and, Giddings and Lewis of USA.

The following tasks were carried out:

1. Preliminary study of CNC machine tools at Boart Longyear and UNZA. The information gathered was related to CNC machine tool design criteria and programming for metal cutting. Literature on machine tool designs was also received from various CNC machine tool builders, who expressed interest in the research.

2. Detailed review of programming features for the VMC at the School of Engineering. Relevant information was obtained from selected builders and suppliers of CNC machine tools, on specific experiences on programming, tooling arrangements and machine setting.

3. Prototype design and programming for turning mechanism.

4. Manufacture of any required additional parts, assembly and testing.

CHAPTER TWO

COMPUTER NUMERICAL CONTROL TECHNOLOGY

2.1 CONCEPT OF NUMERICAL CONTROL

Controlling a machine tool by means of a *prepared program* is called Numerical Control (NC), as defined by the Electronic Industrial Association (EIA) [**6**]. The precursor of the CNC-technology was Numerical Control. The first prototype of NC machine was built in 1952 at the Massachusetts Institute of Technology (MIT) in the United States. With NC, the machine tool was controlled with the help of characters. In this type of control, an electronic apparatus translated the input characters into instructions and also checked if these were correctly executed by the machine tool. The first Numerical Control machine tools were equipped with electronic valves. In the early times the use of NC technology was quite rare because of high costs, but with the introduction of transistors, costs became lower and NC became more common. By 1965, Numerical Control machine tools were fitted with computers and Computer Numerical Control (CNC) became a reality.

2.2 CNC MACHINE TOOL LAYOUT

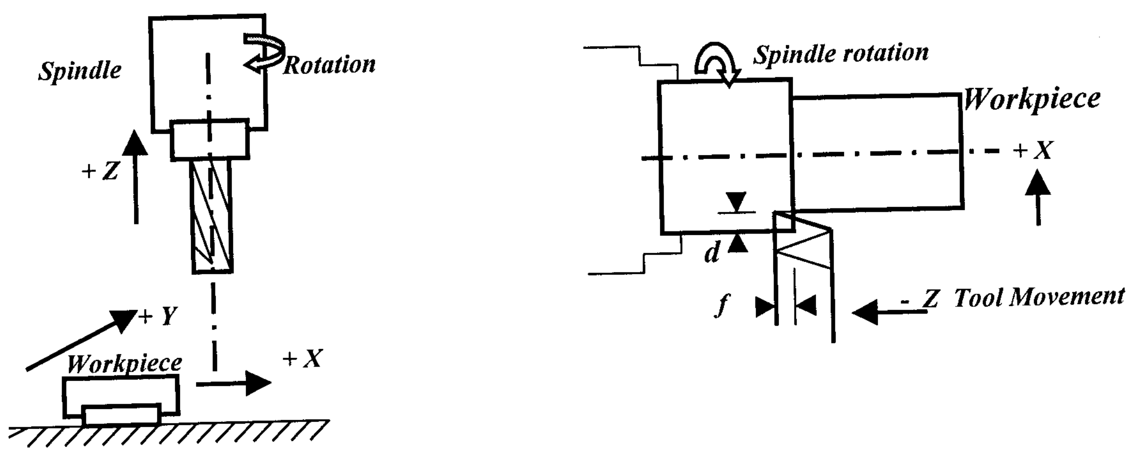
Computer Numerical Control machine tools are generally considered to fall into one of the two categories: Lathe (Turning Centers or Horizontal Machining Centres) and Milling machines (Vertical Machining Centres).

2.2.1 Milling Machine Layout (Vertical Machining Centres)

Milling machines are used for machining of components which are irregular shaped or require operations that cannot be provided by a turning machine. Typical milled components include dies, plates, brackets, or components with slots or holes in them.

Many of the existing vertical centres have vertical spindles that could either be fixed in one axis (Z) or allow tilting. The spindle is used to hold the cutting tool, which may be a milling cutter, drill, reamer or tap. The spindle rotates to facilitate metal cutting. The horizontal table or bed is used as a mounting surface for the workpiece. The table has mutually independent movements in two directions X and Y. The X and Y movements and the spindle Z movement constituent the three basic axes for milling operations (Figure 3a).

Figure 3: Machine Tool Layout



a) Milling Layout   *b) Turning Layout*

2.2.2 Lathe Layout (Turning Centres)

Lathes are used for the manufacture of rotationally symmetrical components by turning, boring, drilling, or thread cutting from solid bars, hollow tubes, fabricated rings or castings. The main components include the chuck, which is used as a workpiece holding device and it rotates to facilitate metal cutting. The tailstock enables long workpieces to rotate true, while clamped in the chuck. There is a saddle that carries the cross slide. The saddle traverses the lathe bed from the headstock to the tailstock, providing movement in the Z direction. Movement at 90 degrees to the lathe bed is facilitated by the cross slide, which carries the tool post. Hence, movement of the tool is possible in the X and Z-axes (Figure 3b).

The turret is used in addition to the tool post for holding cutting tools. Turrets have a number of stations providing a variety of tools, and are capable of being indexed automatically. A turret may be mounted vertically or horizontally depending upon the type of machine tool.

The most important requirement to be fulfilled by any machine tool is that it should be able to produce jobs with high degree of accuracy consistently over a long period of time. For this, it is essential that they possess static as well as dynamic rigidity. Rigidity of a machine tool is its capability to resist deformation produced due to the introduction of cutting forces generated while machining.

2.3 DESIGN OF NUMERICAL CONTROL MACHINE TOOLS

The general objectives behind the development of Numerical Control (NC) technology were:

i) To reduce production cost by reducing production time. This was achieved by reducing non-production time on the number of setups; setup times, workpiece handling time, tool change time and lead time etc.

ii) To increase production. With CNC machine tools, machine setting is done once and the rest of the time is dedicated to production of parts, hence larger numbers of parts are produced.

iii) To improve quality and accuracy of manufactured parts, as a result of less human involvement in actual production. The high repeatability accuracy of CNC machines tools has made this possible.

iv) To ensure that design changes could easily be incorporated without having to change fixtures and tools. NC machining is thus best suited to small and medium batch producers. The function like displacement of machine slides, angular rotation of circular table, start/stop main spindle, changing spindle speeds, reversing spindle direction of rotation, changing feedrate of slides, rotating tool turret, changing tools, switching on/off cutting fluid, locking/clamping table in position could easily be automated to relieve the operator.

v) To enable the manufacturing of complex or complicated jobs. This was done through the provision of more than 2-axes of rotation.

The important design considerations for NC machine tools are:

i) The machine structure should be able to withstand normal weight distributions and the effect of changing the position of a heavy slide. Stresses set up by rapid acceleration or deceleration of moving masses and by cutting forces should not cause objectionable deflections. Inaccuracies due to change in temperature at various locations and different thermal expansions, alignment etc. should also be considered.

ii) Slideways should be destined to respond quickly to command signals, and offer constant frictional resistance. For rapid response rolling friction has to be substituted for sliding friction in slideways. Roller and ball slideway elements achieve this objective but these provide less clamping and have a lesser load carrying capacity.

iii) Usually variable speed drives are used to obtain different spindle speeds and feedrates. Hydrostatic hydraulic drives are used to maximum in NC machine. Hydraulic systems incorporating both variable delivery pumps and variable displacement motors are used.

iv) Slides could be positioned by any of the mechanism listed below:

(a) Hydraulic ram.

(b) Re-circulating ball leadscrew and nut.

(c) Rack and pinion.

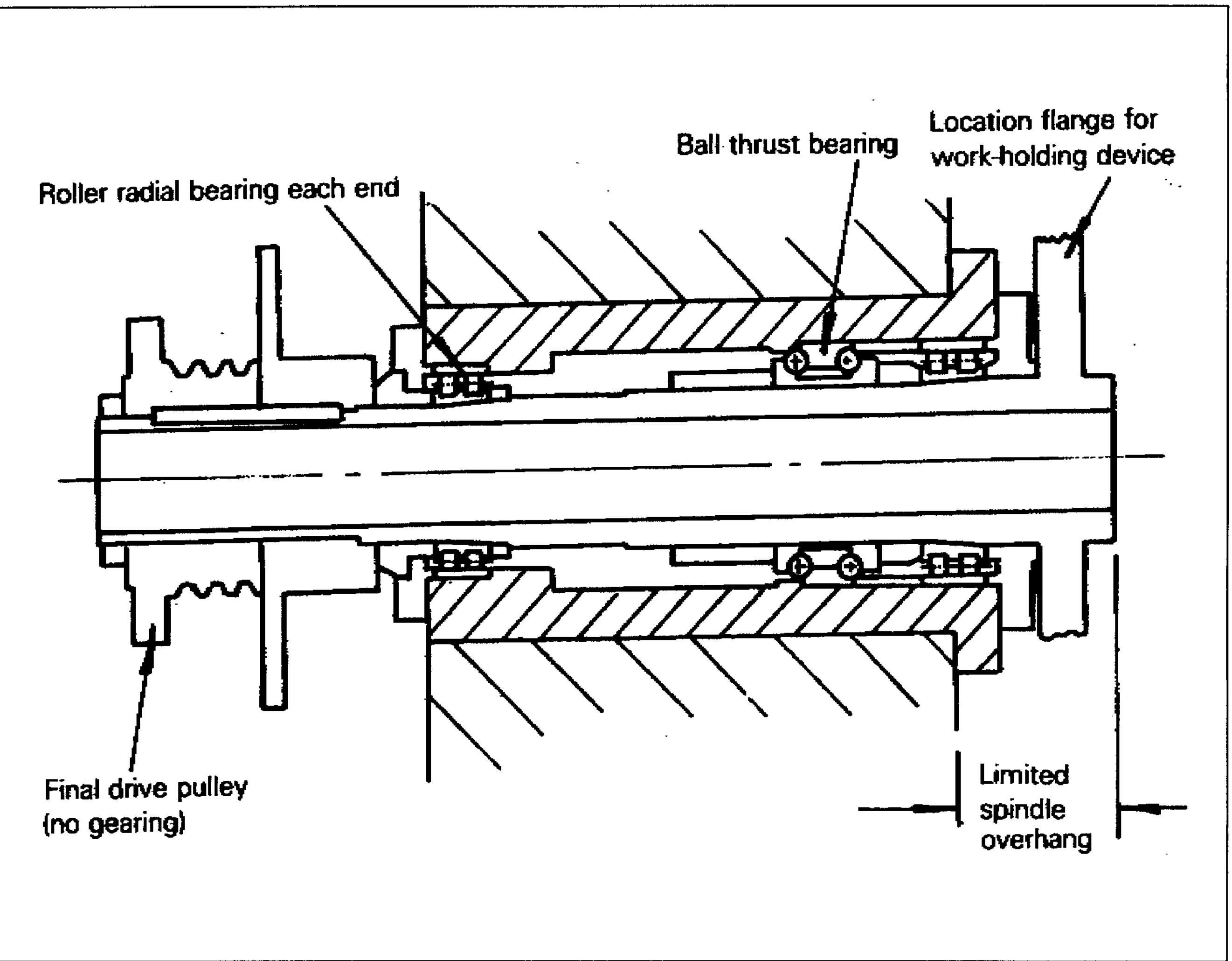
2.3.1 Machine Spindle

The machine spindle is a very important part in any machine tool. The possibility of deflection as a result of dynamic loading should be considered. Deflections such as those caused by radial forces on a milling machine need only be quite small to affect the dimensional accuracy of the workpiece [7]. In addition to the radial loads that cause deflection, a spindle assembly is also subjected to a thrust load acting along its axis. The design of the spindle assembly is such that these loads are adequately contained. Inadequate support results not only in dimensional inaccuracies but also in poor surface finish and chatter. A well-supported spindle assembly design is shown in Figure 4. Note that the spindle overhang is kept to a minimum, a common feature of turning and horizontal machines.

The spindle of vertical machining centres presents additional problems, since it is a traditional feature of this type of machine for spindle to move up and down. Obviously the more the spindle is extended, the greater the risk of deflection. In more recent designs attempt has been made to move away from the moving-spindle concept and instead the whole head assembly moves up and down.

The forces that cause deflection of the spindle also result in a tendency for the complete spindle-housing assembly to twist. This has resulted in an increased use of bifurcated or a two-pillar structure where the spindle housing is located between two substantial slideways that reduce the possibility of twisting.

Figure 4: Spindle Assembly for Turning Centre [7]



2.3.2 Spindle Drives

Two types of electric motors are used for spindle drives: Direct Current (DC) and Alternating Current (AC). They may be coupled directly to the spindle or via belts and/or gears. Many machines have a final belt drive, which is quieter and produces less vibration than a geared drive.

The majority of modern machines use DC motors. By varying the voltage input their speeds are infinitely variable as they rotate and so a constant cutting speed can be maintained. The torque available from a DC motor is constant throughout most of the speed range.

There are many machines fitted with specially designed AC motors that also provide for variable spindle speeds. The use of AC motors usually involves a stepped drive. A series of spindle speeds will be available and the selection of a particular speed may involve switching from one speed range to another, a feature that is common on many conventional machines.

2.4 COMPONENTS OF A CNC MACHINE TOOL

The major components of any CNC machine tool include the following:

i) The main frame that normally consists of a rigid cast-iron column and base, ground guideways, AC feed motors and preloaded ball-screws. This forms the main working station and contains features for workpiece and tool holding.

ii) An Automatic Tool Changer (ATC) or Turret Head (TH) mechanism.

iii) Central lubrication for guideways and ballscrew nuts.

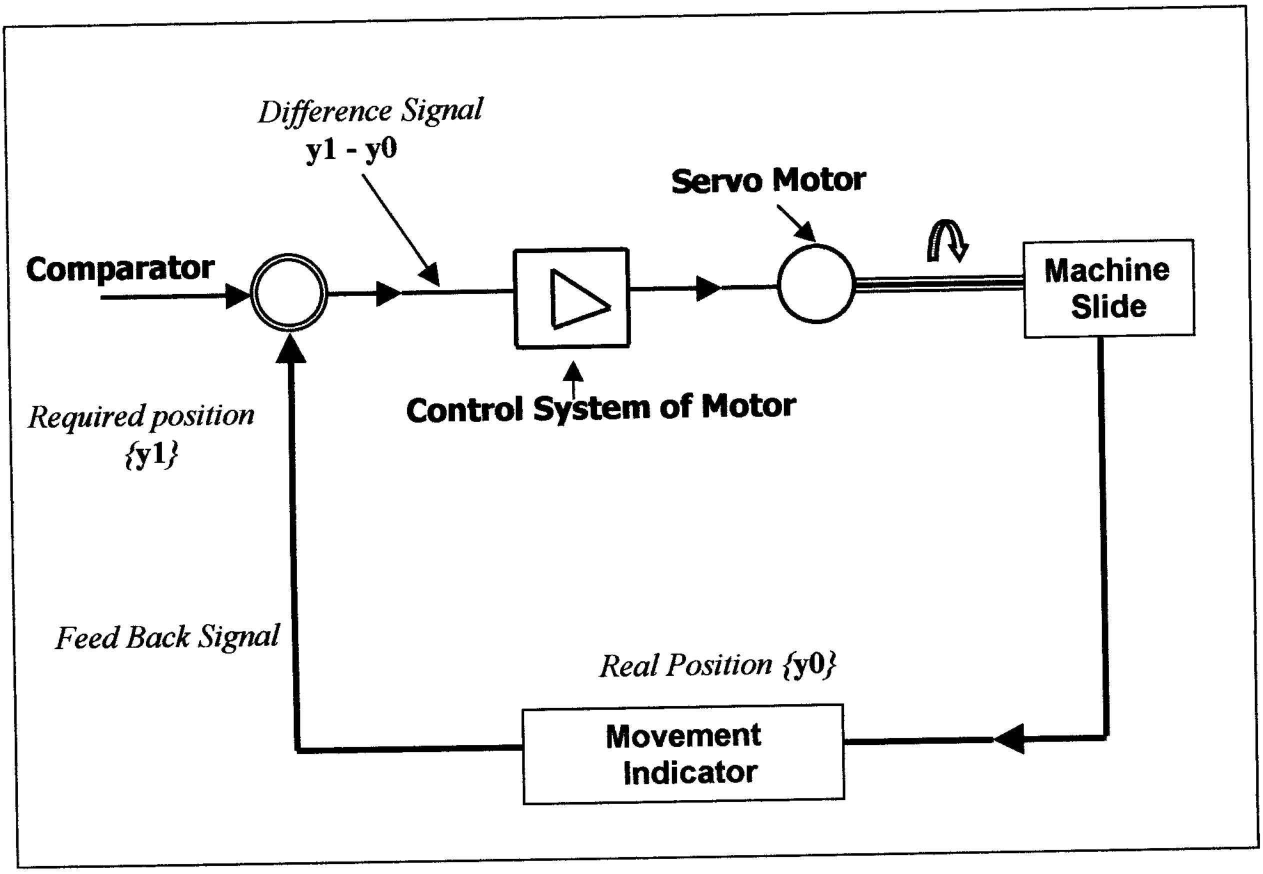
iv) High-pressure coolant system.

v) Computer Numerical Control Controller.

One of the main differences between a conventional and a CNC machine tool is that in a CNC machine, each axis has its own ***SERVO motor***. These special kinds of motors are designed to control slide positioning and acceleration. The servo motor is connected to a "closed loop control system" and is computer controlled. It is possible to make a link between the drive of a slide and the main axis on the CNC machine. This link is made electronically and is necessary to achieve the feed/rotation.

A schematic CNC "closed loop control system" is shown in Figure 5. The movement of the slide is continuously monitored by the measurement system. The measurement signal received by the ***control system*** is compared with the instruction by a ***comparator***. As long as there is a difference between the instruction and the measurement signal, the control system continues to send instructions and the slide continues to move to the instructed position. In this case, possible mechanical deviations are eliminated.

Figure 5: Major functional units of a CNC control system [6].



2.5 VERTICAL MACHINING CENTERS VERSUS TURNING CENTERS

There are certain marked differences and similarities between the two general designs and layouts highlighted in the section 2.2. This was also evident from those that were studied at Boart Longyear and the various pamphlets from CNC machine tool manufacturers.

Some of the major differences include the following:

i) The VMCs have specially designed rotating spindles for carrying standard tool holders. Side by side with the spindle will usually be an automatic tool changing system. The spindle head can either be tilting or not. In TCs, the spindle head is specially designed to carry various workpiece-holding devices.

ii) In TCs, the tool holding mechanism is usually made up of indexable turrets that traverse the length of the main spindle axis. The turret head will usually have all the required turning tools for a particular job. The turret assembly forms the main bed of the turning centre. In VMCs, the main table is used for workpiece holding that allows movements on horizontal X-Y coordinate system.

iii) In VMCs, the cutting process is usually a multi-edge cutting process in which multi-edges of a milling tool are engaged in the formation of the final workpiece profile, hence the requirement for tool rotation. In TCs, the processes are mainly single-point cutting for production of final workpiece profile, hence the need for the workpiece to be rotating against a fixed tool. The only exception is the drilling process in either case.

There are however some major similarities in the design and layout of both milling and lathe machine centres. These similarities have been considered by many CNC machine tool builders in coming up with latest single machines with capabilities to do both milling and turning precision works. The similarities include the fact that:

i) In both layouts, there is a requirement for a rotating system and a translatory system. This is seen in examples of latest combined turning and milling centres where increased machine tool capabilities are enhanced through provision of more axes of rotation. Spinner CNC machine builders **[8]**, have developed machines that have a vertical main spindle and a moveable table mounted spindle that allows turning to be performed on some of their vertical machining centres.

ii) Almost all machine tools (VMCs or TCs) have spindle thrust bearings that are designed to take forces in either direction of rotation, hence any reversal in force direction would not give a major design constraint.

iii) Existing standard tool or workpiece holders could easily be adopted for a particular modification requirement. This could be done either through additional fixtures or adoption of the suitable standard holders. The Hankook CNC Vertical Turning and Boring machines **[9]** have standard taper tool holders attached to specially designed square turrets for carrying turning tools.

It was appreciated that a 3-axes VMC could not operate as TC because of many different design, construction and control systems required in each category, hence the necessity of carry out an adaptive investigation using identifiable similarities.

CHAPTER THREE

DESIGN FEATURES OF THE SUPERMAX 65A

3.1 INTRODUCTION

The Supermax 65A is part of the Vertical Centre Series (VC-Series) of powerful but economic and compact Vertical Machining Centres, made by Yeong Chin Machinery Industries of Taiwan, for small and medium batch productions. They are made from a rigid full-casting design with large sized ball-screws and sliding guide-ways, which provide for most modern digital control and drive systems for highest precision. The machine uses FANUC CNC controller from one of the worldwide accepted suppliers of CNC controls called FANUC Corporation of Japan.

3.2 DESIGN OF SUPERMAX 65A MACHINE TOOL

The Vertical Machining Centre (VMC - milling layout) used to investigate the possibility of including turning operations is called ***YCM - VMC - 65A Supermax***. Its typical design and construction features are given in Figure 6. They include the following:

i) A rigid cast-iron column and base. The machine is equipped with high precision ball-screws and directly coupled with servo motors.

ii) The machine is provided with an Automatic Tool Changer (ATC) system. The tool change mechanism is pneumatically operated. The ATC is equipped with Random Selection System feature that allows it to take the shortest path to access the required tool hence saving time in tool change.

iii) Its main vertical spindle has infinitely variable speeds from 180 to 6,000 r.p.m. It can fit the requirement of various materials and working conditions. Moreover the spindle is provided with an orientation mechanism that allows both clockwise and counterclockwise rotations.

iv) The three axes (X, Y and Z) are installed with ball screw to assure stable head, table and saddle movements. It has elastic cone covers and auto-cycle lubricator.

v) Guide ways of the saddle and spindle head are faced with "Turcite B". The column and bed are hardened, ground, and are automatically lubricated to minimize the friction resistance and stick slip.

