S T A R T U R N / B B C

****

PROGRAMMING, INSTRUCTION &

MAINTENANCE MANUAL

INDEX

Cont/d...

INDEX (Cont/d)

1.

INTRODUCTION

STARTURN is a bench turning centre driven by direct numerical control(D.N.C.) from an external microcomputer. STARTURN is programmed using the I.S.O. format with G & M codes. This, plus the facility to incorporate a milling and drilling attachment, provides a powerful and versatile system for use within a training environment.

The software used to control STARTURN is an easy-to-use, menu-driven package, offering all the facilities required to simulate a full C.N.C. turning centre, with a function to prove out programs by use of Graphical Simulation before the actual turning of the workpiece.

Programs can be easily edited, saved and, as the system will support a printer, hard copies of the program and the Graphical Simulation can be taken.

As with any C.N.C. equipment, the user should have some knowledge of machining practices, and be familiar with shop-orientated terms; this manual has been written under these assumptions.

****

2. INSTALLATION & LUBRICATION

2.1 LIFTING

Starturn can be lifted by 2 people.
Weight: 36 Kilos, Length: 600mm, Height: 250mm, Width: 300mm.

2.2 MACHINE SITING

Starturn can be mounted on a suitably strong desk top, and does not require bolting down.

2.3 CLEANING

Before wiring the machine to the mains supply, first remove all anti-corrosive coatings from the slideways and working parts, Including all bright surfaces, using a kerosene-based cleaner. After cleaning, oil all bright surfaces with light machine oil. Regular cleaning and oiling will ensure a long life for the machine, with a minimum of maintenance.

2.4 ELECTRICAL SUPPLY

The regular electrical mains power supply to the machine is single phase/240v/SOHz. Three phase is not suitable for this product and cannot be supplied.
Connect the mains supply to the supplied adapter.

2.5 LUBRICATION

All oiling and greasing points have been fitted prior to despatch. Before the machine is switched on, oil the ballscrews with the recommended lubricant.
The ballscrews must be oiled weekly to avoid seizure. Add grease sparingly to the headstock bearings, through the grease nipples at the back of the headstock.
All slideways should be lightly oiled daily before movement of the saddle and the tailstock.

Manufacturer	SHELL	CASTROL
Oil	VITREA 60	PERFECTO NN
GREASE	ALVANIA No.3	SPHEEROL AP3

Equivalents of all lubricants are available from other manufacturers.

3. SYSTEM OPERATION

3.1 CONNECTING THE SYSTEM

Before turning on the power to the machine, ensure that the equipment is installed and connected in the following manner: Connect the computer, monitor and disc drive by the method described in the BBC user manual.

Connect the STARTURN to the computer, by means of the 1MHz bus located under the front of the keyboard.

Plug the other end of the lead into the back of the STARTURN machine.

The power supply can now be applied to the equipment to make the system ready for use.

3.2 OPERATION

Insert the system disk into the computer disc drive and press the SHIFT & BREAK keys simultaneously on the computer keyboard this will load the software and display the STARTURN MAIN MENU shown below.

```
STARTURN                          DRIVE 0 PROGRAM =
MAIN MENU                         BLOCKS USED =0 FREE =200

              1)- LOAD PROGRAM FROM DISK
              2)- ENTER NEW PROGRAM
              3)- SAVE PROGRAM TO DISK
              4)- EDIT THE PROGRAM
              5)- CATALOGUE OF PROGRAMS
              6)- EXECUTE PROGRAM
              7)- GRAPHIC SIMULATION
              8)- PRINT PROGRAM
              9)- ASSIGN WORKING DRIVE

                    SELECT OPTION
```

The information now displayed at the top of the screen informs you that the current disc drive is drive number "0".

'PROGRAM =' signifies that, as the system has only just been initialised, then no program is currently in the computer's memory, thus the number of blocks used is 0 and the number of free blocks is 200, which is the maximum number of blocks available for a program.

Option 1 "LOAD PROGRAM FROM DISC" is automatically highlighted with a red background - this highlight will change from option to option as the appropriate key is pressed.