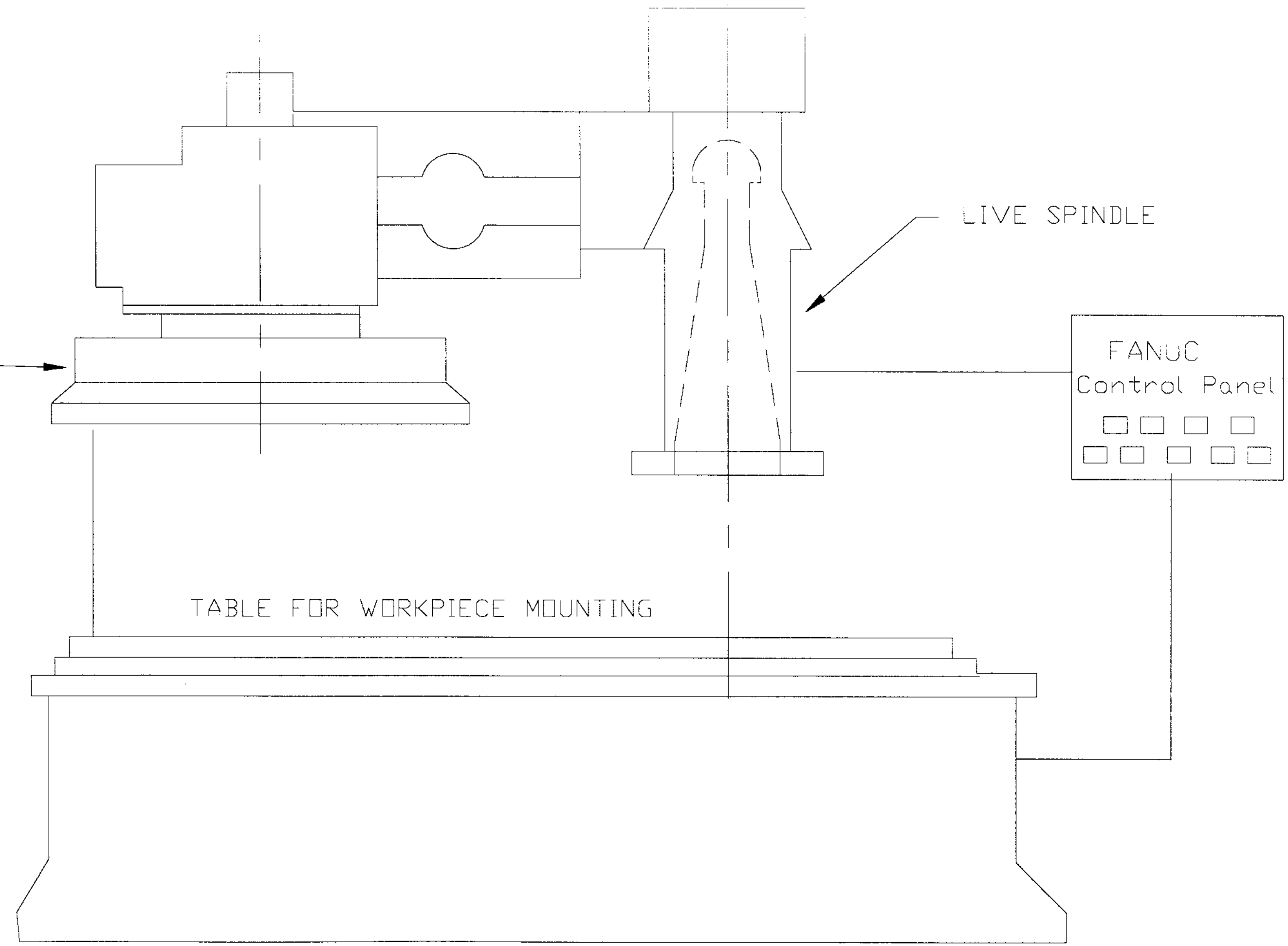
vi) The machine is provided with an automatic controlled coolant system for use during heavy duty cutting operations.

vii) This machine uses a pendent operator panel that can operate together with a computer and machine. A flash lamp connected to the control panel gives a warning signal in case of any danger in the machining process or at the end of each machining operation.

viii) The main spindle is installed with high accuracy and high-speed angular contact ball bearings. The spindle head assembly is shown in Figure 7.

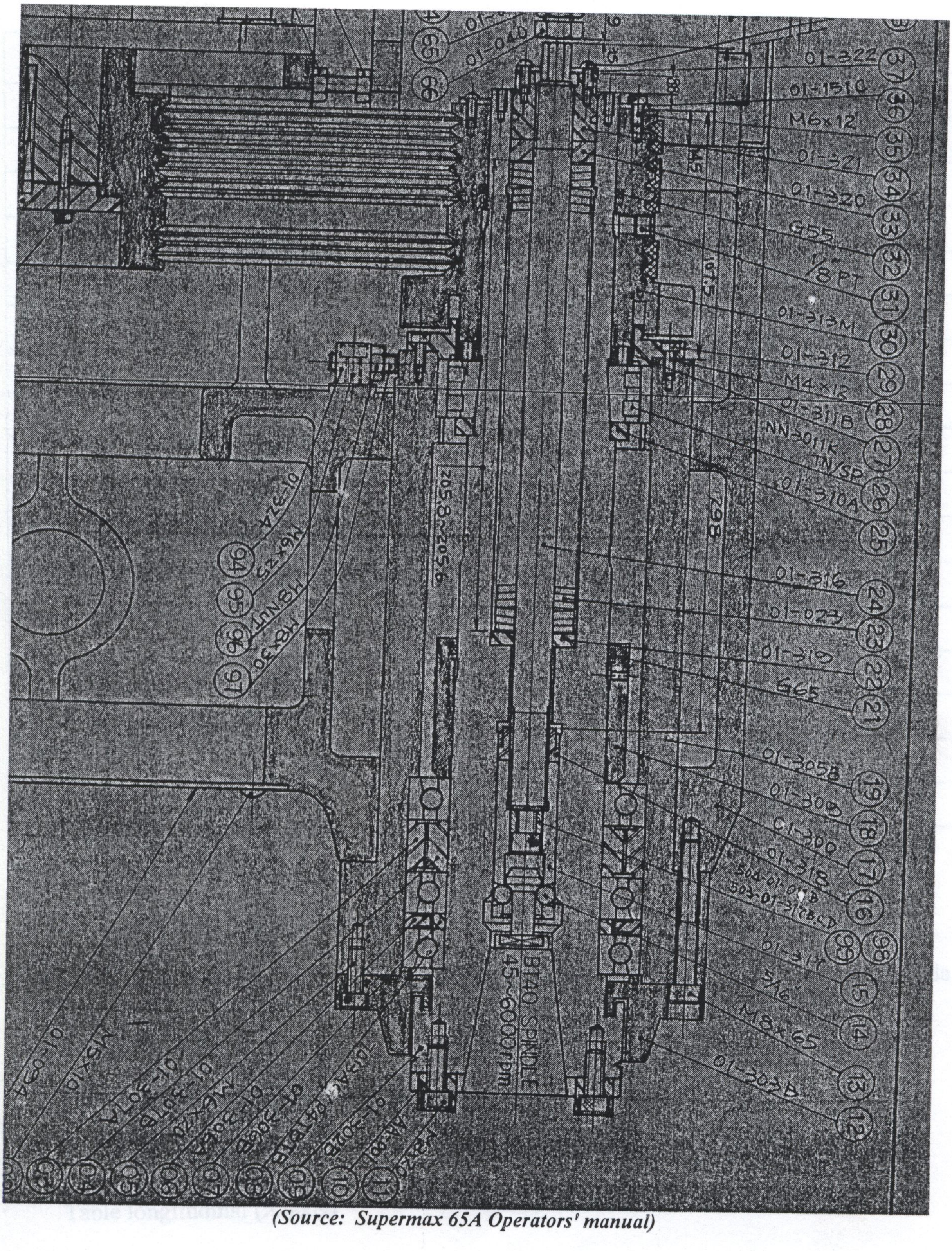
ix) The machine rapid feed rate can reach 12m/min.

Figure 6 SUPERVAX 65A LAYOUT



BASE PLATE		
PAPER SIZE: A4	DRAWN BY: Mwanza	DRAWING No
SCALE: 1 1	CHECKED:	
MATERIAL:	DATE:	BELONGS TO:

Figure 7: Supermax 65A Spindle Head Assembly



(Source: Supermax 65A Operators' manual)

List of Important Parts on Spindle Head Assembly (Figure 7)

ITEM No.	PART NAME	QUANTITY	DESCRIPTION
01	Cover	1	
03	Outer Ring	1	
04	Inner Ring	1	
08	Bearing	1 set	7013A C/P4 TBTB
09	Spindle	1	BT 40
10	Arbor Driving Key	2	
11	HSH Cap Screw	2	M8 x 20
12	Bearing Cover	1	
14	Steel Ball	5	5 / 16”
15	Ball Retainer	1	
17	Spindle Head	1	
22	Washer (6T)	1	
23	Disk Spring	93	
24	Drawing Bar	1	
30	Spindle Pulley	1	
31	Oil Plug	1	1/8” – 28 PT
32	O-Ring	1	G55

3.3 MACHINE SPECIFICATIONS.

The Supermax 65A machine tool operates within certain specified parameters that include the following [**10**]:

1. ***Maximum Machining Capacity*** (Length x Width x Height) 600 x 400 x 460mm

2. ***Range of Movement***

   Table longitudinal (X-axis) 600mm

Table cross (Y-axis)	400mm
Spindle vertical (Z-axis)	460mm
3. Table	1160 x 500mm
Table working size (length x width)	920 x 320mm
T-slots (width x distance x number)	18 x 100 x 3mm
Maximum workpiece weight	350kg
4. Spindle	
Spindle speed	45 - 6000 rpm
Spindle nose (standard)	ISO. No. 40 7/24 taper
OPTION	
(A) 503 - 10 Lo-Hi Gear Unit	Lo: 1200 - 35 rpm Hi: 6000 - 1201 rpm
(B) Spindle nose	ISO. No. 35 7/24 taper
5. Feed rate	
Jog feed rate	0 - 2400 mm/min
Cutting feed rate	1 - 3000 mm/min
Rapid traverse (X, Y-axis)	12 m/min
Rapid traverse (Z-axis)	8 m/min
Minimum increment (Basic Length Unit = ***BLU***)	0.001 mm
6. Automatic Tool change (ATC)	
Tool capacity *(OPTION)*	16 *(22)*
Tool selection system *(OPTION)*	Random/shortest path *(Technical memory random)*

Maximum tool weight	6 kg /piece
Tool change time *(OPTION 22 Tools)*	6 sec. *(3 sec.)*
Tool dimension *(OPTION)*	ISO. No. 40 *(ISO. No. 35)*
Maximum diameter x length *(OPTION)*	ø 76.2 x 300mm *(ø 80 x 300mm)*

7. ***Motors (1 Hp = 746 W)***

Spindle drive *(Standard)*	AC 5.5kw continuous rating
	(AC 7.5kw - 30 min. rating)
Spindle drive *(OPTION)*	AC 3.7kw continuous rating
	(AC 5.5kw - 30 min. rating)
Spindle fan motor	AC 0.4kw
Feed shaft drive	FANUC Model: 5S (X, Y, Z)
ATC motor	AC 40W
Coolant pump drive	AC 0.81kw
Lubrication oil pump drive	AC 10W

8. ***Power requirement*** AC 200/220V + 10% 50/60 Hz

13 KVA

CHAPTER FOUR

PROPOSED DESIGN MODIFICATIONS.

4.1 MODIFICATION CONCEPTS

The features of the Supermax 65A Vertical Machining Centre and its controls outlined in chapter three were studied with the aim of modifying them to allow for additional machining facilities for turning. The additional machining operations were to allow for production of external and internal turned features such as radii, grooves, tapers and recesses. The most important design parameters that were considered (with equal weight) included:

i) ***Allowable spindle motor power.***

ii) ***Consideration for cutting forces during anticipated change in direction and magnitude of cutting forces.***

iii) ***Stress considerations on spindle bearings.***

From the set-out objectives in section 1.4, the similarities discussed in section 2.5, the design construction of Supermax 65A Vertical Machining Centre (Figure 6 and Figure 7) and from the general design of the latest combined turning and milling machines, three most probable design modifications were considered. While the Supermax 65A has restricted 3-axes movement, most of the latest combined turning and milling machines exploit the optional advantages of using more than 3-axes of movement. The concepts investigated during the research were not in any way exhaustive, but formed the major fundamental ideas exploited by machine builders in coming up with combined turning and milling centres. The major design consideration was the fact that the Supermax 65A was a 3-axes machine.

4.1.1 Included Horizontal Spindle Concept

Two spindles providing for machining operations in the vertical and horizontal planes (Figure 8). This required an additional horizontal spindle (headstock) and tailstock along the existing table slides for turning operations. The existing vertical spindle was to retain its movements along Z-axis, while its rotation motion was demobilized and be used as tool-post for carrying a turning tool. This required that an additional horizontal spindle with complete headstock and own motor be designed and constructed or procured. This additional spindle would carry the workpiece during turning operations.

The following factors restricted the design modification concept proposed in 4.1.1:

i) The requirement for space at the bottom of table to hold additional motor to drive the horizontal spindle could have necessitated some construction changes to the existing table assembly.

ii) An additional spindle motor would require to be supported by bearings hence increase in cost.

iii) The construction of the horizontal spindle restricted operations to external turning only.

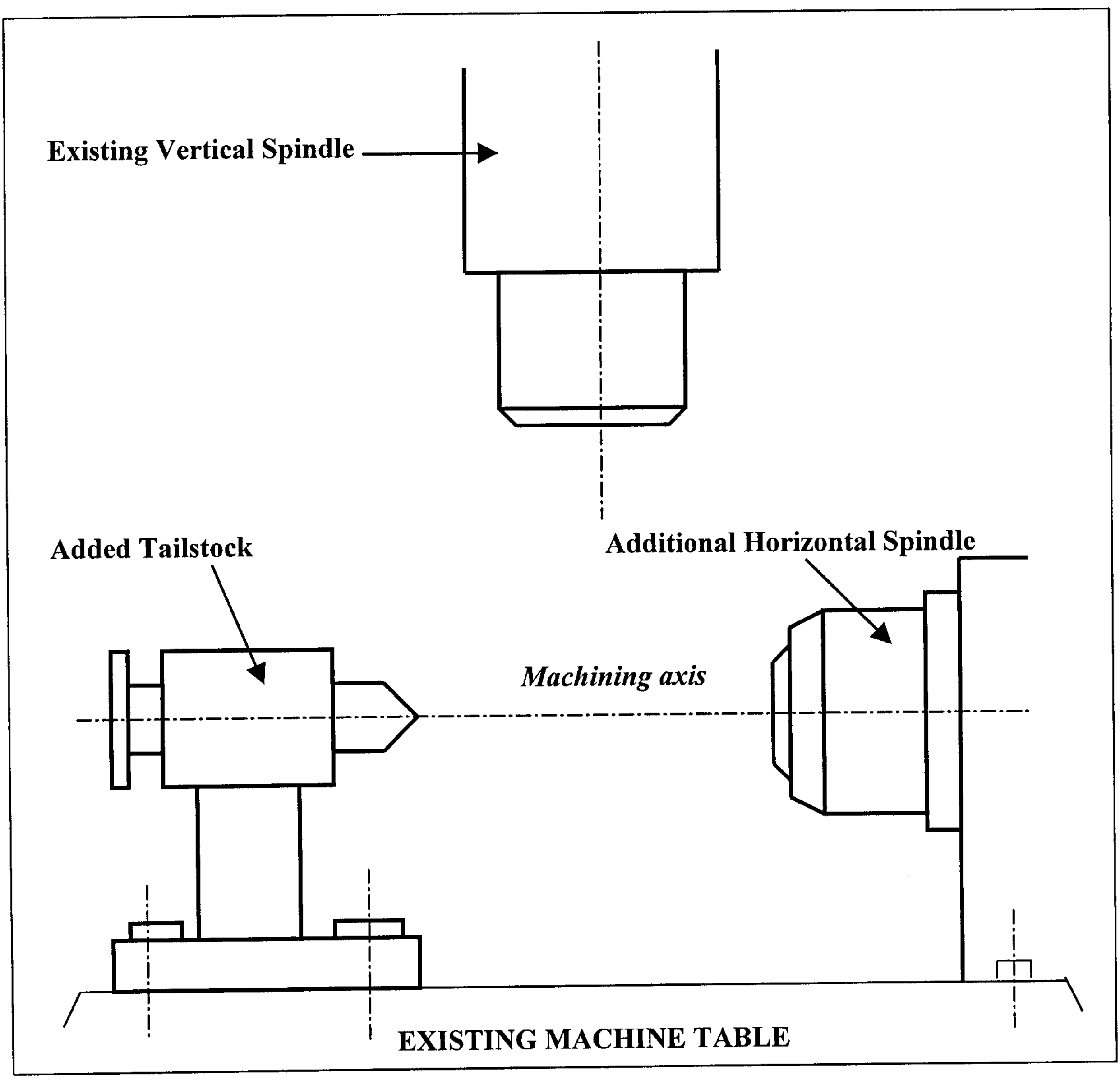
iv) In single-point cutting process, the workpiece rotates while the tool is fixed in position. The construction of the VMC spindle and its controls only allowed locking of rotation motion at tool change position after which it retained its free rotation, hence rendering single-point tool holding impossible.

The advantages of this concept were that:

i) It could allow turning between centres and hence avoiding any deflection of the workpiece during machining.

ii) Much longer turned profiles could be done with this concept, taking advantage of the full length of the existing table size.

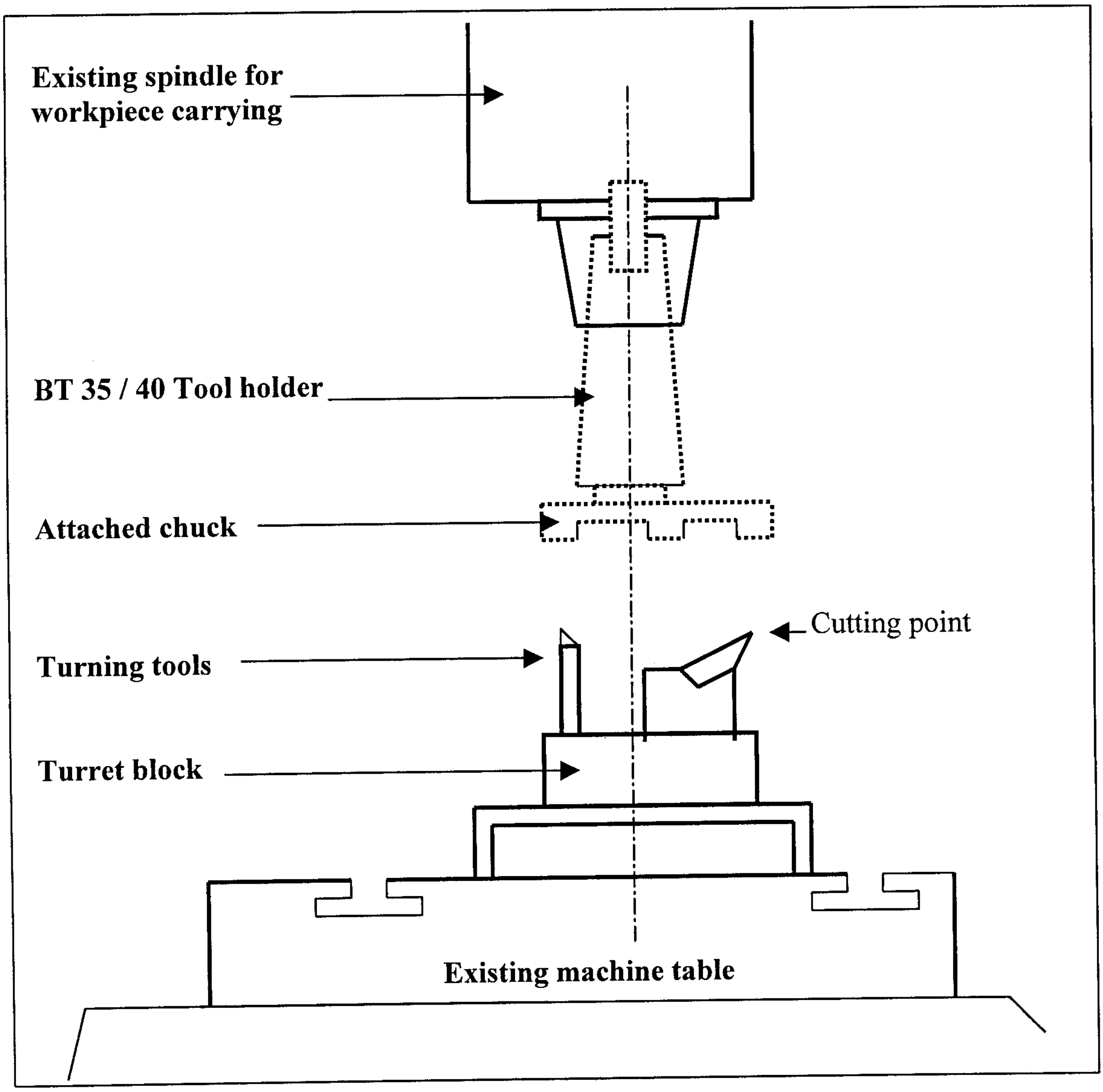
Figure 8: Additional Horizontal Spindle



4.1.2 Indexable Horizontal Turret Concept

An indexable and replaceable horizontal turret to be carried on compound and cross slides (Figure 9). The slides were to be fixed on the existing VMC table. The existing 'live' vertical spindle was to be provided with a specially designed attachment to allow for workpiece holding. The workpiece would be manually loaded onto the proposed chucking fixture.

Figure 9: Indexable Turret



The following factors restricted the design modification concept proposed in 4.1.2:

i) The total weight of the workpiece and the holding fixture should be 6 kg or less. This conformed to the Supermax 65A design specifications for maximum weight of the tool. Therefore, considerations were to be made on the effect of added weight on the spindle thrust bearings and disc springs, which might cause the temperature inside the spindle drive motor to increase due to overload. The spindle motor alarm would go ***ON*** when the temperature rises.

ii) The proposed loading of workpiece could result in deflections. This would become considerable at low spindle speeds, low feedrate, larger depth of cut and unsatisfactory locations of workpiece in chuck.

iii) Machining of slender pieces would require additional support to avoid any deflection during turning.

The advantages of using this concept include:

i) Its simplicity and easy to construct fixtures.

ii) Certain standard tool holders could be used for holding smaller diameter workpieces hence avoiding any need for purchasing additional components.