4.1 PROGRAMMING

4.1 PROGRAMMING STARTURN

From the menu, select option, as detailed below:-
"OPTION 2)- ENTER NEW PROGRAM"

Once the component has been planned the program can be entered into the computer by taking option 2 on the STARTURN MAIN MENU. After pressing 2 and RETURN, the following screen will be displayed:-

```
STARTURN                              DRIVE 0 PROGRAM=NEW FILE
EDITOR ADD                            BLOCKS USED= 0 FREE= 200
------------------------------------------------------------
N       G      X/I      Z      F      S      T      M

>-
 -
 -
 -
 -
 -
 -
 -
 -
 -
 ------------------------------------------------------------
 1
 ------------------------------------------------------------
ENTER COLUMN -
     f0-ENTER    f1-HELP    f2-MODE    f3-GOTO
     f4-DELETE   f5-QUIT
CURSOR UP/DOWN MOVES LINE POINTER
```

The information displayed at the top of the screen tells you that the current disc drive is drive number 0. As no program exists in memory, the computer displays the message 'PROGRAM=NEW FILE', indicating the start of a new program. The 'ADD' message means that as the function is 'ENTER NEW PROGRAM', then lines or blocks of program can be added. The number of blocks at this point is 0, so as the maximum number is 200 then this is displayed as the amount of 'FREE' blocks.

The next section of the screen contains the column headings N, G, X/I, Z, F, S, T, M - these are explained in detail on Page 6.

At the bottom of the screen Is displayed a red block containing options corresponding to the red function keys at the top of the computer keyboard. These function keys perform the functions detailed overleaf.

f0-ENTER

This function key causes the data contained within a program line or block to be inserted into the program memory. When in the program editing facility, a block of program can be removed and altered, then reinserted into the program by pressing this key.

f1-HELP

Pressing this key will display a help screen giving access to further, more detailed help screens by pressing the appropriate key - e.g. if information is required on the use of any of the function keys, simply press f1 followed by the desired key. Access to help screens to explain the vaious G & M codes and the meaning and use of the column headings is described later in this section.

f2-MODE

When pressed, this function key causes the program to alternate between the actual insert mode and the overwrite mode, used to edit programs. The current mode is indicated at the top of the screen by the message 'ADD' or 'OVER' - i.e. ADD new program blocks or OVERwrite existing information.
When entering a new program into memory, the mode must be 'ADD'.

f3-GOTO

The f3 key, when pressed, will request a program block number. When entered, the program will be listed on the screen from that point onwards. This search facility saves time when searching through a program for a particular block of information. An error message will be displayed if the block entered is non-existent.

f4-DELETE

This option will delete the block in the program marked by the cursor ">" on the left hand side of the block. This same block is displayed between the two green lines at the bottom of the screen. You will be asked if you are sure you want to delete the block and entry of "Y" will execute the delete, whilst an entry of "N" will exit from the deletion routine.

f5-QUIT

When f5 is entered, the message "ARE YOU SURE?" is displayed. Simply enter "Y" to quit the current program, or "N" to return to the previous stage.

COLUMN HEADINGS

The column headings can be seen in the green band at the top of the screen; they are: N, G, X/I, Z, F, S, T, M. Help screens are available which explain their function. Access to the function is gained by simply entering the column heading followed by the the question mark "?" - e.g. to display the information relating to the N column, input "N?", followed by "RETURN". To exit from a help screen, press "RETURN". A further explanation of the column headings is given below:

N - This column heading represents the number of the block or line of program. The computer automatically allocates the program numbers in increments of 1.

G - The G column containes the G code for the operation being performed. A full list of the codes is given at the back of this manual, but they can also be displayed on the screen by entering "G?", followed by "RETUR". Information on each individual code can be displayed by inputting the code again, followed by a question mark "I" - e.g. if an explanation of the G code G01 is required, enter "G01?" and "RETURN".

X/I- This column heading has various uses, each one explained below:-
Firstly it contains the X co-ordinate of the tool movement, which is perpendicular to the workpiece.
X can also be used to give the depth of cut for a threading cycle (G33), the dwell time in seconds (G04), the subroutine number (G28, G65) and the number of repeats to be performed in a do-loop.
The column also has another value, I, which has only two different uses:
First it is used for the radius parameter in a G02 or G03 circular interpolation movement, and secondly the I column can contain the pitch for threading cycle (G33).

Z - The Z column contains the parameter for the distance the tool will move in the Z axis (parallel to the workpiece). In the case of a threading cycle (G33) Z represents the length of the thread.