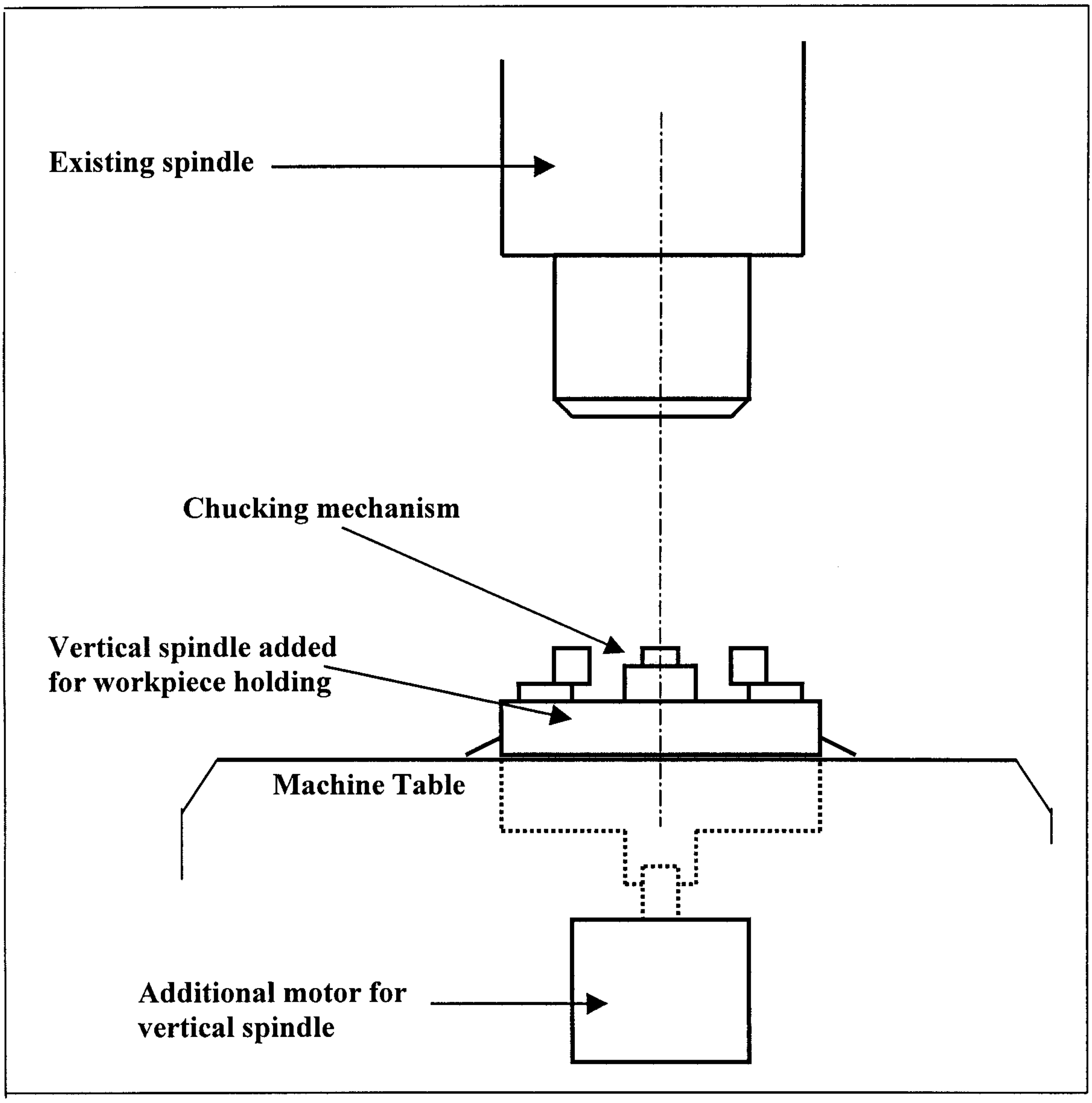
iii) Controlling movements of workpiece and tool could easily be simulated from the existing controls.

iv) There would be no need for a locking mechanism for the 'live' spindle since free rotation was desirable for turning operations.

4.1.3 Included Vertical Spindle Concept

An additional vertical spindle with complete headstock was considered (Figure 10). A coupling to added drive motor was proposed.

Figure 10: Additional Vertical Spindle



The following factors restricted the design modification concept proposed in 4.1.3:

i) The proposed additional vertical spindle would require a motor with much higher power to support both the weights of the spindle, headstock and workpiece.

ii) The construction of the VMC spindle and its controls only allowed locking of rotation motion at tool change position after which it retained its free rotation, hence rendering single-point tool holding impossible.

iii) The additional spindle would require a separate servo motor which would have to be incorporated into the existing control system.

The main advantage of using this concept was that vertical turning operations (internal and external features) could easily be done within the existing 3-axes movements.

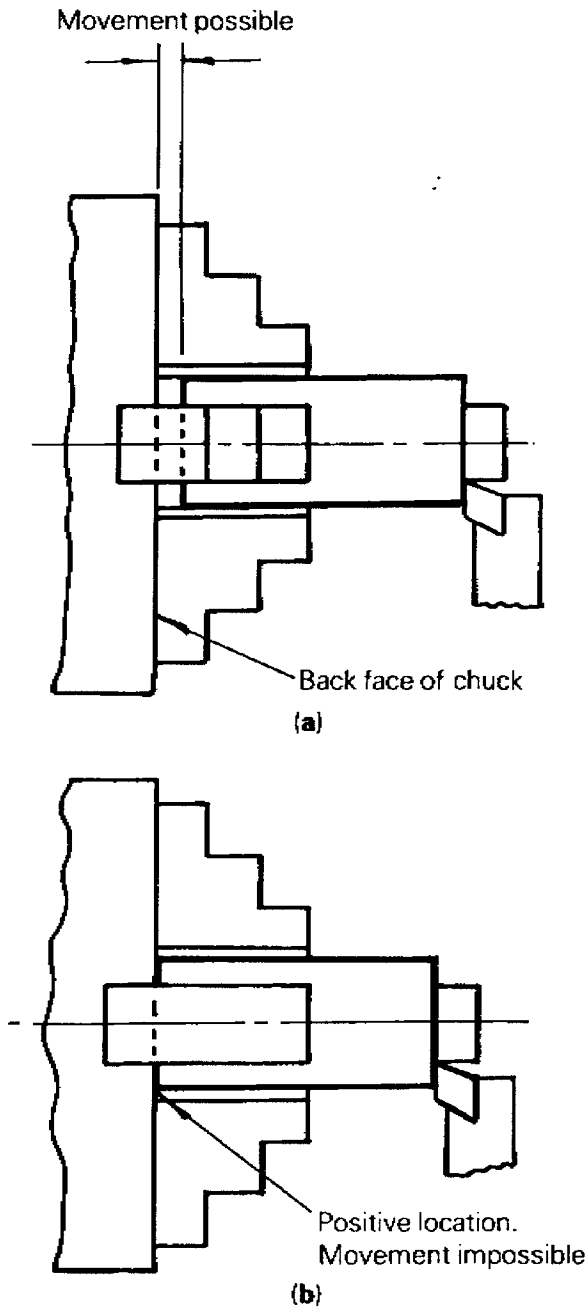
4.2 DETAILED DESIGN OF SUITABLE MODIFICATION CONCEPT

It should be noted that the main design criteria were to have as far as possible minimum and less costly modifications to the existing VMC. From the analysis done on each of the proposed design concepts individually and comparatively, it was apparent that the concept in 4.1.2 would be the most suitable design modification. This concept required the use of the existing spindle as a workpiece carrier and a table mounted tool turret. The detailed design fixtures for this particular concept were therefore investigated further.

4.2.1. Application of common work-holding devices

It is an established working practice that, wherever possible, work should be positively located (Figure 11). This requires that it be positioned in such a way that when the cutting forces are applied there should be no possibility of movements taking place.

Figure 11: Positive Location in Turning Operations [7]



Positive location of a turned component **(a)** *unsatisfactory* **(b)** *satisfactory.*

In any machining process the possibility of movement of the workpiece is unacceptable for safety reasons and dimensional control. In NC machining processes there is also the problem that movement, however slight, means a loss of dimensional accuracy **[11]**. The basic requirements for any work-holding device are therefore that it must:

i) Securely hold the work as any movements may result in workpiece breaking away from the chuck and result in wrong final workpiece dimensions or possibly cause injury to the operator.

ii) Be able to provide positive location against applied tool cutting forces. This ensures that there are no noticeable or unnoticeable deflections in workpiece while machining is in progress.

iii) Be quick and easy to operate. As much as possible less time should be spent on placing and removing workpiece on the chucking mechanism, this reduces production time considerably, hence the cost of production.

There are a variety of devices in general use that have been tried and tested in conventional machining situations. Chucks, collets and vices are obvious examples, and these are also used on numerically controlled machines. Work-holding devices such as these may be mechanical, pneumatic or hydraulic in operation. If the components were not precisely positioned in relation to assumed program datums, then the profile features required would not be achieved.

4.2.2 Work Support for Turning Modification

The maximum diameter and length to be turned was to conform to the size of the work-holding fixture and available space (Z-axis) between tip of tool and workpiece location points. The work-holding fixture designed included a standard ***4-Independent Jaw Chuck 152-mm diameter, Model 34 - UK, which weighed 2.6 kg***. This particular chuck was the most suitable available (on the workshop shelf) during the design and fabrication stages of the research project. Further support was however needed for coupling to the existing standard tool holders (***BT 40 taper***). The choice of this particular chuck was based on the requirement for allowable design weight (less than 6-kg) for the chucking mechanism. The maximum allowable diameter and length of workpiece for the design were 100mm and 250mm respectively.

One standard tool holder (***Fournel/401290 - Face mill tool holder***) that could effectively and easily be assembled to the chosen chuck was selected. The coupling plate (Figure 12) was designed and produced from 100-mm diameter mild steel billet. The plate was connected using four M8 x 40 bolts (**shear strength = 600 N/mm^2**) as provided for in the chuck. The total weight of the fixture was 4.2 kg. The weight of the workpieces was limited to 1.8 kg (**6 kg minus 4.2 kg**). The complete assembly of the work holding mechanism is shown in Figure 13. The shear force was assumed distributed equally on the four bolts. Most of the profiles that were to be turned were shorter due to the requirement for the spindle to move up and down during turning operations hence further support on the free end was not required. For much slender pieces, support could be achieved through the use of any suitable steady.

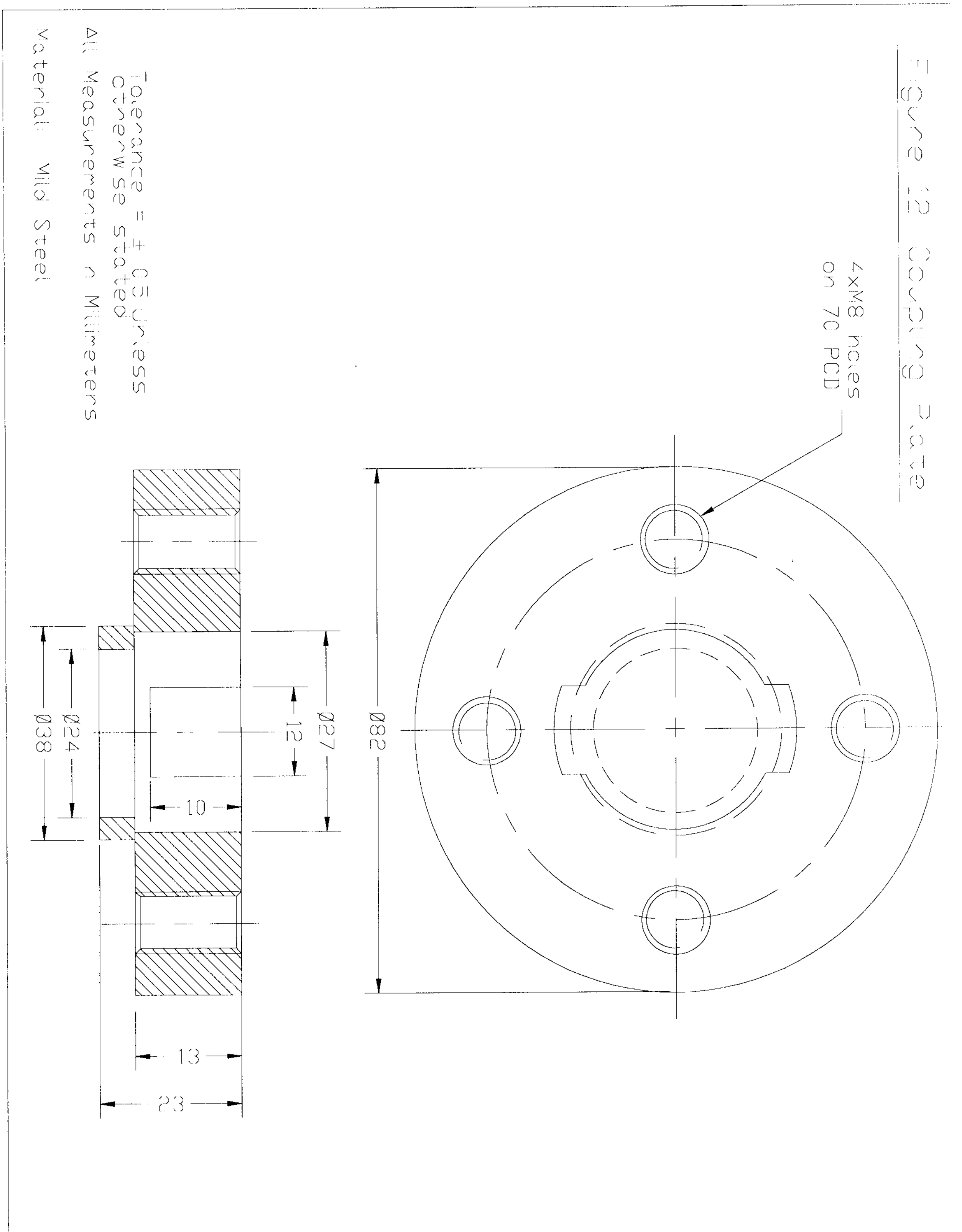
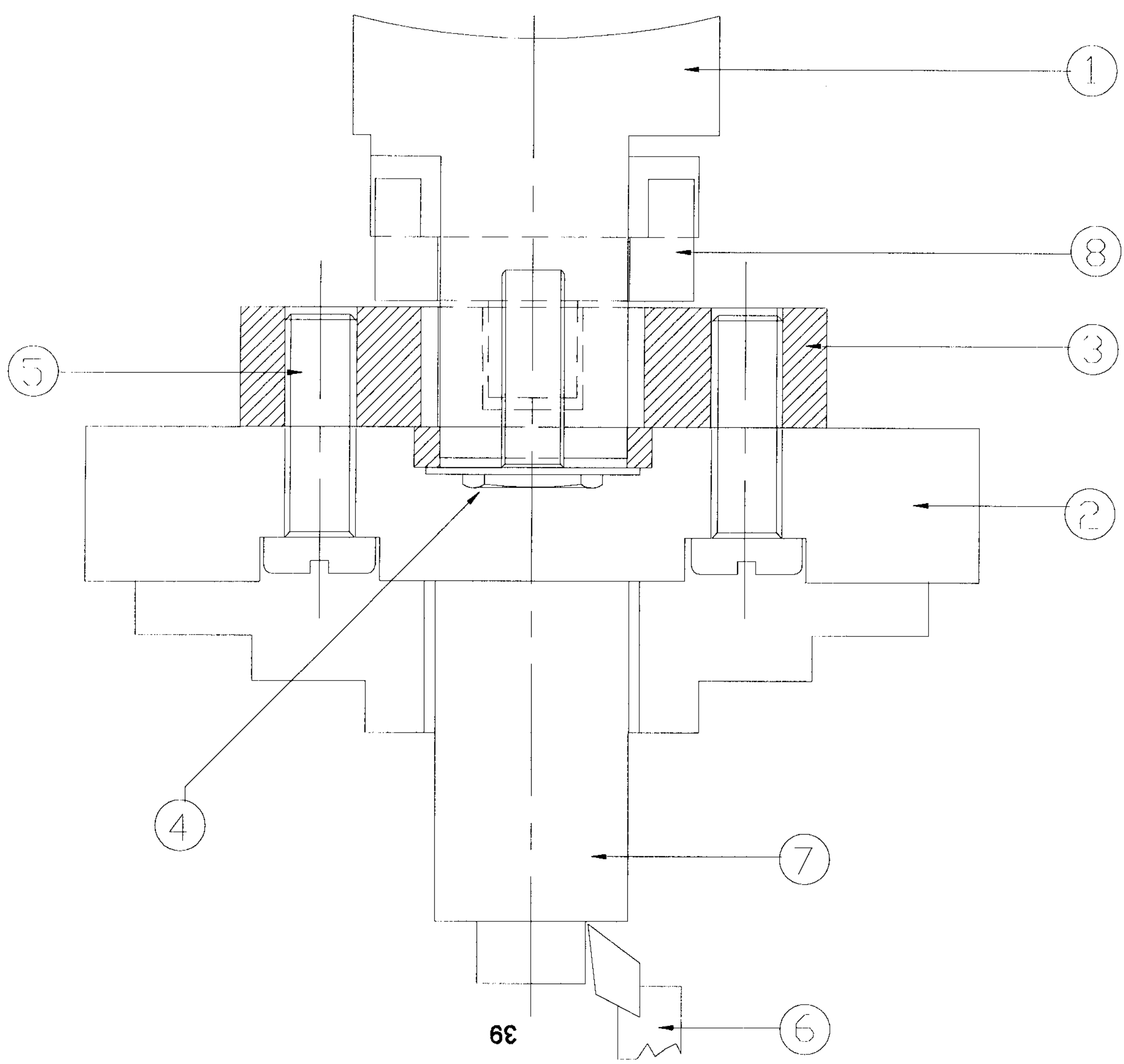


Figure 12 Coupling Plate

Figure 13: Assembly of Work Holding Fixture

PART NUMBER	NAME
1.	Tool Holder
2.	4-Jaw Chuck
3.	Coupling Plate
4.	Locking Bolt
5.	Four M8x40 Bolts
6.	Cutting Tool
7.	Workpiece
8.	Base Plate

Total Length = 236 mm



4.2.3 Turret and Tool Holder Arrangements for Turning

There are several types of turret arrangements that are used for turning operations. Some of these arrangements depend on the construction of the machine tool bed and the spindle orientation. For the proposed modification and the required turning operations (facing, external and internal turned features, parting-off), a suitable position of the turret was one with its axis along the direction of the existing spindle (existing Z-axis). This allowed for indexing on the existing table X-Y plane.

Many different tool holder designs are in use depending on the design of the turret arrangement. Their designs largely depend on the type of tool shank being used. A typical tooling system for turning centre is given in Figure 14.

The design for turret and tool holder fixture, for this research work is shown in Figure 15. It consisted of main block (Figure 16) and tool holder with section along XZ plane of turning tool in position (Figure 17). The size of the tooling assembly was determined by the existing table size (Figure 18). As an initial investigation, it was proposed to start with a single tool on the turret. However, the final tool holder fixture used during this study had provisions for carrying two horizontally located tools hence horizontal slots. For internal turning operations, vertical location of tools is essential and could be achieved using a similar size of tool holder with vertical slots. Suffice to mention here that the final design of the tool holder would depend on the turning operations to be carried out. Considerations could also be made on the use of suitable tool holders that can carry insert shapes for various turning operations.

Figure 14: Tooling System for turning [7]

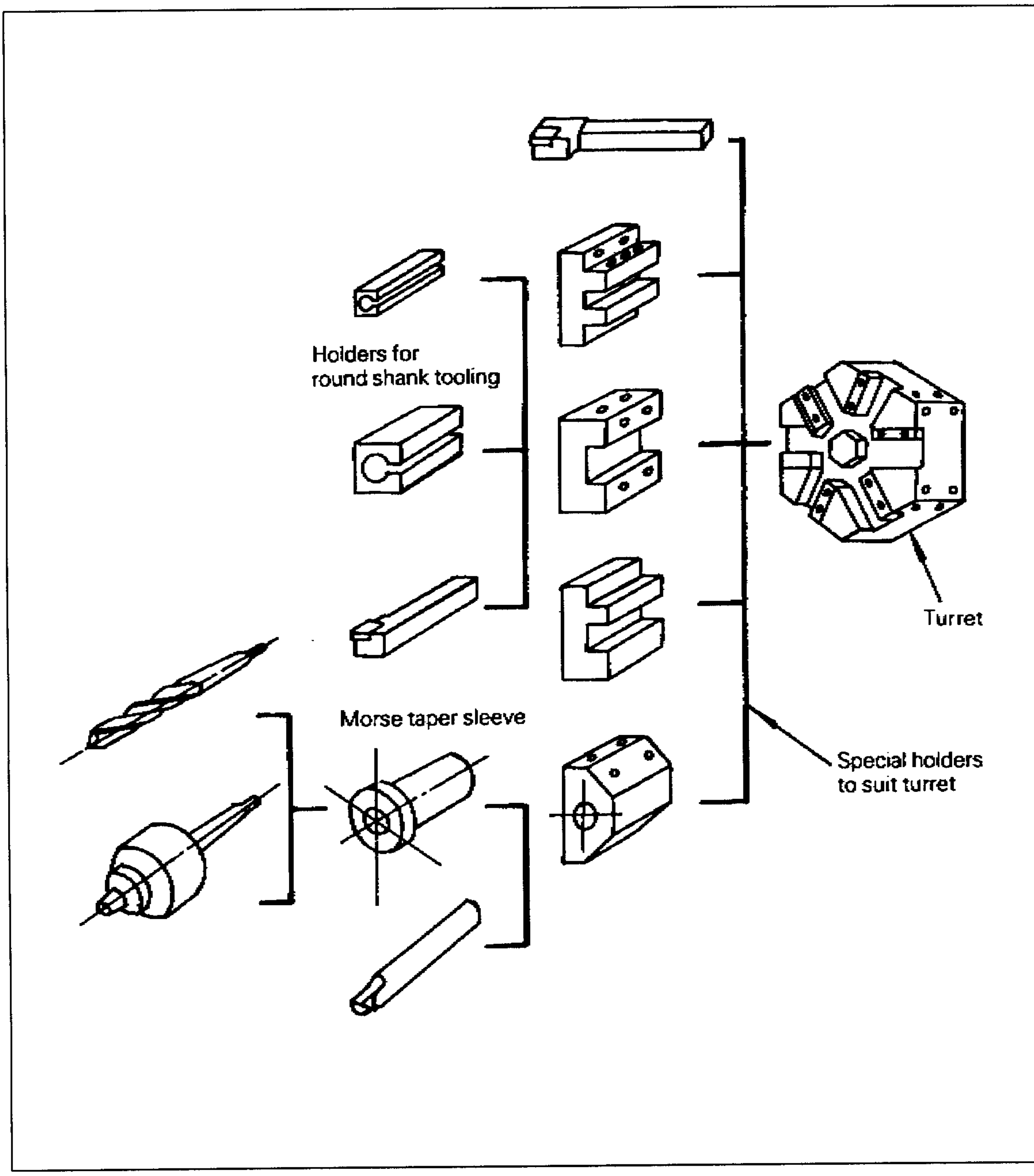
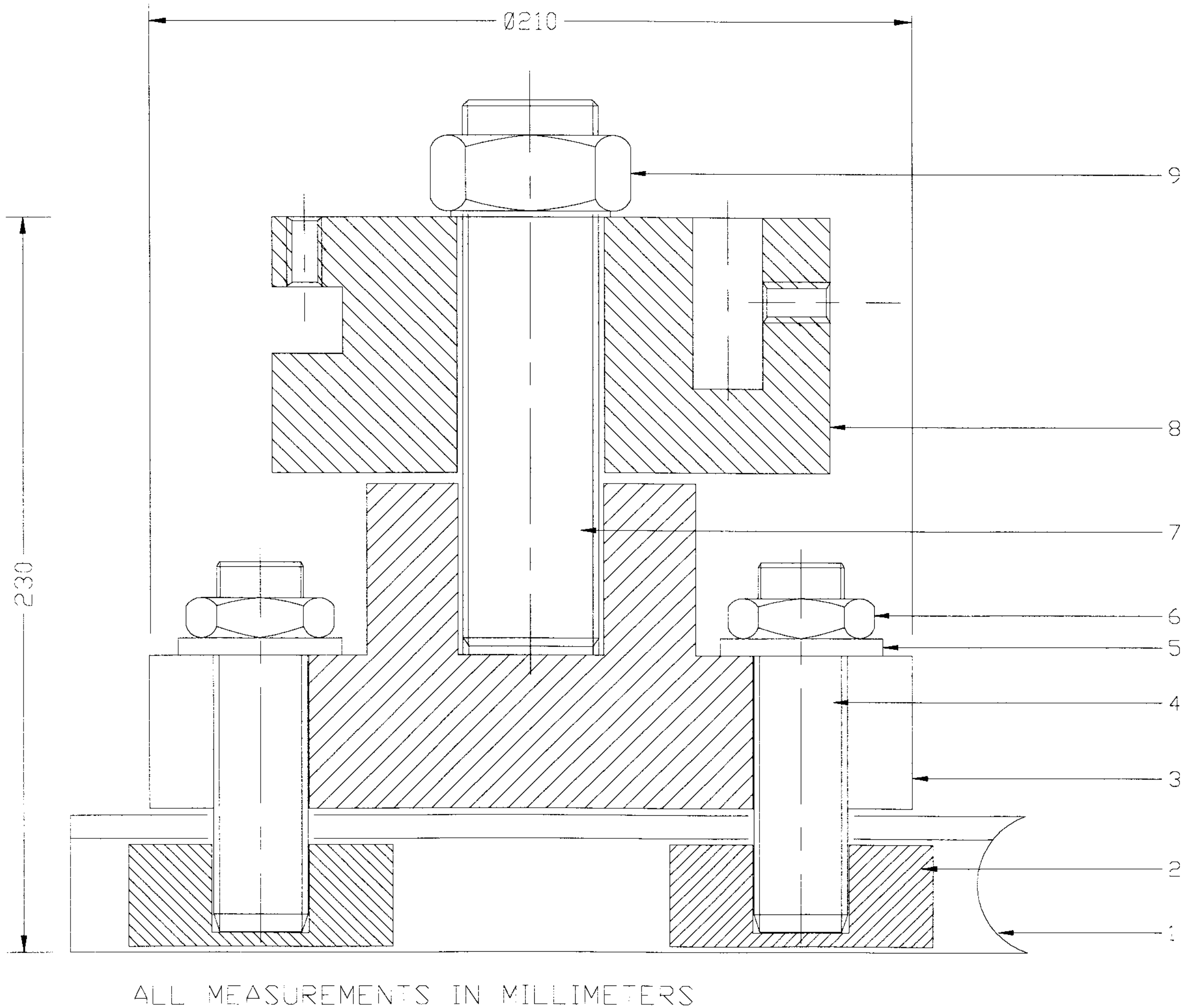
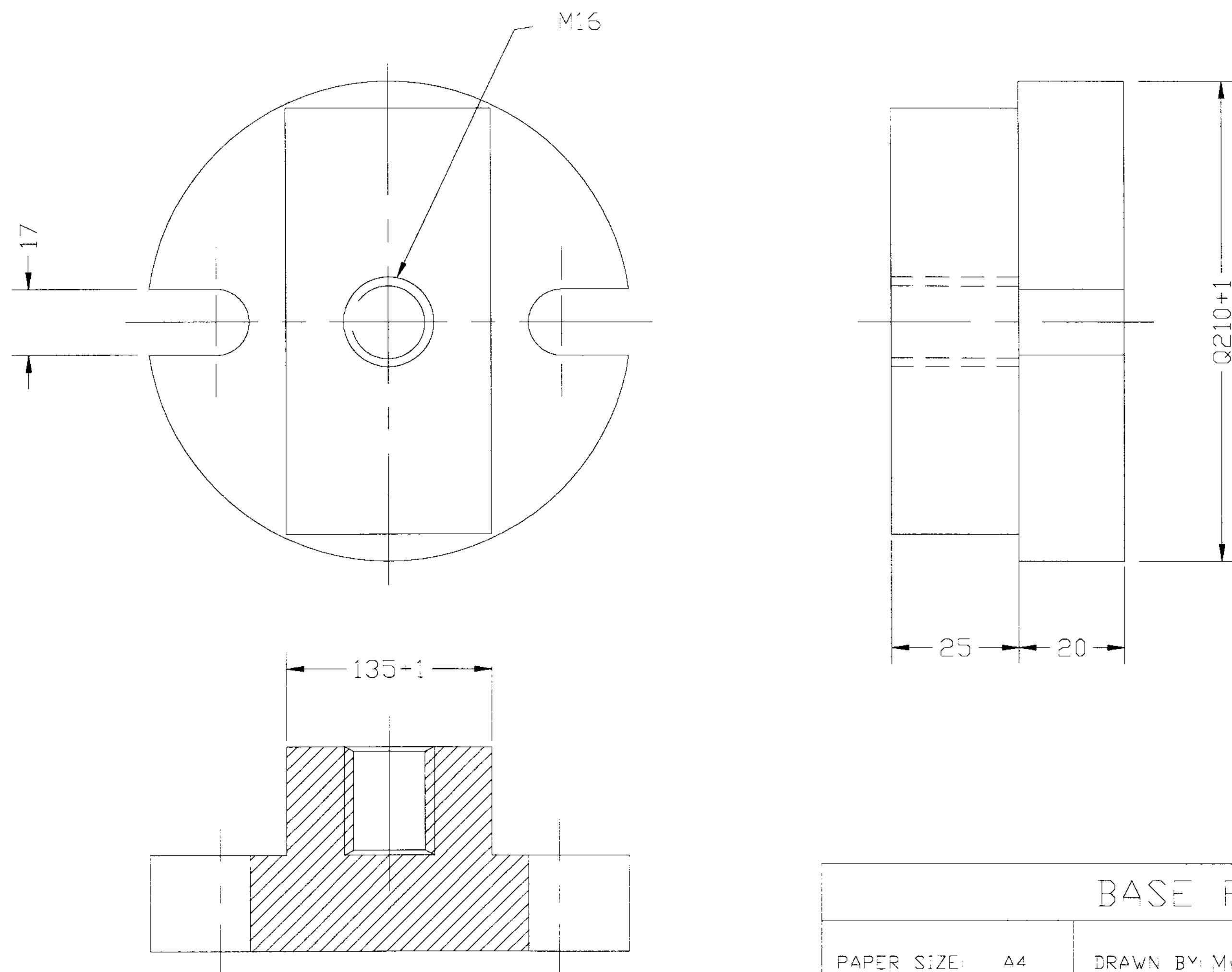


Figure 15 Main Block and Toolholder Assembly



PART No.	NAME:
1	Table T-Slot
2	Locking Block
3	Turret
4	M16 X 40 Bolt
5	Spring Washer
6	Locking Nut
7	M16 Bolt
8	Tool Holder
9	Locking Nut

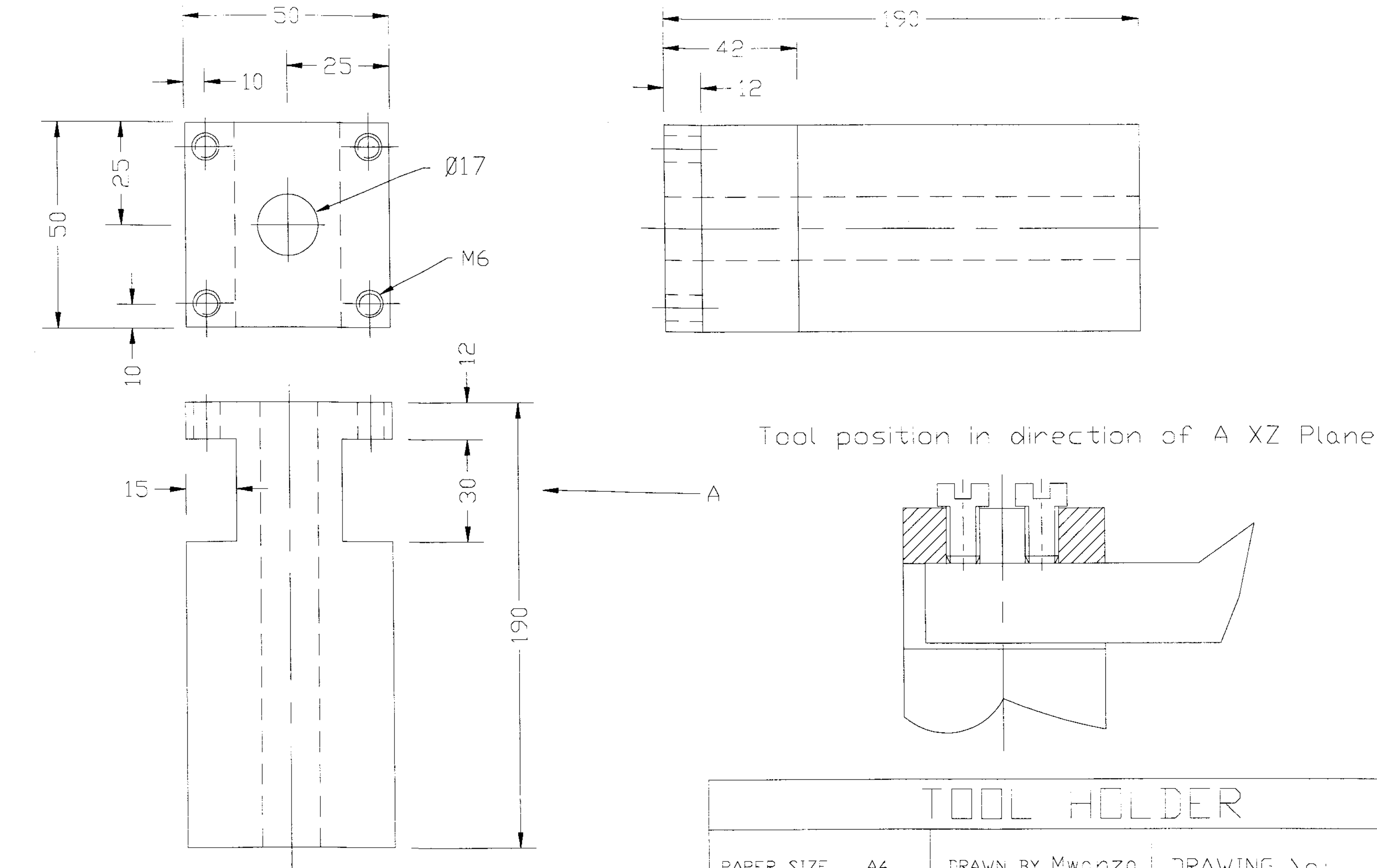
Figure 16 Main Turret Block



All measurements in millimeters
Tolerarce ±0.3 unless otherwise stated

BASE PLATE		
PAPER SIZE: A4	DRAWN BY: Mwanza	DRAWING No.
SCALE: 1 : 1	CHECKED:	
MATERIAL: M.S	DATE:	BELONGS TO:

Figure 17 EXTERNAL TURNING AND FACING TOOL HOLDER DESIGN



All measurements in millimeters
Tolerance ±0.3 unless otherwise stated

TOOL HOLDER		
PAPER SIZE: A4	DRAWN BY Mwanza	DRAWING No.:
SCALE: 1 : 1	CHECKED:	
MATERIAL: M.S	DATE:	BELONGS TO:

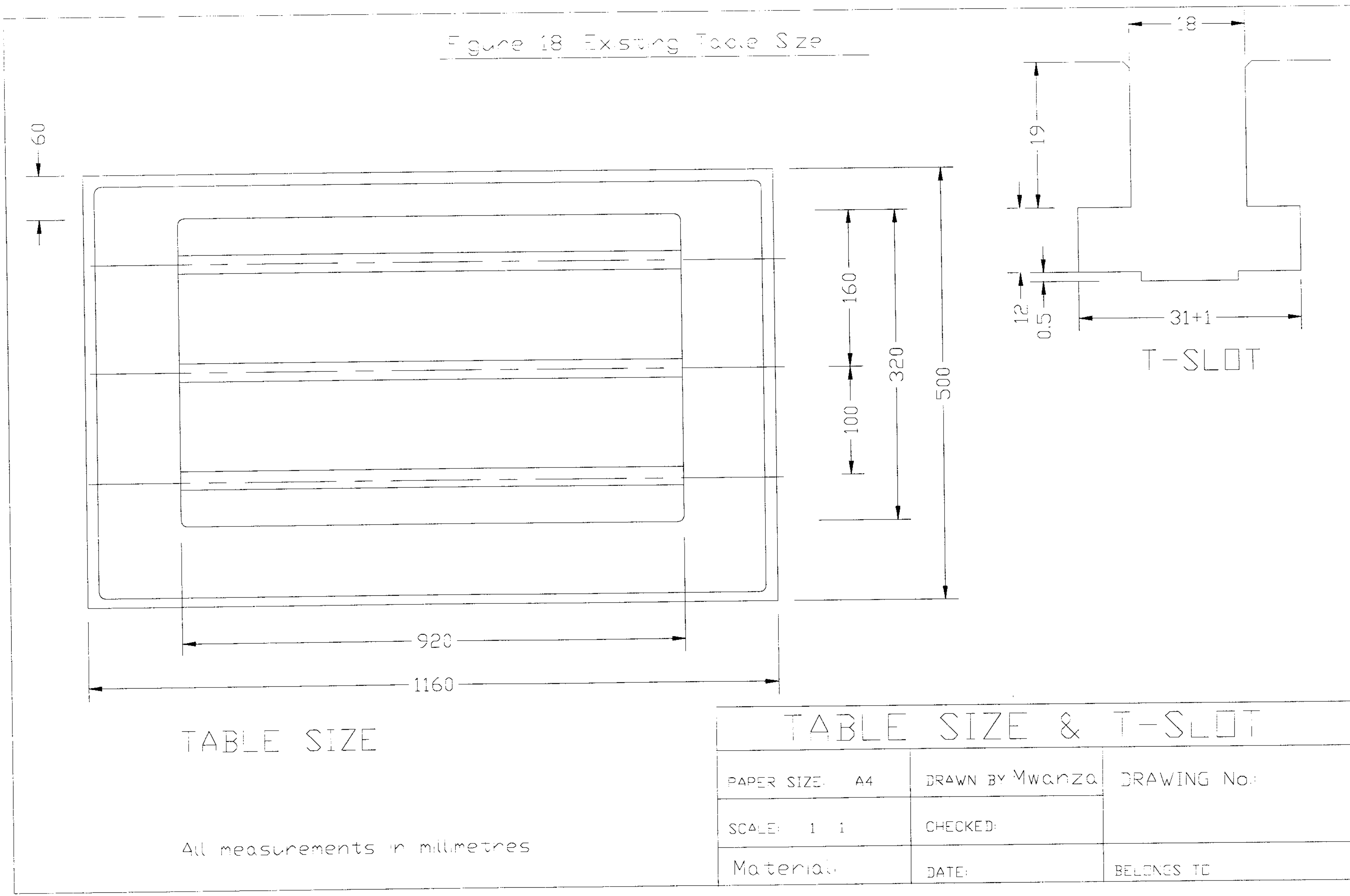


Figure 18 Existing Table Size

CHAPTER FIVE

PART PROGRAMMING FOR TURNING OPERATIONS

5.1 INTRODUCTION

Programming is determining the tool's positions with regard to the workpiece in codes. In programming, the technological aspects of the production process, machine tool capabilities, tool and workpiece are taken into account. When writing a CNC program the following aspects are considered:

i) Workpiece related data; this data can be read from a drawing and work instructions.

ii) Methods of production; type of machine tool, fixtures and tools available.

The programmer must have thorough knowledge with respect to:

i) Workpiece drawings.

ii) Sequence of machining.

iii) Metal removal technologies, e.g. milling, drilling, turning, grinding, etc.

iv) Workpiece holding techniques.

To make a working program, the programmer must answer the following questions:

i) Which method of production will be used?

ii) How will the workpiece be fixed?

iii) Which tools are going to be used?

iv) What are the technological data (e.g. cutting speed, feedrate, etc.)?

The starting point when planning to machine a workpiece is to carefully consider the initial dimensions of the material and determine the part that has to be cut to get the final workpiece profile. To produce a finished product as effective as possible, it is important to work out the best order of machining. There is need to take into account the time needed for tool changes and possible re-setting up of the workpiece in the clamping device.

Not until the stated requirements have been undertaken and the manner of machining is known should further process details, such as the technological data of the cutting process be determined. When all this information is properly arranged on a process sheet, programming can be started, which will now just be translating the worksheet into CNC-language.

The program is built in International Standards Organization code adopted from British Standards **(BS 3635),** written instructions. These instructions are read by the control system. A program is divided in lines and each line starts with a line-number. The sequence of words in a program is freely chosen; some control systems put the words in a certain order themselves. It is very important to program in the order of machining; it decreases the chance for mistakes, because the program is easily recognized.

An example of a program line is:

N10 G00 X35 Y10 Z12 F100 S250 T1 M3;

As can be seen, the information offered to the CNC system consists of characters and digits (***ALPHA-NUMERICAL***), all with their own information and meaning. This information can be divided into:

i) Preparatory Function (**G-codes**).

ii) Miscellaneous Functions (**M-codes**).

iii) Geometrical information (**X, Y, Z**).

iv) Technological information (**S, F, T**).

5.1.1 Preparatory Functions (G-Codes)

The meaning of the G-function is to prepare the machine to fulfil instructions. The most basic motion types are:

i) Rapid Speed (**G00**). This is for tool positioning, hence the tool will not go in a straight line to the instructed position, but it gets full axes speed to the position where it is instructed to move. It is not used in cutting.

ii) Linear Interpolation (**G01**). The tool will move in feed in a straight line to the instructed position. The value of the feed can be programmed.

iii) Circular Interpolation Clockwise (**G02**).

iv) Circular Interpolation Counter Clockwise (**G03**).

There are also G-codes reserved for cycles like boring, tapping, drilling, etc, motions. These additional motions are either one or a combination of the basic motion types. A complete list of G-codes is given in Appendix I.

5.1.2 Miscellaneous Functions (M-Codes)

These are assisting functions, they can also be considered as switch-functions. They manage things like switching **ON** or **OFF** the spindle, coolant, etc.

5.1.3 Geometrical Information (X, Y, Z)

These indicate the geometric directions where the tool has to move in the **X, Y, Z** cubic space. Positions are defined by **X, Y, Z** coordinates.

5.1.4 Feed Function

The feed function controls the cutting feedrate of the tool. The cutting feedrate can be given in mm/min (**G94**) or mm/rev (**G95**).

5.1.5 Speed Function

The spindle speed can be controlled by specifying a value following address **S**. It is possible to give a constant cutting speed (**G96**) or a constant spindle speed (**G97**).

5.1.6 Tool Function

The tool address (**T**) is provided with a value to indicate the tool station, which is going to be used.

5.2 METHODS OF PROGRAMMING

Computer Numerical Control programming can be done directly at the machine or somewhere else behind a computer. Programming at the machine (online) can be done by G-code programming or by conversational programming. Off-line programming can be done with a word-processor or Computer Aided Design / Computer Aided Manufacturing (CAD/CAM) software.

5.2.1 Online Programming

G-code programming on the machine can only be meaningful when it concerns simple products with a minimum of calculations. The more modern CNC-machine tools offer the possibility to simulate the cutting process graphically on the monitor. In this way the machinist can easily check the program.

The main advantages of G-code programming directly at the machine are:

i) It is easier to bring little changes into the program and the production process is more flexible.

ii) There is no need for a different department to make the worksheet and the program.

The main disadvantages of G-code programming at the machine are that:

i) During the programming, it is (at most machines) not possible to run the machine.

ii) There is need for very skilled operators.

5.2.2 Conversational Programming

With conversational programming, the operator has to define a pre-defined cycle. After a cycle is selected, the control system then offers a menu of options. By confirming to the right options, the geometrical and technological information necessary for machining is given into the control system. With this information the machine tool is able to machine the workpiece.

The advantage of conversational programming is that it is impossible to forget an address. However, it is possible to make a mistake.

For a skilled programmer, conversational programming is rather redundant and it takes a lot of time. This way of programming is hardly being used nowadays due to the increased use of the CAD/CAM software.

5.2.3 External Programming on a Word Processor

In this method of programming a program is made on an external computer and when it is finished, it is transferred to the machine by Direct Numerical Control (DNC).

The advantages of this method of programming are that:

i) There is no stand still of the machine during programming.

ii) It is easy to store the programs for repeated machining.

The main disadvantage is that you still have to calculate all the points of the contour.

5.2.4 External Programming with CAD/CAM Software

With CAD/CAM the geometry of the workpiece is drawn with CAD. When the geometry is finished, the toolpaths are selected and the technological data can be given into the computer. The software generates a program in its own language. A post-processor translates this language into a recognizable code for the concerned machine tool.