F - The F column contains the feedrate of the tool (maximum feedrate 999mm/min), which is taken in the direction of tool movement, and can be either mm/min or inches/min, depending on which units of measurement have been adopted for the program in question: G70 imperial or G71 metric.
As Starturn uses metric ballscrews to drive the slides, this is the prime unit of measurement; however, these units can be changed at any time during a program, if required.

COLUMN HEADINGS (Cont/d)

S - The S column contains the spindle speed specified in revs/min. STARTURN's speed range is infinitely variable between 0-2000rpm.

T - The T column contains the tool number to be used for the block being programmed. Once a tool has been selected, it will remain the same throughout the rest of the program, or until it sees a different tool number in the T column. The tool number can only be changed within the program by using a G50 datum movement.

M - The M column holds the value of any M codes present within the program. There are two valid codes that can be used in this column - M00 and M02.
M00 is used to suspend a program execution until the operator intervenes by pressing the space bar.
M02 marks the end of the program. When the computer reads this code, the program will terminate. Although M02 is the end of program statement, all subroutines must be programmed after this point. This is explained in detail under the subroutine section of this manual - Section 4.7.

4.2 PROGRAMMING FUNCTIONS

An explanation of the various programming functions of STARTURN, along with an example, where appropriate, is given below.

4.3 POINT TO POINT G00, G01

The codes G00 and G01 are used to move both axes either individually or simultaneously (linear interpolation).
G00 is a rapid traverse positioning move and is used to position the tool at the correct co-ordinates prior to cutting. A G00 move is always executed at the fastest feedrate (999mm/min or 39.37 inches/min).
An entry of G00 or G01 at the ENTER COLUMN prompt of the program entry screen shown on Page 4 will prompt for X and Z values. These must be typed in turn and RETURN pressed after each one to accept the information into the program.
If the same X or Z values are required in the current block, as are displayed in the previous block, these values will automatically be carried forward by simply pressing "RETURN" - i.e. X and Z values are "MODAL".

4.3 POINT TO POINT G00, G01 (cont/d)

An example of absolute and incremental input is shown in Fig.i.

G01 is a cutting movement, that is to say a movement under a feedrate specified by the programmer. Various inputs are required for a G01. As with a G00, rapid traverse move, the X and Z values are entered in the same manner. A prompt will now ask for a feedrate for the move - this can be any value between 0-999 mm/min or 0-39 inches/min. The next item of data is the spindle speed under the "S" column - this figure can be any value between 0-2000 rpm. Now enter the tool number to be used for the operation, enter T1 for tool 1, T2 for tool 2, etc. Once a tool number has been specified, it will remain the same throughout the program, or until the computer sees a different figure under the T column. The last item of data on the line will be any M code that may be required; either M02 if it is the last block in the program, or M00 if a pause is required at this point; whereupon the prompt "TYPE 'C' TO CONTINUE" will inform you at program run time, as to the procedure for restarting the cycle.
The block of information can now be entered into the program memory by pressing the red function key f0-ENTER - the completed block will now be displayed in the middle of the screen at the relevant line number.