The advantages of CAD/CAM programming are that:

i) The technological and geometrical information is general. With the right post-processor, the program can be made to suit a particular control system.

ii) Complex workpieces are easy to program.

iii) The software calculates the machining time.

iv) Long programs can be produced quickly unlike in the manual case.

The main disadvantage of CAD/CAM programming is that the software is quite expensive for small-scale use.

5.3 EXISTING MACHINE CONTROLS AND PROGRAMMING SOFTWARE

All turning machine tools (vertical or horizontal) consist of two basic components; spindle for workpiece holding and rotation, and tool-holding mechanism for feeds. The existing operational components of the CNC Supermax 65A Vertical Machining Centre were adopted to include turning operations. The design concept considered the machine tool parameters; spindle motor power, cutting force and bearing loading. The added modifications were done with the view to operate within the existing machine tool-programming, using available CAD/CAM software and manual G-code programming. This required the adoption of milling programming features for carrying out turning operations.

The existing Supermax 65A has ***Fanuc*** control system that operates with ***Mastercam Mill*** software. Mastercam is an integrated Computer Aided Design / Computer Aided Manufacturing (CAD/CAM) software package, which enables a programmer to make geometric drawing of the workpiece, specify a drawing, generate complete workpiece drawings, show graphically toolpaths and generate a Numerical Control File (NCI file). Mastercam-Design is the CAD part of the total program [**12**]. By this module, a 3-D geometry could be constructed with the help of points, lines, arcs, fillets, splines, curves, surfaces and rectangles. By the CAM module, actual machining sequence could be simulated and controlled. Therefore, Mastercam Mill software enables a programmer to carry out Computer Aided Part Programming (CAPP) for milling operations on the existing VMC machine tool. It basically offers a way of doing 'remote' (away from the machine tool) design of a part, simulate machining and control the machining sequence using a computer. This software was meant for programming milling related operations on the existing VMC, in which the final profile of the workpiece forms the CAD drawing. The CAD drawing is then post-processed into actual machining sequence, assuming movement of the cutting tool. In the proposed turning modification, the final profile was to be made on the assumed turning tool (fixed into the spindle), by a 'fixed' point (tip of cutting tool). In this case, the material was assumed to do actual cutting to get desired profile, a situation similar to drilling feature in Mastercam Mill. In a typical drilling cycle, the desired hole made in a given workpiece, is determined by the size of drill, i.e., the drill makes the final shape of the hole.

Since machining of workpiece always takes place at a surface or contour, it was obvious that Mastercam Mill (from the viewpoint of these machining technologies) pays a lot of attention to the definition of surfaces. This means that the designer has to make a part profile on a material, which then was post-processed to the required sequence of machining the profile. There are many different versions of Mastercam that are being used for machining sequence programming on different processors for controlling machine tools. The research attempted to employ the existing milling software and manual G-code programming (at machine) to program turning sequences.

5.4 PROGRAMMING FOR TURNING OPERATIONS

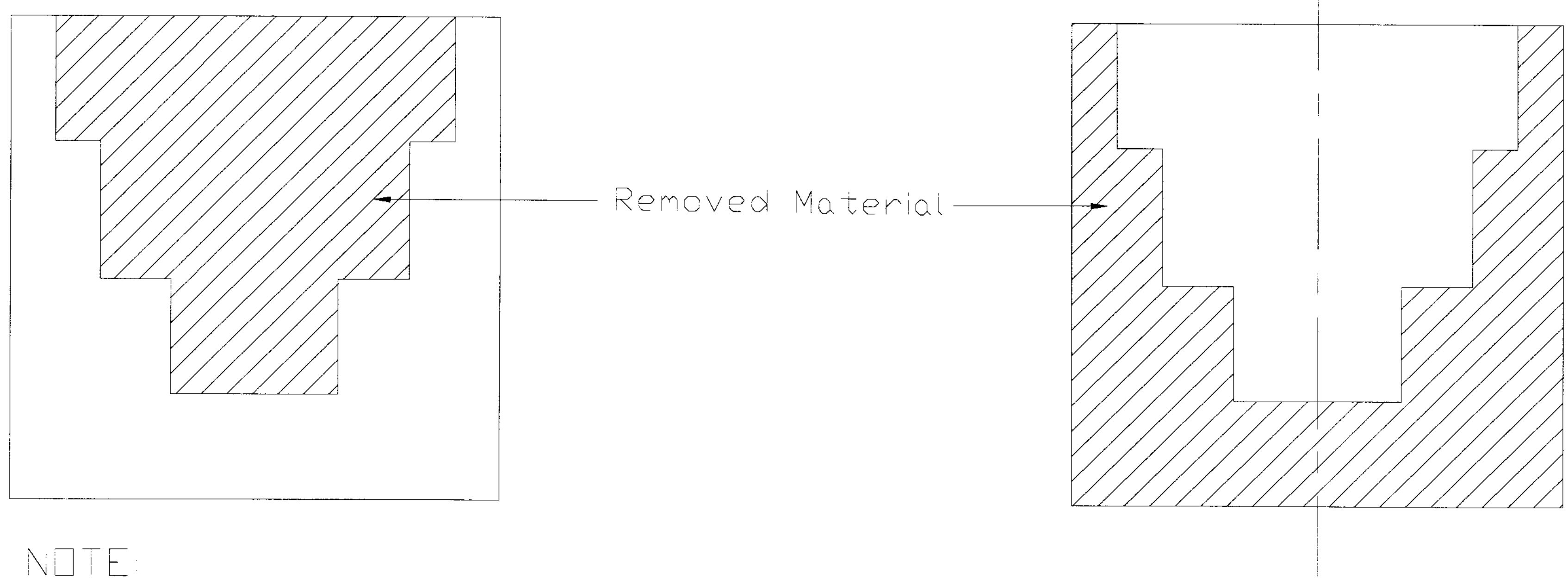
The most suitable programming operation for the included turning operation was found to be a drilling cycle feature. With this feature, the workpiece was assumed to be a drilling tool approaching a single 'fixed' point in the XY plane (*Z movements*). The existing Mastercam Mill software required creation of workpiece profile for generation of NC-Files that calculates toolpaths and machining sequence. With the adoption of workpiece as a 'drilling tool', the requirement for workpiece profile for the generation and definition of NC-toolpaths was not a straightforward case. The main reason being that the Supermax 65A could only cut in vertical direction (milling operation), where the required profile was formed on the material, i.e. material removal was necessary. Therefore, the Tool-plane could not be changed to allow for machining in other planes (YZ or XZ) that formed cutting planes of turning proposal.

However, a reversed operation could be simulated in turning, where the desired profile was formed upon an assumed addition of material to get the final profile. The simulation is shown in Figure 19. While drilling involves removal of the unwanted material from the workpiece to get the required hole, the proposed turning simulation attempted to mould the unwanted material into a turned workpiece. For cases of straight turning, movement of the turning tool (Y-axis movement of table) simulated the depth of cut and movements of the workpiece fixed into the spindle (Z-axis movement) simulated the size of the feed. For the formation of chamfers and radii features, this required simultaneous movements of both the tool (machine table) and the workpiece (spindle).

Figure 19 Simulator for Material Removal for Turning

a) Milling Process

Turning Process



NOTE:
For the two processes, the opposite is true for material removed which fact the research attempted to exploit during simulated programming

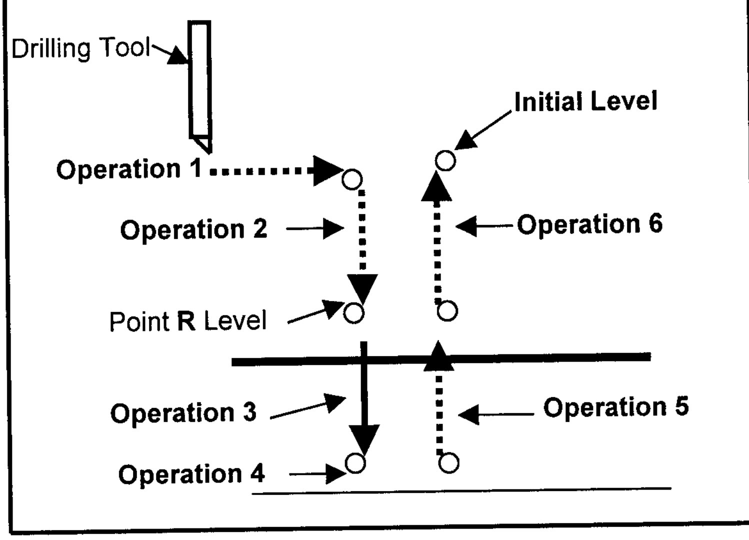
MATERIAL REMOVAL PROCESSES		
PAPER SIZE A4	DRAWN BY Mwanza	DRAWING No:
SCALE 1 : 1	CHECKED:	
MATERIAL: M.S	DATE:	BELONGS TO:

Programming for machining sequence could either be done using Manual Data Input (MDI) at the control panel or Direct Numerical Control (DNC) using a computer. Programs for simple turned profile were done using MDI. However, for complex turned profiles whose programs were big and could not be accommodated by resident memory on the control panel, these were done using DNC. In DNC, computer aided part programming was employed using Mastercam Mill software.

5.5 DRILL TOOLPATH FEATURE

The Drill toolpath feature in Mastercam mill involves the selection of a series of points or entities, create drill paths between them, assign drilling cycles and parameters to them, and generate a drilling toolpath **[13]**. In a typical drilling cycle, the middle point of the drill is programmed with respect to the middle point of hole to be drilled in the workpiece. Typical drilling cycles (Canned Cycles) on the VMC can either be programmed as **G82** (Drilling Cycle Counter) or **G73** (High-Speed Peck Drilling). With canned-cycles, drilling operations can be specified with one **G-code** and its parameters. A typical drilling canned cycle consists of a sequence of six operations as illustrated in the Figure 20.

Figure 20: Drilling Cycle Feature [13]



Operation 1: Positioning of X and Y-axes

Operation 2: Rapid traverse to point R level

Operation 3: Hole machining

Operation 4: Operation at bottom of the hole

Operation 5: Retraction to point R level

Operation 6: Rapid traverse up to the initial point

When the tool reaches the bottom of the hole, it could be returned to point R or initial level. **G98** would retract the tool to initial level, while **G99** would retract the tool to point **R** level. In order to repeat a drilling cycle for many holes over the same incremental distance, the **parameter K** is used. For drilling deep holes and when drilling tough materials, High-Speed Peck drilling (**G73**) is normally used.

Both online and external programming methods were used for part programming of selected typical turned profiles. For external programming, the drill cycle feature was used which implied programming the centre of our workpiece (assumed variable drill) with respect to-the tip of the turning tool (workpiece) fixed on the table. In turning operations, only two axes are used due to the symmetry of the turned parts, hence the X-axis movements of the table constituted the depth of cut, whereas the Z-axis movements was the feed. A similar scenario was employed when calculating the actual positions under manual programming, for simpler profiles.

5.6 ESTABLISHING ZERO-REFERENCE POINT.

The major difference of the simulated turning programming to the actual operations on the VMC using Mastercam was in establishing the Zero-Reference Point. This was the tip-point of the turning tool with respect to the surface coordinates (datum) of workpiece, before any programming was done. This was done as a form of ***OFF-SET*** (employing G-codes) for the fixture that was used. In simulated programming, the turning tool-tip was used to establish the Zero-reference point as opposed to the use of the measuring probe. Typical values obtained during experiments are given in ***Program O1234*** of chapter six. The following was the procedure adopted:

1. Selected turning tool was mounted into the fixture designed for tool holding (Fig. 17).
2. The tool holder assembly was then fixed onto the machine table as secure as possible.
3. The workpiece material was fixed in the standard chuck assembly (Fig. 13).
4. Using manual tool clamp button, the tool-holder was fixed into the machine spindle.

5. Using the ***Manual Pulse Generator Dial*** (allowed for manual operation of axes movements using the handle) and ***Axis Select Switch*** (for selection of axis along which movement was to be made), the reference coordinates of X, Y and Z that conformed to the actual positions of tool tip with respect to workpiece (**G54**) were determined (see Figures 21 and 22). These coordinate values were then **Preset** to Zero ***(OFF-SETTING)*** using the MDI program **Edit**. This then became the Zero-reference point, from which depths of cut were determined or referenced.
6. The machining process was then performed using single block execution, i.e., the operation was done line by line in single block mode (machine stopped every after each line until told to proceed). Figure 23 shows a picture of the tool and workpiece fixtures in position during one of the experiments.

Figure 21: Establishing Reference Point

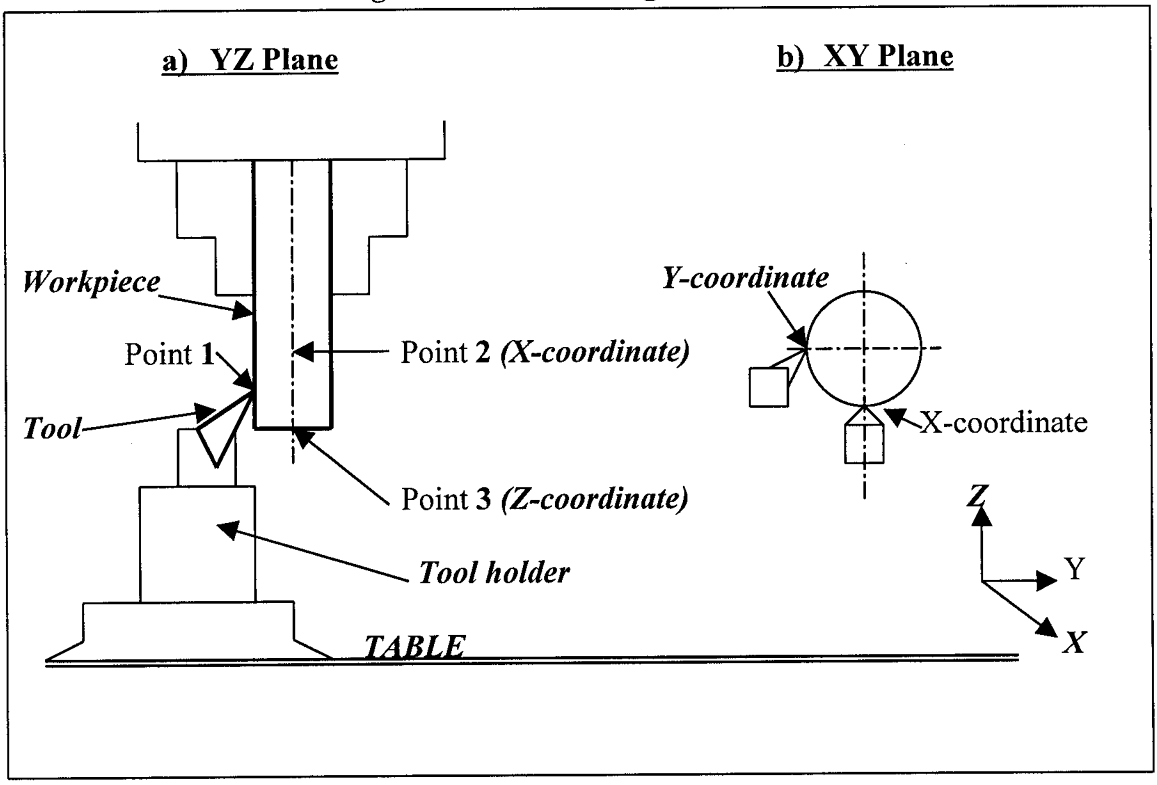


Figure 22: Establishing Reference point

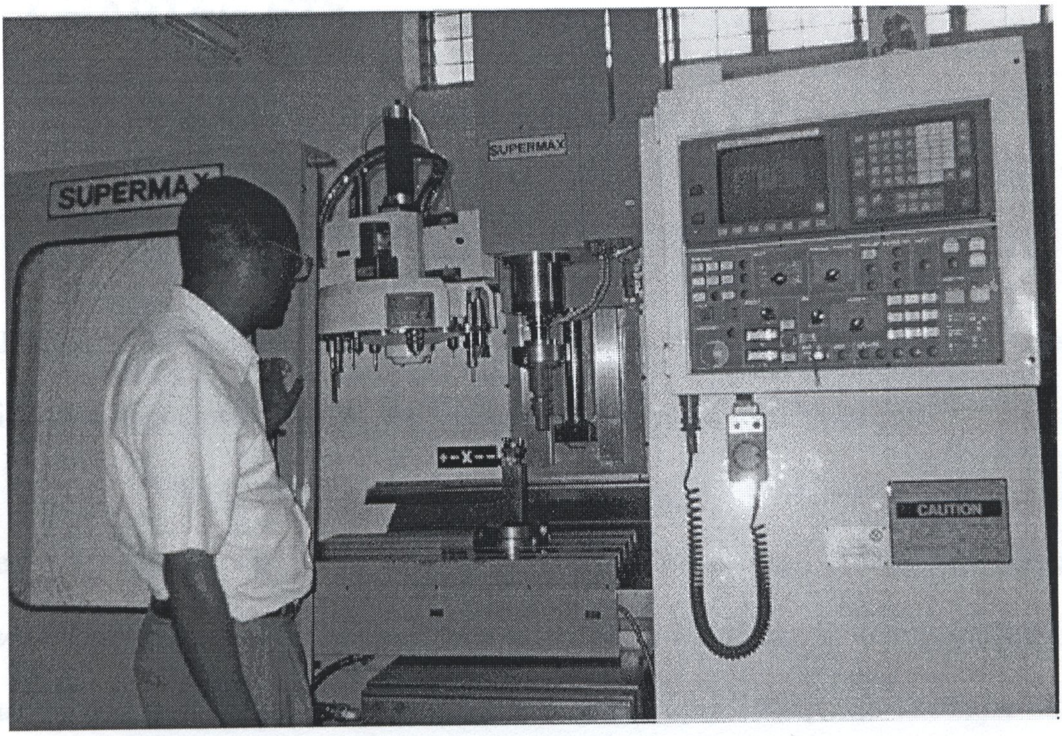
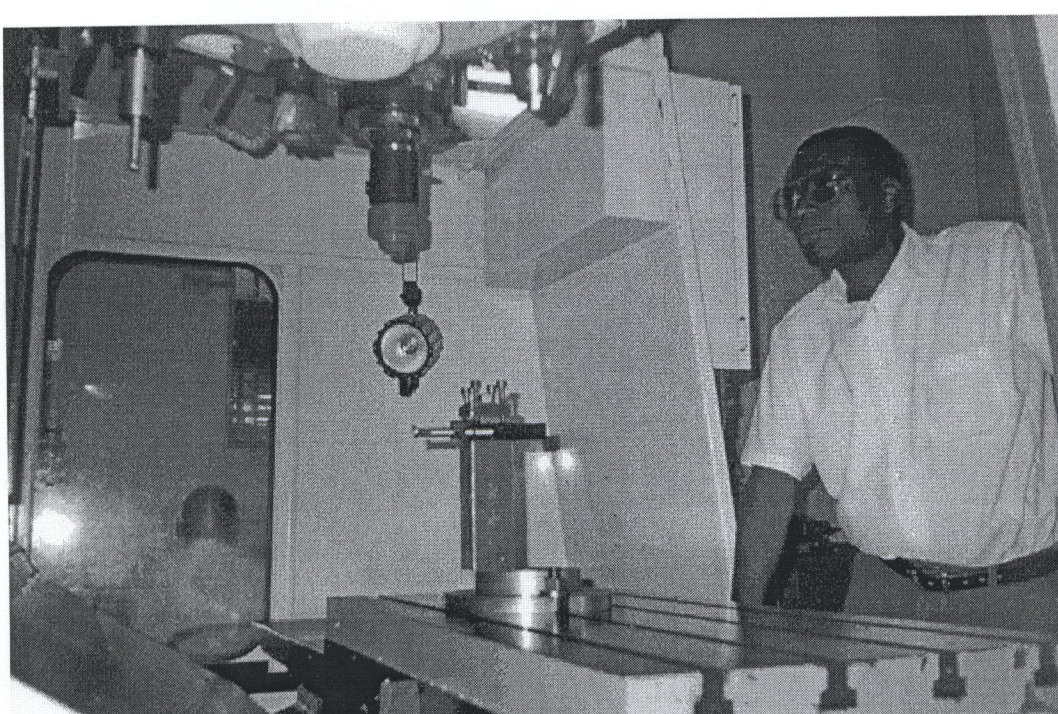


Figure 23: Setting appropriate tool in position



CHAPTER SIX

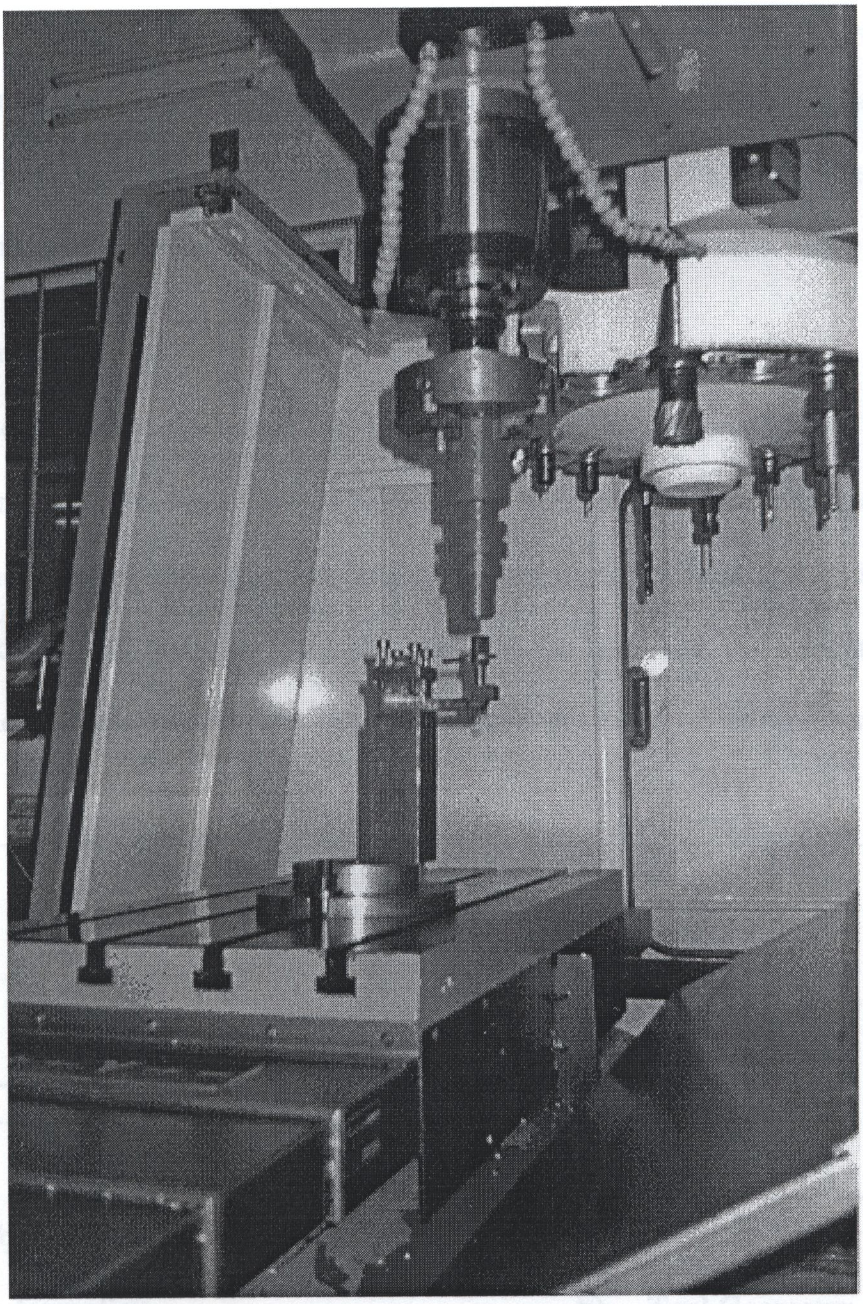
TESTING AND OBSERVATIONS

6.1 TRIAL - RUN OF WORKPIECE HOLDING FIXTURE

A trial test-run of the complete assembly of the workpiece holding fixture without workpiece was conducted. The aim was to find out how well the designed fixture would run on the existing VMC at variable speeds without causing any stress on existing machine parameters (spindle motor power and load on spindle bearings). Using manual tool ***Clamp/Unclamp button*** in the Manual Data Input (MDI) Mode, the fixture was fitted into the machine tool spindle. Spindle speed was then varied from 500 rpm to 1200 rpm using the ***Spindle Speed Override-Percent button*** for a programmed spindle speed ***S1000.*** This is a range within which any typical conventional turning operations are performed.