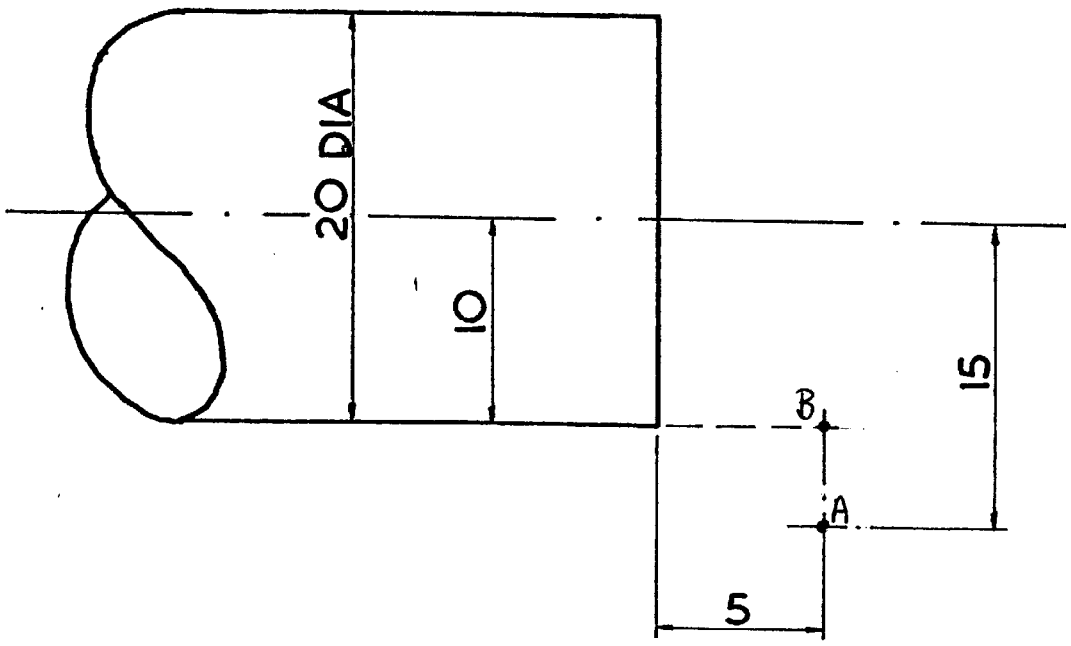
NOTE: If both axes are required to move together, then simply input the required X and Z values on to the same line, then calculate the required vectoral feedrate so both axes arrive at their respective destinations simultaneously.

4.4 CIRCULAR INTERPOLATION - CLOCKWISE G02 & COUNTER-CLOCKWISE G03

Circular interpolation movements are used to produce radii. Two possible arcs are available i.e. clockwise and counter-clockwise. The direction of the arc (CW or CW can be determined by taking a plan view of the machine and by looking in which direction the tool is moving. It can be seen, therefore, that a clockwise arc will produce a convex radius, whereas a counter-clockwise arc will produce a concave form. These are shown in Fig.II.
To produce an arc, adopt the following procedure.
Position the tool in X and Z to the starting point of the arc this is done by means of the previous block - i.e. if the radius is to be produced at block 30 in the program, then block 29 would be a linear move to the starting point of the arc.
The first item of information to be entered is the X co-ordinate. This is the X position for the end of the arc. The Z co-ordinate is next entered, and again it represents the end position of the arc in the Z direction.
A prompt will now appear requesting the I value to be entered, this figure is the radius of the arc being turned and can be any value between 2 and 300mm.
Starturn will allow for the centre point of the arc to be outside the machine limits, providing the end points are within the limits of the machine axis.