The trial run was successful as a test run, i.e. without any loading. Successive experiments were done during which actual turning operations and observations were made on the success of the modification design. Figure 24 shows a picture of a typical experimental set-up of fixtures in position.

Figure 24: Workpiece and Tool Holder Fixtures in position



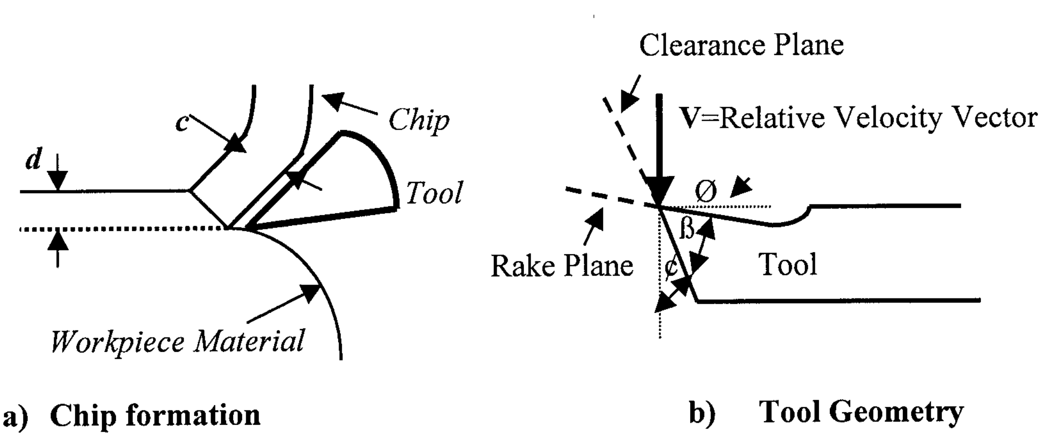
6.2 CHIP FORMATION IN SINGLE-POINT CUTTING

Turning is arguably the most extensively used variation of single point metal cutting. It has received special attention in almost all metal working processes and research. While it was not the objective of this research to detail the process of chip formation and tool geometry during the proposed modification, it was found necessary to point out the requirements for successful operation of the adopted turning operations on the Supermax 65A.

6.2.1 Chip formation

Turning is predominantly performed on lathes discussed in section 2.2.2. The basic requirements are that the workpiece is carried by the rotating chucking mechanism connected to the spindle, while the tool holder mechanism (compound and cross slides) moves back and forward, and also along the length of the lathe bed. As the cutting tool moves along the workpiece, a surface layer of constant thickness (***d***) is removed (Figure 25a).

Figure 25: Single Point Cutting Analysis



This material slides up the front (face) of the tool and breaks off in chips. The chip thickness (*c*) is normally greater than that of the material prior to removal and can give an indication of the degree of difficulty of machining process [14]. A high chip thickness ratio (*c/d*) tends to indicate a material which is difficult to machine. There are a number of variables that can affect chip creation. These include cutting speed, material composition and tool geometry.

6.2.2 Tool Geometry

A cutting tool may be viewed as a wedge, with its cutting edge being the intersection of two planar surfaces. The two planar surfaces generating the cutting edge can be referred to as the rake plane and the clearance plane (Figure 25b). The acute angle (ß) generated between the two planes is referred to as the tool wedge angle. For cutting to occur, there must be relative motion between the cutting tool and the workpiece, hence, this motion is described by a relative velocity vector (V). There are basically two types of geometric cutting conditions in turning operations [15]. These are orthogonal cutting and Oblique cutting. In orthogonal cutting, the tool cutting edge lies in a plane normal to the relative velocity vector. The following angles can be identified in orthogonal cutting; Tool clearance angle (¢), Tool wedge angle (ß) and Tool rake angle (Ø). In oblique cutting, the tool cutting edge does not lie in a plane normal to the relative velocity vector, yielding angle of obliguity and angle of chip flow (complete geometry for the two cutting representation are shown in Appendix II).

For the simple turning operations (Figure 26) orthogonal cutting set-up was employed. This was basically a simulation of turning operational set-up that occurs in lathe work except that it was done vertically (vertical turning). The tool rake angle and clearance angles were both set at 5°. However, because of the expected difficulties in turning chamfers and radii, in complex turned pieces, oblique cutting was employed. A boring bar was found to be the best option for this work as it offered one of the most effective cutting edge at its tip end for chamfers and radii. However, care was taken to make sure that the angle of obliquity was less than 10°, in which case orthogonal conditions were rendered valid.

6.3 EXAMPLES OF TURNING OPERATIONS

a) The profile given in Figure 26 was considered as an example of a simple turned profile. The adopted manual point to point linear interpolation programming was used to prepare the turning operation sequence. The ***Program O1234*** was a typical CNC program generated manually. The language explanations were included and for any further information on CNC programming, this could be found in the machine-programming manuals that came with every CNC machine tool.

Original Diameter of Workpiece		=	ϕ 42.35 mm (irregular)
Material		=	Cast Aluminium
Tool		=	16-mm HSS turning tool
Established Reference points	X	=	**-166.299**
	Y	=	**-117.499**
	Z	=	**-179.461**

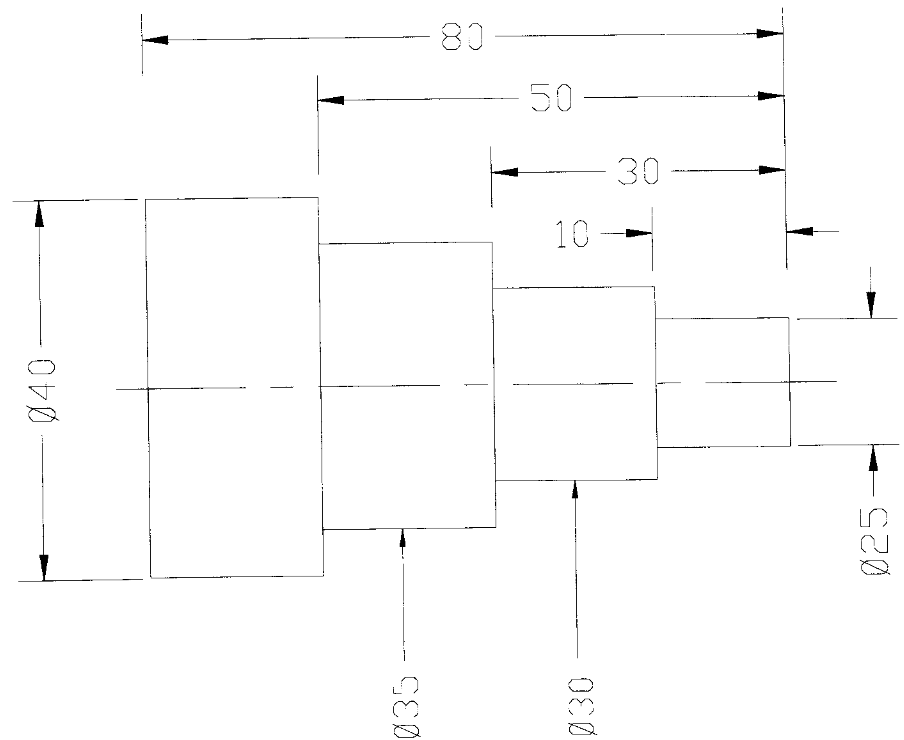
O1234	*Program Number*
N05 G40 G54 G17 G69 G80;	*Standard Preparation statement:*
N10 G91 G28 Z0;	*Machine reference point return:*
N15 S600 M3;	*Spindle turn clockwise:*
N20 G00 G90 X0 Y0 Z5;	*Rapid speed to point above tip of tool:*
N25 G01 G82 G99 Y-1.5 Z-35 R1 P100 F32;	*Effect* canned *cycle in feed:*
N26 Y-1.5 Z-35;	*Regularize diameter:*
N30 Y-4.5 Z-25;	*Feed in Y and turn up to25 mm in Z*

N35 Y-6 Z-15;	*Feed in Y and turn up to 15 mm in Z*
N40 G80;	*Cancel canned cycle:*
N45 G91 G28 Z0;	*Return to machine reference point:*
N50 M30;	*End Programme (**23 Kilobytes**)*

The manually generated program was loaded onto the machine control panel and a test run was conducted to simulate the actual machining cycle, i.e. movements of machining axes. After a successful test run, the workpiece was loaded onto the spindle using the designed fixture, while the selected tool was placed onto the designed tool holder. The turning process was then conducted through single block (per line program execution) processing from the control panel.

The whole program and turning process was successful. The workpiece produced is shown in Figure 32.

Figure 26: Simple Turned Profile



Tolerance ± 0.5 unless otherwise stated

ALL DIMENSIONS IN MILLIMETRES

BASE PLATE		
PAPER SIZE: A4	DRAWN BY: Mwanza	DRAWING No
SCALE: 1:1	CHECKED:	
MATERIAL: M.S	DATE:	BELONGS TO:

b) Another case of a much more complex (chamfer and radius) profile given in Figure 27 was considered. The available memory for the control panel (FANUC Series O-M) is 60-*Megabyte* with an option for a *200-Megabyte* panel (for programming at machine panel). Even though programs for turned profiles tend to be much smaller than milling programs, the available memory might not be enough for storage of many programs or for much more complicated profiles, hence it was desirable to consider external programming on the computer. In creating a CAD drawing for drilling toolpath, both the **Construction Plane** (*Cplane*) and **Graphics View** (*Gview*) were set to **Top View.** A 2D-mold plate was created giving a series of circles for the different sections or diameters of the profile (**plan view**) as shown in Figure 28. The selected entities were then entered at different Cplane levels, corresponding to different sections of the shaft. The software generated program (***O2340***) could not be run straight away as a machining process because of the machine setup in which the workpiece was assumed to be the machining tool. This was opposite to what the Supermax- 65A post-processor was designed for, hence much editing had to be done, and in which case knowledge of manual programming became very necessary.

For different sections of the workpiece, it was assumed that different drills were used. Hence the required tool data (diameter, length and off-sets) for drills were preloaded into the machine tool library through the ***Define Tool option***. The final stage in the generation of the program was the running of the postprocessor. This was done through the ***Post proc. menu***.

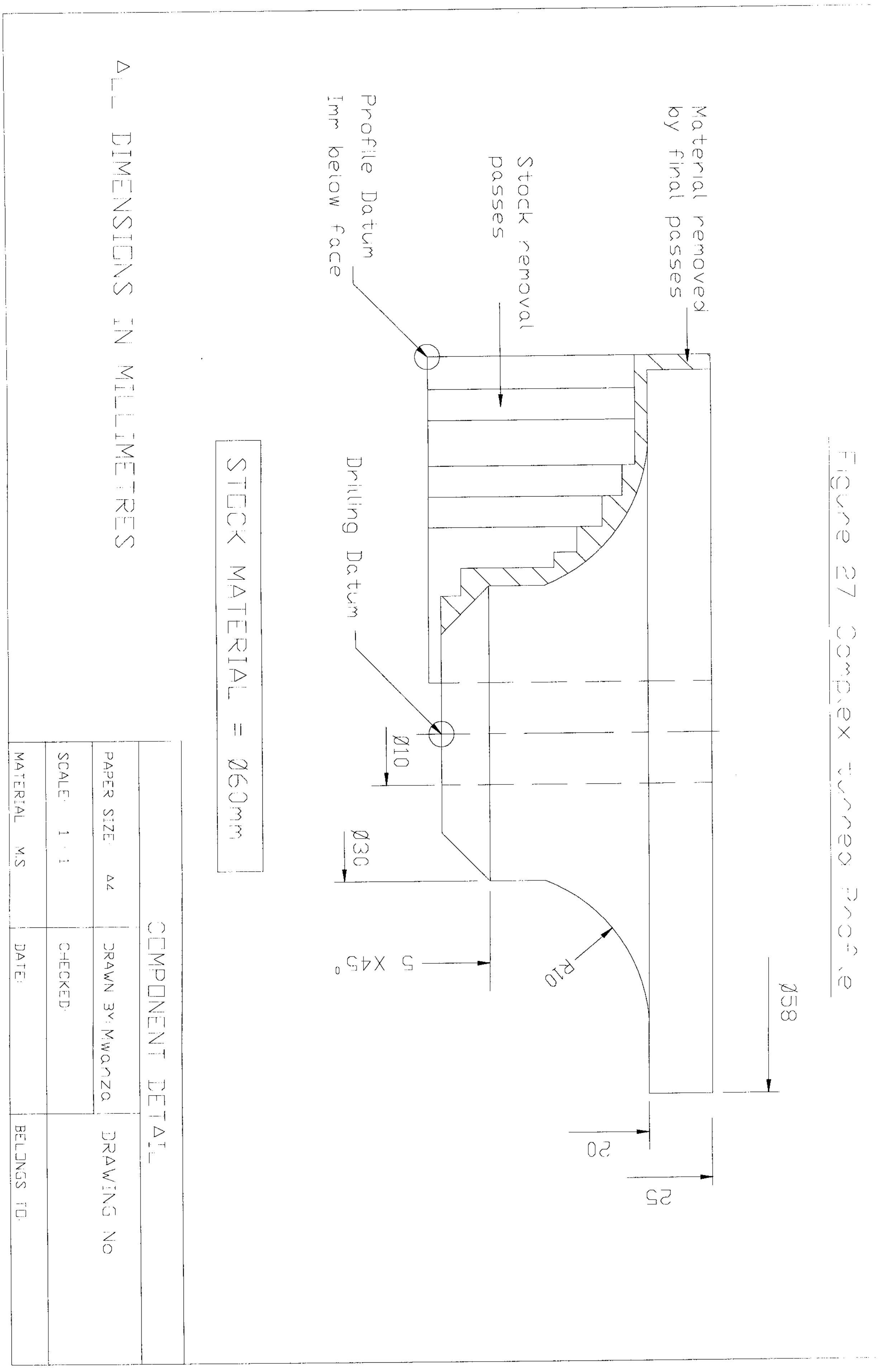
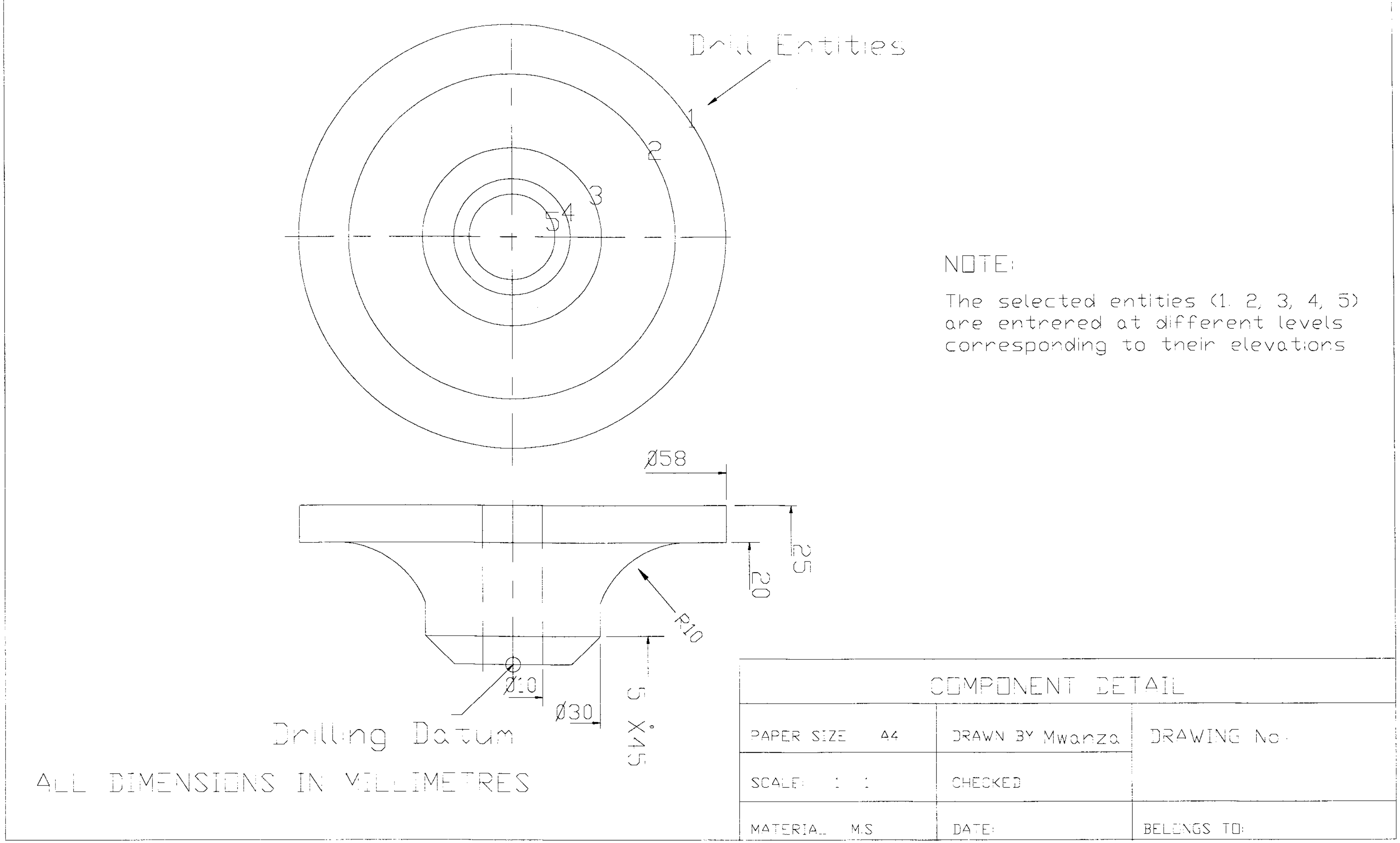


Figure 27 Complex Turned Profile
Material removed by final passes
Stock removal passes
Profile Datum 1mm below face
Drilling Datum
Ø58
Ø10
Ø30
R10
5 X45°
20
25
STOCK MATERIAL = Ø60mm
ALL DIMENSIONS IN MILLIMETRES
COMPONENT DETAIL
PAPER SIZE A4
DRAWN BY: Mwanza
DRAWING No
SCALE 1:1
CHECKED
MATERIAL M.S
DATE
BELONGS TO

Figure 28 2-D Mode Plate

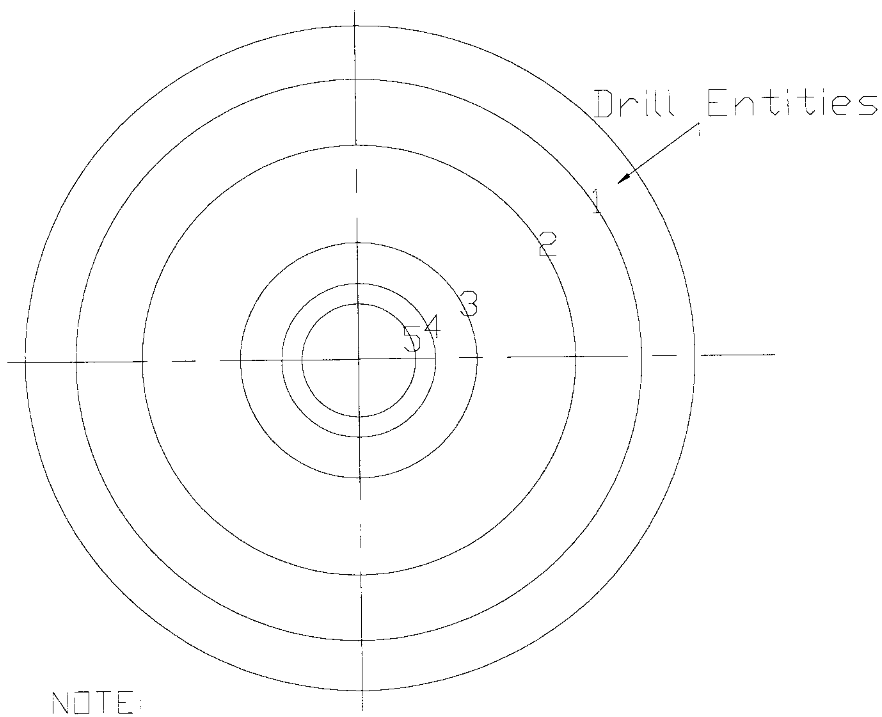


The generated ***NC-file Program 02340*** for assumed drilling cycle was as follows:

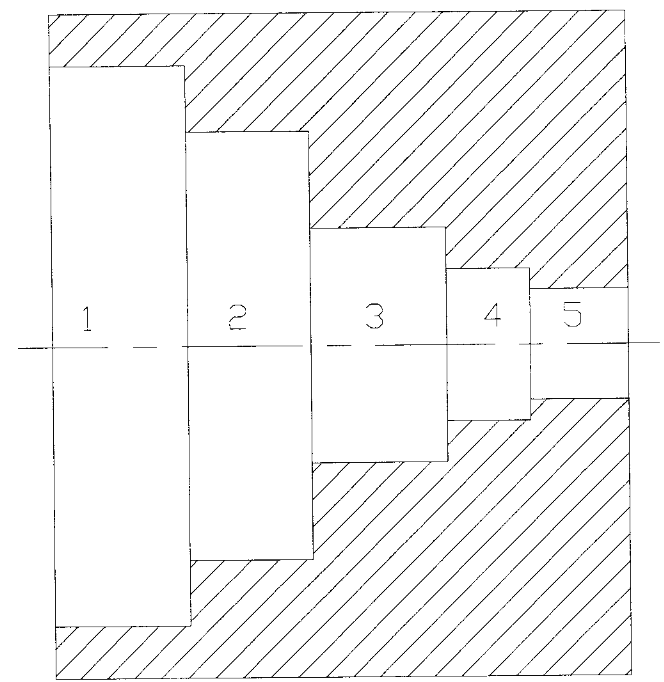
```
%
02340  ( MCEMM1 )
( DATE   :    10  05  99      TIME :      10:30 )
( TOOL=02     LENGTH  OFF.=02        DIA.  OFF.=52       DIA. = 8            TOOL NAME = SLOT-4 )
( TOOL=03     LENGTH  OFF.=03        DIA.  OFF.=53       DIA. = 10.000       TOOL NAME = SLOT-5 )
( TOOL=04     LENGTH  OFF.=04        DIA.  OFF.=54       DIA. =58.000        TOOL NAME = SLOT-6 )
( TOOL=05     LENGTH  OFF.=05        DIA.  OFF.=55       DIA. = 40.000       TOOL NAME = SLOT-7 )
( TOOL=06     LENGTH  OFF.=06        DIA.  OFF.=56       DIA. = 30.000       TOOL NAME = SLOT-8 )
( TOOL=07     LENGTH  OFF.=07        DIA.  OFF.=57       DIA. = 20.000       TOOL NAME = SLOT-9 )
N05  G40 G54 G17 G69 G80;
N10  G91 G28 Z0;
N15  T2 M6;
N20  S1000 M3;
N25  G00 G90 X0 Y0;
N30  G43 Z5 H2 M8;
N35  G82 Z-2 R2 F150;
N40  G80;
N45  G91 G28 Z0;
N50  T3 M6;
N55  S900 M3;
N60  G43 Z5 H3 M8;
N65  G73 Z-30 R2 Q10 F160;
N70  G80;
N75  G91 G28 Z0;
N80  T4M6;
N85  S600 M3;
N90  G43 Z5 H4 M8;
N95  G73 Z-5 R2 F160;
N100 G80;
N105 G91 G28 Z0;
N110  T5 M6;
N115  S700 M3;
N120  G43 Z5 H5 M8;
N125  G73 Z-15 R2 Q5 F160;
N130  G80;
N135 G91 G28 Z0;
N140  T6 M6;
N145  S800 M3;
N150  G43 Z5 H6 M8;
N155  G73 Z-20 R2 Q5 F160;
N160  G80;
N165 G91 G28 Z0;
N170 T7 M6;
N175  S900 M3;
N180  G43 Z5 H7 M8;
N185  G73 Z-25 R2 Q5 F160;
N195 G80
N200  M30;
%
```

The cross section of profile produced by this program is given in Figure 29. As pointed out earlier, it was obvious that turning programs generated from Mastercam Mill software needed to be edited if correct profiles were to be made. The most crucial statements being tool change (all those with **M6**), plane of cutting (between N140 and N185 where radius and chamfer occurred) and direction of cutting feeds (Y-axis movements included as depth of cut at each statement with Z-axis movement).

Figure 29 Cross-section of Profile generated



Cross-section of actual drilled profile



NOTE:

The selected entities (1, 2, 3, 4, 5) are entered at different levels corresponding to their elevations for each specified drill size in the tool menu

CROSS-SECTION OF PROFILE GENERATED		
PAPER SIZE A4	DRAWN BY: Mwanza	DRAWING No
SCALE: 1 :	CHECKED	
MATERIAL: M.S	DATE:	BELONGS TO:

Since the centre of the drilled holes remained the same, the difference in diameter of the holes (assumed being drilled) constituted the depth of cut in our case, hence movement in Y-axis.

The editing of the NC-file was done using the *Edit NCI option*, while being controlled with the ***Backplot option***. These relevant options were accessed via the **Main Menu** of the software. For clarity, the edited program was given a different program number (O2341).

For the proposed programming, all tool-change statements were skipped during machining. The cutting plane G-codes were incorporated in relevant program statements, (e.g. **N150** of **O2341** where **G19** was used instead of **G17**) to allow for circular interpolation in YZ plane. Statements following each tool change were incorporated with a movement in Y-axis to give a depth of cut, hence each drill cycle has a series of cuts.

The final edited NC-file was again postprocessed using the *Reverse option* on *the **Post proc menu.*** This allowed for the conversion of the NC-file into an NCI-file for the selection of the required machine control system. This control system allowed the execution of machining on the Supermax, which is equipped with a ***Fanuc 0 m*** control system. This gave an option of machining in single block mode.

The edited ***Program O2341*** with relevant explanation was as follows:

Material = **Prowax:** **Original Diameter** = **65 mm**

Tool = **HSS ϕ 10 mm - length 90 mm Boring bar**

O2341	*Program Number*
N05 G40 G54 G17 G69 G80;	*Standard Preparation statement:*
N10 G91 G28 Z0;	*Machine reference point return:*
N15 T6 M6;	*Tool change statement for centre Drill:*
N20 S1000 M3;	*Spindle turn:*
N25 G00 G90 X0 Y0;	*Position above centre of workpiece:*
N30 G43 Z5 H6;	*Effect length compensation:*
N35 G82 Z-2 R2 F150;	*Call drilling cycle, drilling depth Z-2, reference point R2*
N40 G80;	*Cancel canned cycle:*
N45 G91 G28 Z0;	*End Centre drill:*
N50 T7 M6;	*Effect Tool change for Drill:*
N55 S900 M3;	
N60 G43 Z5 H7;	*Start Drilling:*
N65 G73 Z-30 R2 Q10 F160;	*Peck drill depth Z-30, with steps of 10:*
N70 G80;	*Cancel canned cycle:*
N75 G91 G28 Z0;	*Zero return and end drilling:*
N80 M30;	*End Program and fix turning fixtures:*
N85 S600 M3;	*Spindle clockwise turn:*
N90 G90 G00 X0 Y0 Z5;	*Rapid speed to point above tip of tool:*
N95 G01 G91 G82 G99 Y-1 Z-20.5 R1 P100 F50;	*Effect canned cycle with depth of cut:*
N100 Y-1 Z-20;	*Feed in Y and turn up to 20 in Z:*
N105 Y-1.5 Z-19;	*Feed in Y and turn up to 19 in Z*
N110 Y-1.5 Z-17.5;	*Feed in Y and turn up to 17.5 in Z*
N115 Y-1.5 Z-14;	
N120 Y-1 Z-4.5;	
N125 G90 G00 X0 Y0;	*Rapid return to workpiece zero position:*
N130 S800 F32;	*Spindle speed and feedrate change for profile finish:*
N135 G01 Y-30 Z-1;	*Machine face to ϕ 20:*
N136 G00 Y-20 Z-1;	*Go to beginning of chamfer*

N140 G01 Y-15 Z-6;	*Machine chamfer:*
N145 Y-15 Z-11;	*Linear move to radius start:*
N150 G19 G03 Y-10 Z-21 J10;	*Circular interpolation for radius profile:*
N155 G17 G01 Y-1 Z-21;	*linear move to ϕ 58:*
N160 Y-1 Z-26;	*Linear move to length:*
N165 G80;	*Canned cycle cancel:*
N170 G91 G28 Z0;	*Zero return;*
N175 M30;	*End Program;* ***(40 Kilobytes)***

Figure 30 shows a picture of actual machine set-up during this experiment.

For the component in Figure 27, it was necessary to provide an operation sequence and tooling information for the machine operator. There are various standard forms that are used to enter this information. These forms contain as much information as possible for the operator to set the machine. Their standard format depends on the machine type being used. For this trial ***Form I*** below was adopted and used:

FORM I: OPERATION SCHEDULING

OPERATION SCHEDULE	PART No. / MACHINE TYPE:		DESCRIPTION: / COMPILED BY:		SHEET No. / DATE:	
OPERATION No.	**DESCRIPTION**	**TOOLING TYPE AND SIZE**	**WORK HOLDING**	**CUTTING SPEED (m/min)**	**FEEDRATE (mm/rev)**	**SPINDLE SPEED (rpm)**
1	Centre Drill	HSS C/Drill	Table vice	28	0.12	1500
2	Drill	HSS Drill ϕ10	Table vice	28	0.18	890
3	Turn Profile	HSS - B.Bar	Chuck	170	0.25	1350
4	Part Off	CC Insert	Chuck	170	0.16	1350

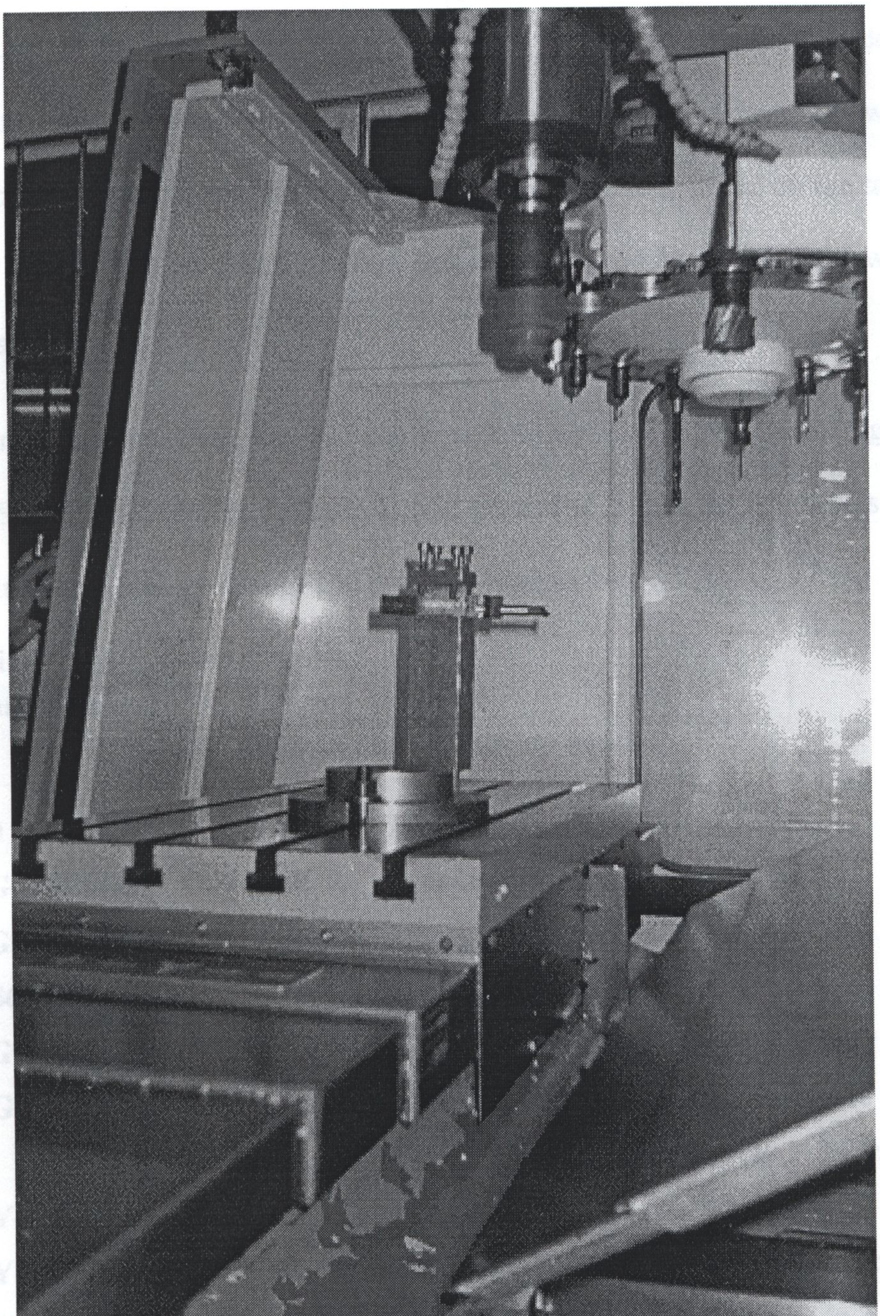
The sequence of operations therefore involved:

1. Centre drilling and actual drilling of hole. These were done on the VMC in the usual way of drilling operation, before profile turning. This required careful setting during the two stages, but could also have been done in one setting, which might have required a fixture for drill holding on the existing machine table.
2. Turning to profile using the designed fixture and DNC (edited) programming.
3. Parting-off using the modified turning operation or on conventional lathe.

There was however, an observed change in the size of the chip during machining of radius and chamfer (Fig. 27). The measured samples (assume change was in chip width) of the chips from these sections were in some cases double those from previous machined sections. It was difficult to determine whether this was due to change in chip thickness, width or indeed both. However, this was investigated using more rough-cuts in these areas, before the final finishing cuts. There was an observed large reduction in the chip sizes at those sections. However, it was difficult to establish exactly the optimum number of rough-cuts to be done, nevertheless it was recommended to do as many as is practically possible. This reduces cutting pressure on the cutting tool especially when cutting metals and improves surface quality.

The two manufactured pieces are given in Figure 32.

Figure 30: Boring bar used for turning complex workpiece



c) A turned profile (Figure 31) with more external features (grooves and chamfer) was also considered. Since the profile was made up of straight turned sections, G-code manual programming was followed. The programming involved considering the workpiece (fixed in spindle) rotating and feeding in Z-axis, while the tool (fixed on the table) was feeding in Y-axis. The co-ordinates were calculated depending on the final workpiece dimensions.

It was important to select a single tool that could perform both turning and grooving. A 10-mm High-Speed Steel (HSS) parting-off tool, ground to 3-mm tip width was used for both rough cutting and final grooving. The program is given below:

Material	=	**Prowax**
Original Diameter	=	**85 mm**
Tool	=	**HSS 16 mm - 3-mm tip width**

O1410	*Program Number*
N05 G40 G54 G17 G69 G80;	*Standard Preparation statement:*
N10 G91 G28 Z0;	*Machine reference point return:*
N15 S600 M3 M8;	*Spindle turn and coolant on:*
N20 G90 G00 X0 Y0 Z5;	*Rapid speed to point above tip of tool:*
N25 G01 G91 G82 G99 Y-1.5 Z-57 R1 P100 F50;	*Effect canned cycle with depth of cut Y-2.5 in feed:*
N30 Y-1 Z-57;	*Feed depth Y-1and turn up to 57 in Z:*
N35 Y-2 Z-48;	*Feed depth Y-2 and turn up to 48 in Z:*
N40 Y-2 Z-48;	*Feed depth Y-2 and turn up to 48 in Z:*
N45 Y-1 Z-48;	*Feed depth Y-1 and turn up to 48 in Z:*
N50 Y-1 Z-45;	
N55 Y-0.25 Z-45;	
N60 Y-1.25 Z-33;	

N65 Y-1 Z-33;

N70 Y-2 Z-18;

N75 Y-1.5 Z-18;

N80 Y-0.25 Z-10; *Feed depth Y-0.25 and turn up to 10 in Z:*

N85 G80; *Cancel canned cycle:*

N90 G00 G90 X0 Y0; *Rapid return to reference point:*

N95 Y-11.5 Z-45; *Go to point Y-11.5 and Z-45 (absolute)in rapid speed:*

N100 G01 G91 Y-3.75 Z-41 F50; *Feed depth Y-3.75 (incremental), turn up to Z-41 in Z in-feed motion:*

N105 Y2 Z-33; *Feed depth Y-2, turn up to Z-33:*

N110 Y-3.5 Z-28; *Feed depth Y-3.5, turn up to Z-28:*

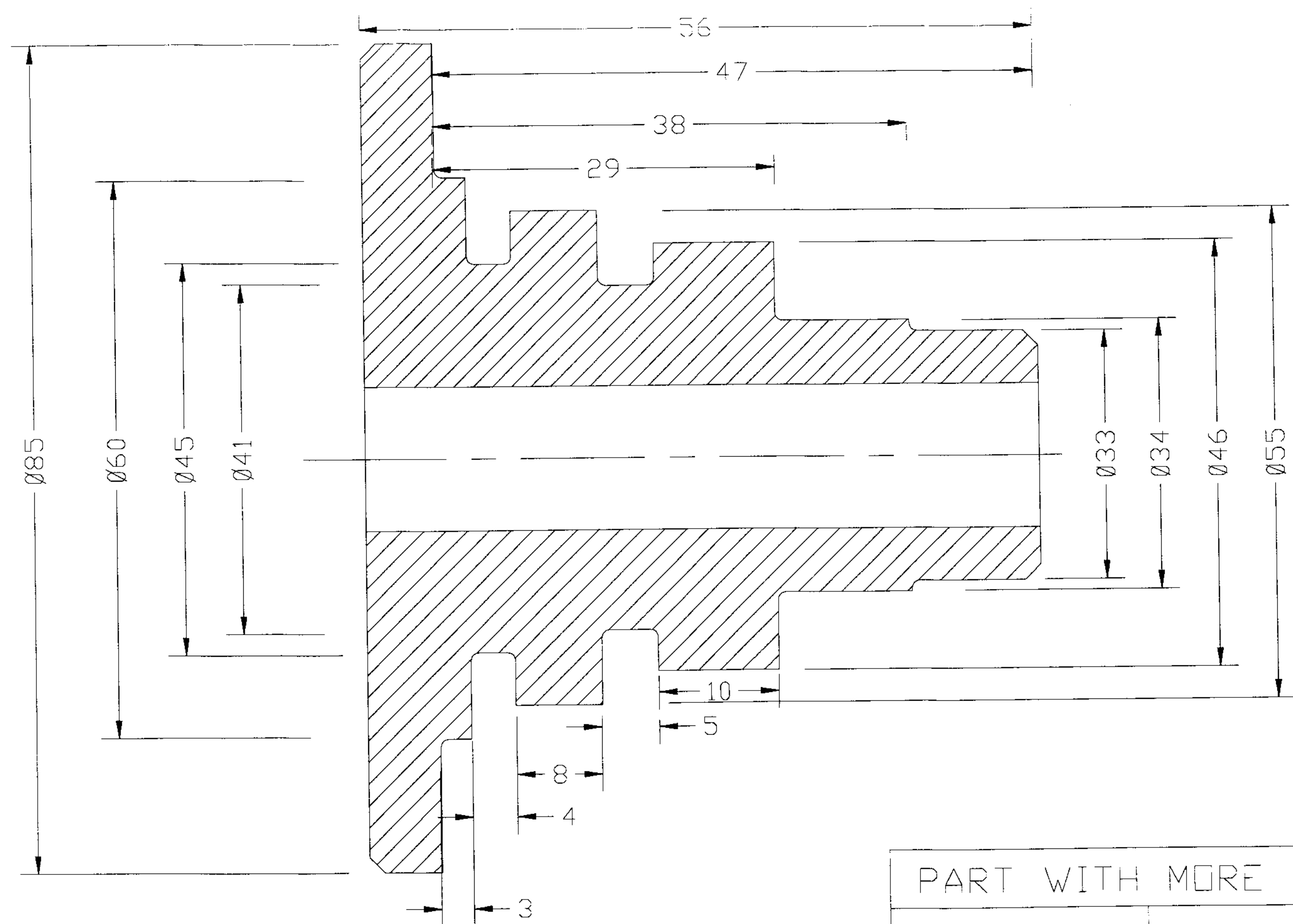
N115 Y4; *Go to Y+4 in-feed*

N120 G91 G28 Z0; *Zero return;*

N125 M30; *End Program:*

The machining of grooves required very small depth of cut and lower feeds. These were taken so as to reduce cutting pressure on the tool. The machining progressed very well and the manufactured piece is shown in Figure 32.