FIG I

POINT TO POINT G00, G01



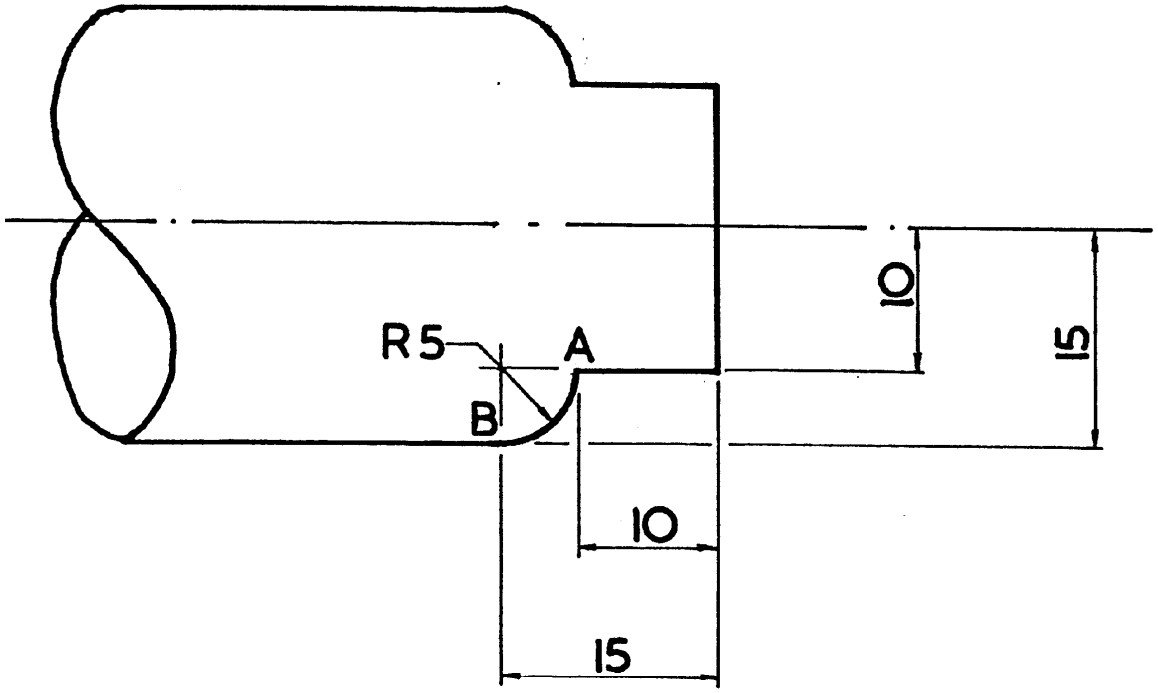
From point A X15 Z5

Point B incremental X-5 Z0

Point B absolute X 10 Z5

FIG II

CIRCULAR INTERPOLATION



ABSOLUTE

With tool at pos. A
X = 10,00 Z = 10,00
circular coordinates
X 15,0 Z -15,0
RAD (I)=5,0 'G02' CW

INCREMENTAL

With tool at pos. A
circular coordinates
X 5,0 Z -5,0
RAD (I)=5,0 'G02' CW

4.5 DWELL G04

The dwell facility halts the movement of the slides for specified number of seconds. The function can be used, for example, when turning a groove or undercut to leave the tool in position for a period of time. On the entry of G04 the only information which need be entered is the X value, representing the time of the dwell required in seconds.

4.6 DO-LOOP (REPEAT LOOP) G73, G06

Should the operator require the same operation to be perfomed a number of times, then the DO-LOOP facility is of prime importance. The function need only be performed once, and the computer told how many times to repeat the operation. To specify a DO-LOOP, adopt the following procedure:-

As DO-LOOPS should be programmed in incremental format, the first block of the repeat procedure will be G91 to change the format to incremental. Secondly, the code G73 START DO LOOP is entered. The only data to be input is the X value, which represents the number of times the loop is to be repeated. The next lines of program represent the operation to be performed, this could be, for example, a roughing down operation, or facing operation. When these blocks of program have been entered, input a G06 END DO to specify the end of the loop procedure. Finally, if the format for the rest of the program is absolute, change the format back by entering a G90. The way in which the function works is that the computer reads the START DO LOOP block and executes the operation to be repeated. When the computer reaches the END DO command, it will return to the start of the loop again. It does this until the correct number of loops has been executed.

In the following example, a bar of 20mm diameter requires turning to a finished diameter of 10mm by 25mm long. The program shown in Fig.III will accomplish the procedure.

In this DO-LOOP the tool moves towards the centre line of the billet, a distance of 1mm, and a cut is taken for a length of 25mm. The tool is then retracted by 0.5mm to avoid leaving trail lines on the work surface. The tool now returns to the Z start position, followed by an X movement of 0.5mm towards the centre line. This operation is then repeated 5 times, leaving the bar at the correct diameter of 1Omm.

The time savings are obvious; rather than having to define each individual tool movement for the 5 passes, which would take 20 blocks of program, the whole procedure is programmed in only 8 blocks.

In the above example it can now be seen why DO-LOOPS are programmed incrementally - an absolute DO-LOOP would simply execute each cut at the same position.

FIG III

```
    PART NO: DO LOOP .............
DESCRIPTION: EXAMPLE DO LOOP .....
             PROGRAM .............
              .....................
              .....................
     AUTHOR: S.CROWTHER ..........
       DATE: 24/04/85 ............
    VERSION: 1.0 ................
```

N	1	G	71	X		Z		I	F		S	T	M
N	2	G	90	X		Z		I	F		S	T	M
N	3	G	50	X	10.000	Z	5.000	I	F		S	T	1 M
N	4	G	0	X	10.000	Z	0.500	I	F		S	1000 T	M
N	5	G	73	X	5.000	Z		I	F		S	T	M
N	6	G	91	X		Z		I	F		S	T	M
N	7	G	0	X	-1.000	Z	0.000	I	F		S	1000 T	M
N	8	G	1	X	0.000	Z	-25.500	I	F	100	S	1000 T	M
N	9	G	0	X	0.500	Z	0.000	I	F		S	1000 T	M
N	10	G	0	X	0.000	Z	25.500	I	F		S	1000 T	M
N	11	G	0	X	-0.500	Z	0.000	I	F		S	1000 T	M
N	12	G	6	X		Z		I	F		S	T	M

4.7 SUBROUTINES G28, G05, G65

The Subroutines function allows for a group of operations to be re-executed more than once within a program, and as many times as required. The Subroutine function must not be confused with the Do loop facility. The example in Fig.IV highlights the differences.

A Subroutine must always be defined at the end of the program, after the M02 End of Program statement. It can then be called into use at any point in the program, by use of the G65 Subroutine Call command.

To define a Subroutine, adopt the following procedure:-

The code G28 is used to declare the start of a Subroutine. When input, you will be required to enter an X value -this represents the Subroutine identity number, which can be any number of your choice.
The next blocks of the program are those to perform the operation required.

NOTE: A Subroutine should be programmed incrementally - when defined in absolute mode it would be executed at the same place each time. Therefore, the first block of program after the G28 should be a G91 incremental.

A code of G05 End Subroutine will signify the end of a Subroutine. The block before this code should contain a G90 to return to the format being used in the rest of the program (if incremental format is being used throughout the program, then this will not have to be done).

The Subroutine, now it is defined, can be called into use at any time within the program, by use of the G65 Subroutine call function, which will require the Subroutine number to be entered. The same Subroutine can be used any number of times, when required.

In the example the Subroutine, when called, will produce two grooves, four grooves are turned by calling up the Subroutine and re-executing it at a different position.

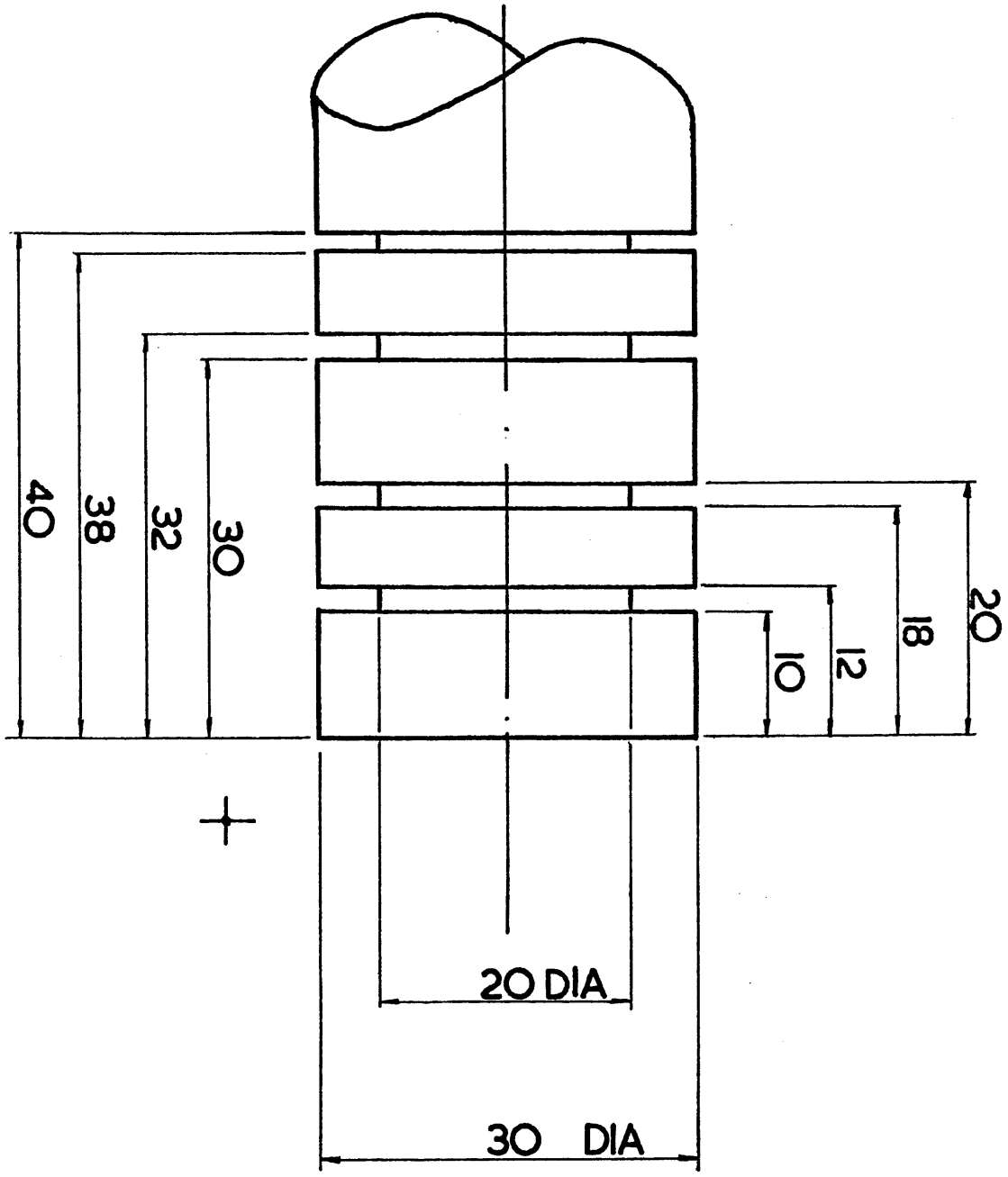
FIG IV

```
PART NO: SUBROUTINE .........
DESCRIPTION: SAMPLE SUBROUTINE ...
             PROGRAM .............
AUTHOR: S.CROWTHER ..........
DATE: 24/04/85 ............
VERSION: 1.0 ................
```

```
N   1 G  71 X          Z          I          F        S        T     M
N   2 G  90 X          Z          I          F        S        T     M
N   3 G  50 X  20.000 Z  15.000 I          F        S        T  1  M
N   4 G   0 X  16.000 Z -12.000 I          F        S   600  T     M
N   5 G  65 X   1.000 Z          I          F        S        T     M
N   6 G   0 X  16.000 Z -32.000 I          F        S   600  T     M
N   7 G  65 X   1.000 Z          I          F        S        T     M