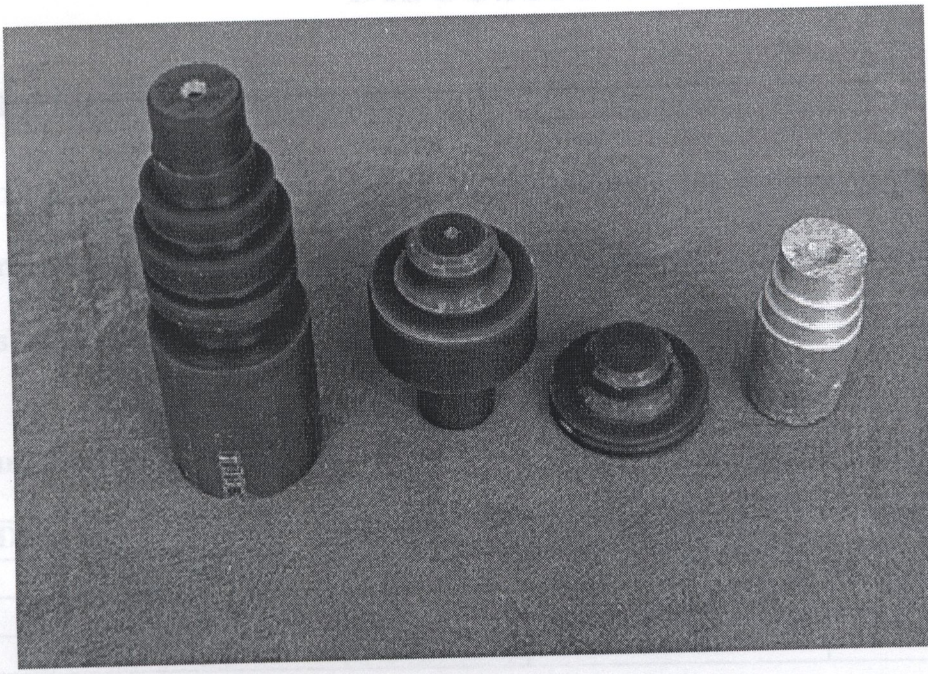
Figure 31 Profile with more external features

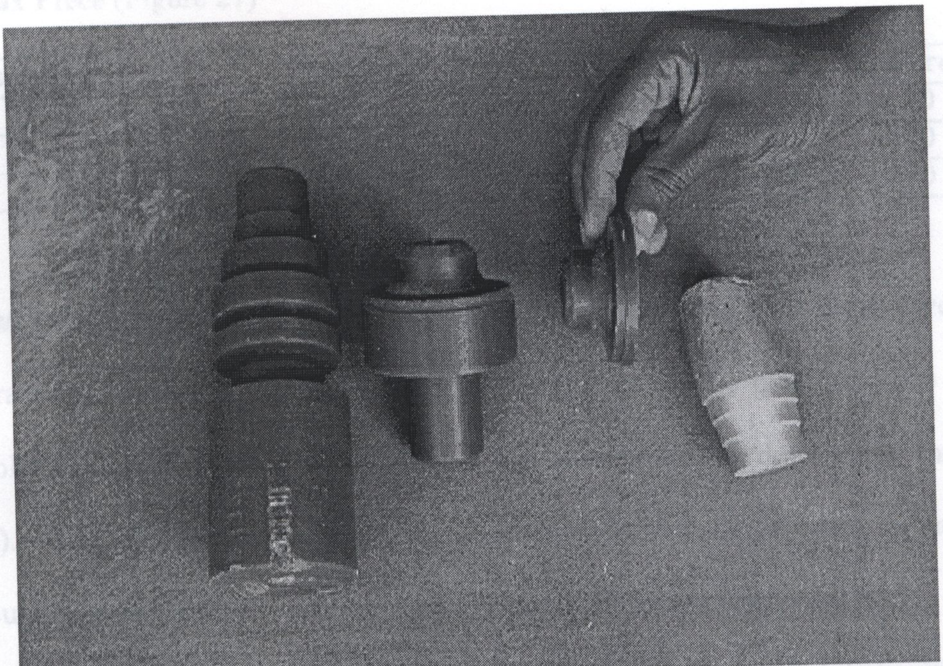


Measurements in millimeters
Tolerance ±0.3 unless otherwise stated

PART WITH MORE EXTERNAL FEATURES		
CHECKED:	DRAWN BY Mwanza	DRAWING No.
MATERIAL M.S	DATE:	BELONGS TO:

Figure 32: Samples of turned pieces





CHAPTER SEVEN

DISCUSSION

7.1 DIMENSIONAL ACCURACY

As an indication of dimensional accuracy, the dimensions in the drawings (expected) and on final turned workpieces were compared. A vernier caliper was used for measuring selected sections of turned pieces:

a) **Aluminium Piece** (Figure 26)

Diameter	Drawing	Machined	Difference
1	38.45	38.45	0
2	35.45	35.45	0
3	32.45	32.45	0

b) **Prowax Piece** (Figure 27)

Diameter	Drawing	Machined	Difference
1	64.70	64.70	0
2	34.80	34.80	0
3	27.30	27.30	0

The dimensional results obtained from machined parts were similar to those given on the design drawings. This was expected since measurements on the machine were of much higher tolerance (0.001 mm) compared to the measurements done with the caliper (0.01mm). Minor differences could appear in the third decimal figure when measured with more accurate instruments e.g. a micrometer. These differences would not be big since the machine tool being used was of high precision. It was therefore concluded that, within given design, product and process tolerances, the CNC simulated turning fixture should give appreciable higher dimensional accuracy and surface finish on turned workpieces.

7.2 ESTIMATION OF CUTTING FORCE AND POWER

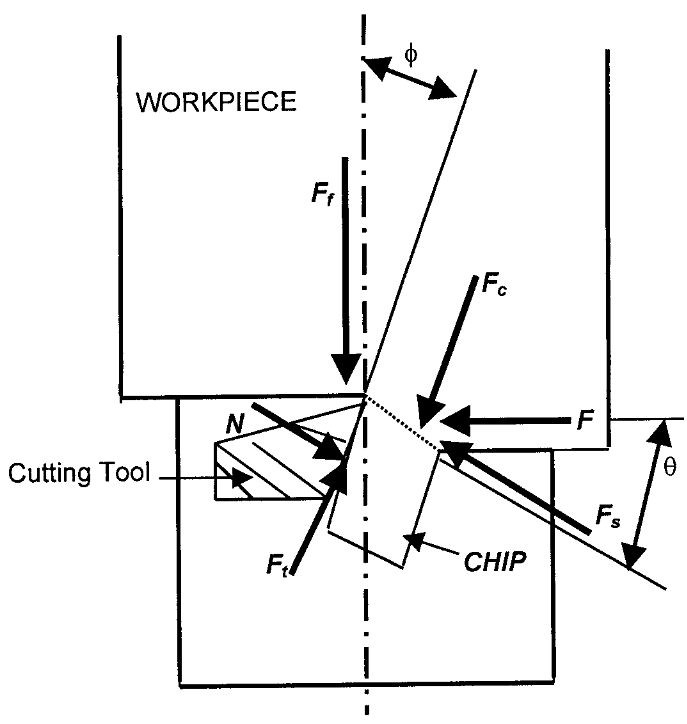
An indication of cutting force and power consumed during simulated turning operations on the Supermax could be calculated using cutting-tool dynamometer setup or standard formulae obtained in single-point cutting **[16]**. The consideration was important given the fact that, the cutting forces were reversed during turning operations, (Figure 33) i.e., workpiece moving against stationery tool.

Nevertheless, reversed direction did not give any concern for the machine tool, because the spindle bearings were designed to carry thrust loads in either direction. Therefore, the most important consideration was the estimation of cutting force and power consumed. These parameters would also give an indication of the suitability of the design.

Figure 33: Chip formation and Force system on Simulated Turning Operations

F_c = compressive force on the shear plane
F_s = shear force on the shear plane

Figure 33: Force System



7.2.1 Cutting Force

The cutting speed *V* (m/min) is the velocity of the workpiece and tool at the tool edge. For any point on that tool edge it was considered as a vector and the cutting force *F* (N) was measured along the line of this vector. For certain cutting conditions i.e., workpiece material, shape of cut (d x *f* = depth of cut x feed), tool shape, approach angle and cutting speed, the cutting force could be related to the depth of cut *d* (mm) and feed *f* (mm/rev) by the empirical equation of the form $F = C.\, d^x f^y$, where C is a constant for the material cut.

7.2.2 Power consumed in cutting

The power required to remove a unit volume of material per minute was a measure of its resistance to cutting; it also gave an indication of the effectiveness of the cutting conditions. Measurement of the input to the electric motor (using Watt-meters) of a machine tool provides a convenient method of measuring the power consumed in cutting. If the value of the power supplied when the machine is running idle is subtracted from the power reading taken under the cutting load, a reasonable estimate of the power consumed in cutting could be obtained. This gives an approximation only because the efficiency of the drive under varying loads is not taken into account.

The other standard methods [17] for estimating the Power (P) consumed in cutting are given below:

i) Lathe-work

$$P = \frac{k_l\, d\, f\, V}{60,000}$$

ii) Drilling

$$T = k_d\, f^{0.75}\, D^{1.8}$$

$$P = \frac{2\pi N\, T}{60,000}$$

iii) Milling

$$P = \frac{k_m\, d\, W\, f_m}{60}$$

P = power used (kW)
d = depth of cut (mm)
f = feed (mm rev^{-1})
V = cutting speed (m min^{-1})
T = torque (N m)
D = Drill diameter (mm)
N = rotational speed (rev min^{-1})
W = width of cut (mm)
f_m = machine table feed (mm min^{-1})
k_l = factor for lathe (N mm^{-2})
k_d = factor for drilling
k_m = factor for milling (J mm^{-3})

TABLE A: Different Material factors for machining [17]

Material	k_l (N mm^{-2})	k_d	k_m (J mm^{-3})
Aluminium	700	0.110	0.9
Brass	1250	0.084	1.6
Cast Iron	900	0.070	1.9
Mild steel	1200	0.360	2.7
Tool steel	3000	0.400	7.0

Using the formula (i) and the values for aluminium in Table A, the power used in cutting was calculated and subsequently the cutting force in our trial. The following were obtained:

$$\text{Power} = \frac{700 * 1.5 * 0.3 * 120}{60{,}000} = \underline{\mathbf{0.63\ Kw}}$$

$$\text{Cutting Force (F)} = \frac{0.63 * 60}{120} = \underline{\mathbf{0.315\ kN}}$$

The estimated force and power experienced during simulated turning were expected to be much lower than those expected during typical milling or drilling cycles on the Supermax. Even though the materials used during the trials were less tough than steels, it was not expected that the forces and power would be more than double those calculated above. The simulated turning operations could hence be used on most materials without causing any concern on power requirements. It is however, recommended that, much smaller depth of cuts and feeds should be employed when machining tougher materials like steels. Care needs to be taken when machining radii, fillets, chamfers, grooves, etc, as these tend to form areas of maximum material removal rates. It was recommended to do more roughing and finishing cuts as a caution against excessive pressure on cutting tools and changes in chip formation during machining.

CHAPTER EIGHT

CONCLUSIONS

1. The design fixtures for tool and workpiece holding performed very well on softer materials (Aluminium and Prowax). The fixtures used in this research were not in anyway restrictive. Any suitable designs could be adopted depending on the machining requirements. For example, during one of the experimental trials (Figure 27), a standard tool-holder with a larger bore (ϕ 32 with grab screws) was used as workpiece holder. However, whatever fixture design is used, movements and deflections should not be allowed. Any deflections will result in poor dimensional accuracy and surface finish of workpiece. In this research, a locking-pin was provided (in turret block) to counter any movement of the tool-holder during turning operations.

2. The project research used manual programming and available Mastercam Mill software to do selected turned profiles. The drill feature that was used in this research might not be the only suitable feature for complex turned profiles. For example, for much shorter turned profiles, pocketing or contour milling features could be simulated. It is obvious that for complex profiles, adequate editing needed to be done if the required profile were to be attainable. This requires that the programmer understand in detail the operations of the simulated turning. It is recommended that a rough manual program be done before using the programming software to give an idea of expected program. Direct Numerical Control machining offers a much quicker way of programming and editing for complex turned profiles. For special features (internal and external) like radii, grooves, recesses, it

is recommended to use form tools that could produce the required respective feature on the final workpiece.

3. This project has been able to carry out successful research activities in enhancing machine tool capabilities. The increased capabilities of the machine tool will obviously result in increased range of outputs and hence less cost of production per unit. While the cost of modification was not considered in detail, it is appreciated that this could not be compared to the cost of another turning machine. The knowledge should set precedence in any such future activities.

CHAPTER NINE

RECOMMENDATIONS FOR FURTHER RESERACH

The following could be considered for further research:

1. Experimental setup of the deflection type cutting-tool dynamometer, to accurately determine the cutting force. The setup must simulate orthogonal conditions i.e., the tool edge should be made to lie in the plane containing the work axis, and hence making the force opposite the direction of depth of cut negligible. The two-dial gauges which measures the remaining deflections could be calibrated to read units of force. This could give a much more accurate estimate of cutting force and power.

2. An analysis of chip formation during turning operation could be done to determine the actual change in chip area. This is as a result of the observed change in the chip size when machining chamfers or radii.

3. The performance of similar fixture arrangements for form tool and workpiece holding for turned internal features could also be investigated.

REFERENCES

1. **NORMAN, G.; (1996)** - *Production and Operations Management*, 7th Edition, pp. 171.
2. **The Japan of Today (1993)** - *The International Society for Educational Information Journal.*
3. **COMESA Journal of Industry and Trade (1996)**, Volume 4, pp. 10.
4. **Machine Design Journal (November, 1991)**, A Penton Publication, pp. 28.
5. **Internet Resource** - *http://www.spinner- wzm.de. Machine Tool Magazines,* ***Nr.: 60163***
6. **Internet Resource** - *http://www.me.mtu.edu. CNC Technology course ME 405.*
7. **GIGGS, D.;** "*An Introduction to CNC Machining*", (1990). Second Edition, Cassell Publishers, pp. 20.
8. **Internet Resource** - *http://www.spinner- wzm.de. Machine Tool Magazines.*
9. **Internet Resource** - *http://www.hanmachine.com. Machine Tool Magazines.*
10. **YCM - VMC - 60A, Supermax Operator Manual.**
11. **GIGGS, D.;** "*An Introduction to CNC Machining*", (1990). Second Edition, Cassell Publishers, pp. 72.
12. **Mastercam Design Reference Manual**, Version 7.0, 1984-1998 CNC Software Inc.
13. **Mastercam Mill Reference Manual**, Version 7.0, 1984-1998 CNC Software Inc.
14. **LISSAMAN, A. J. and MARTIN, S. J.** (1982). *Principles of Engineering Production*, pp. 169 – 175.
15. **LISSAMAN, A. J. and MARTIN, S. J.** (1982). *Principles of Engineering Production*, pp. 183 - 198.
16. **LISSAMAN, A. J. and MARTIN, S. J.** (1982). *Principles of Engineering Production*, pp. 198.
17. **GREER, A. and HANCOX, D. J.**; (1990). *Tables, Data and Formulae for Engineers and Mathematicians.*
18: **FANUC SERIES 0/00/0** - *Mate for Machining Centre Operator's Manual - B-61404E*

RESEARCH PUBLICATIONS

1. **Mwanza, S. M., Mwenda H. M., Siaminwe, L.; (1999).** Adaptive widening of the capability of the Computer Numerical Control Vertical Machining Centre to include turning operations - An Investigation. Paper presented at Engineering Institution of Zambia Seminar, LUSAKA - ZAMBIA

2. **Mwanza, S. M., Mwenda H. M., Siaminwe, L.; (November, 1999).** Using Mastercam Mill Software to program simulated turning operations on Computer Numerical Control (CNC) Vertical Machining Centre. The Zambian Engineer, Vol. 34 N0. 1, Engineering Institution of Zambia, LUSAKA - ZAMBIA

APPENDICES

APPENDIX I

INTERNATIONAL STANDARD ORGANIZATION G-CODES [18]

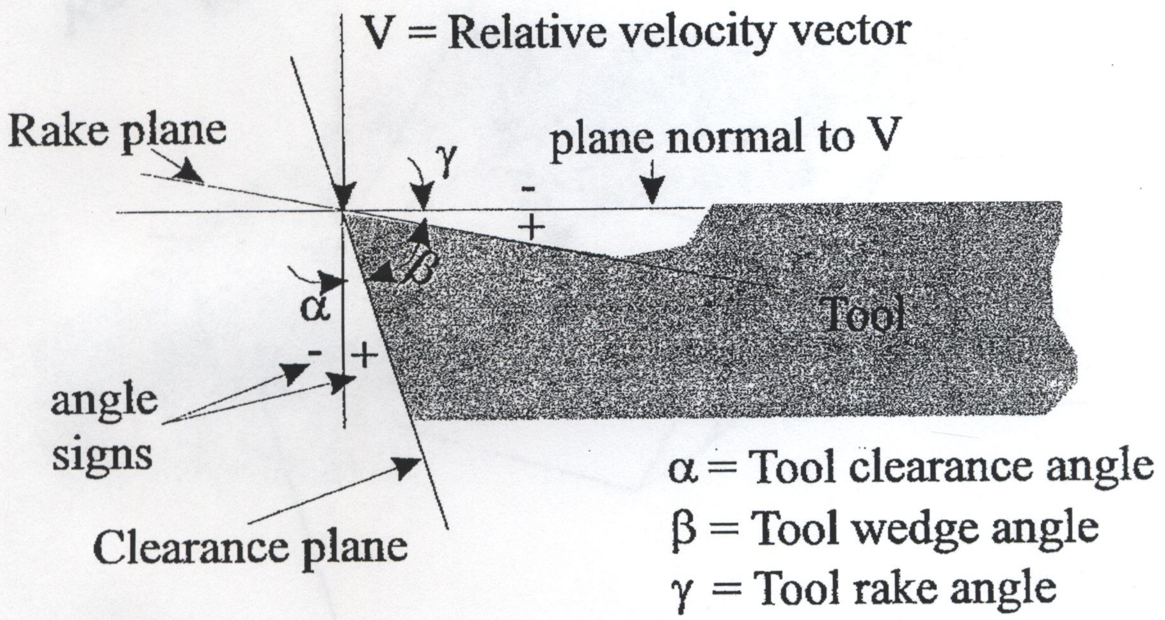
G code	Group	Function	
G00	01	Positioning	
G01		Linear interpolation	
G02		Circular interpolation/Helical interpolation CW	
G03		Circular interpolation/Helical interpolation CCW	
G04	00	Dwell, Exact stop	
G05		High speed cycle machining	
G09		Exact stop	
G10		Data setting	
G11		Data setting mode cancel	
G15	17	Polar coordinate command cancel	
G16		Polar coordinate command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane section	Zp: Z axis or its parallel axis
G20	06	Input in inch	
G21		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G27	00	Reference position return check	
G28		Return to reference position	
G29		Return from reference position	
G30		2nd, 3rd and 4th reference position return	
G31		Skip function	
G33	01	Thread cutting	
G37	00	Automatic tool length measurement	
G39		Corner offset circular interpolation	
G40	07	Cutter compensation cancel	
G41		Cutter compensation left	
G42		Cutter compensation right	
G43	08	Tool length compensation + direction	
G44		Tool length compensation - direction	
G45	00	Tool offset increase	
G46		Tool offset decrease	
G47		Tool offset double increase	
G48		Tool offset double decrease	
G49	08	Tool length compensation cancel	
G50	11	Scaling cancel	
G51		Scaling	

G code	Group	Function
G52	00	Local coordinate system setting
G53		Machine coordinate system selection
G54	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call
G66	12	Macro mode call
G67		Macro mode call cancel
G68	16	Coordinate rotation
G69		Coordinate rotation cancel
G73	09	Peck drilling cycle
G74		Counter tapping cycle
G75	01	Plunge grinding cycle
G76	09	Fine boring cycle
G77	01	Direct constant-dimension plunge grinding cycle
G78		Continuous-feed surface grinding cycle
G79		Intermittent-feed surface grinding cycle
G80	09	Canned cycle cancel/external operation function cancel
G81		Drilling cycle, spot boring cycle or external operation function
G82		Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle

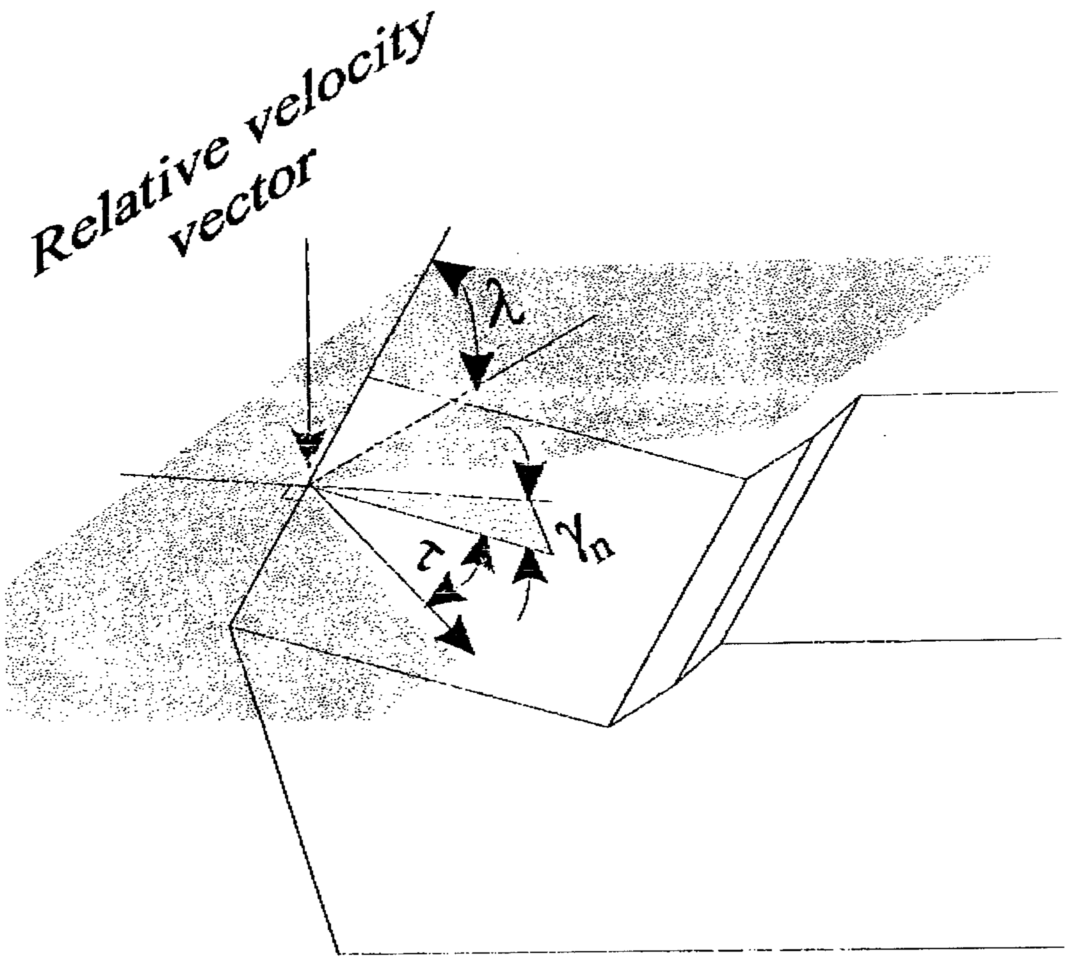
G code	Group	Function
G90	03	Absolute command
G91		Increment command
G92	00	Setting for work coordinate system or clamp at maximum spindle speed
G94	05	Feed pre minute
G95		Feed per rotation
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G98	10	Return to initial point in canned cycle
G99		Return to R point in canned cycle
G107	00	Cylindrical interpolation
G150	19	Normal direction control cancel mode
G151		Normal direction control left side on
G152		Normal direction control right side on
G160	20	In-feed control function cancel
G161		In-feed control function

APPENDIX II

ORTHOGONAL CUTTING



OBLIQUE CUTTING



Cutting angles in Oblique cutting

λ = the angle of obliquity

τ = the chip flow angle

γ_t = the true or effective rake angle