N   3 G   0 X  20.000 Z  15.000 I          F        S   600  T     M  2
N   9 G  28 X   1.000 Z          I          F        S        T     M
N  10 G  91 X          Z          I          F        S        T     M
N  11 G   1 X  -6.000 Z   0.000 I          F   40   S   100  T     M
N  12 G   0 X   6.000 Z   0.000 I          F        S   600  T     M
N  13 G   0 X   0.000 Z  -8.000 I          F        S   600  T     M
N  14 G   1 X  -6.000 Z   0.000 I          F   40   S   600  T     M
N  15 G   0 X   6.000 Z   0.000 I          F        S   600  T     M
N  16 G  90 X          Z          I          F        S        T     M
N  17 G   5 X          Z          I          F        S        T     M
```

FIG V

SUBROUTINE



4.8 THREADING G33

The Threading routine is used by adopting the following procedure:

Firstly a Do-Loop must be set up for the number of cuts, by using G73 Start Do-Loop.

The next block of program is a G33 Threading. The relevant column values are as follows:

X - Represents the depth of thread to be cut.

Z - Represents the length of the thread as a negative value from the datum position at the end of the bar.

S - Represents the spindle speed.

I - Represents the pitch of the thread.

The next block of information changes the format to incremental, by using the G91 code.
The next block is G00 rapid traverse for the increment of cut this will be on the X axis.
The format must then be changed back to absolute mode by use of the G90 code, followed by G06 to signify the end of the threading procedure.
The thread will then be cut to the desired dimensions in the number of Do-Loop passes.

An example program to cut a thread of 2mm pitch, 1.75mm deep and 10mm long, would be as follows:

N	G	X/I	Z	F	S	T	M
1	73	40					
2	33	-2	-10		150		
		1.75					
3	91						
4	0	-0.040	0.00				
5	90						
6	6						

In the above program, block 2 gives a depth of 2mm, a length of 10mm and pitch of thread of 1.75mm. Block number 4 displays the incremental cut of -0.04.

4.9 IMPERIAL UNITS G70

The code G70 signifies that all the tool movements from this point onwards will be in inches. The code can be input at any time in the program. No parameters are required for this code.

4.10 METRIC UNITS G71

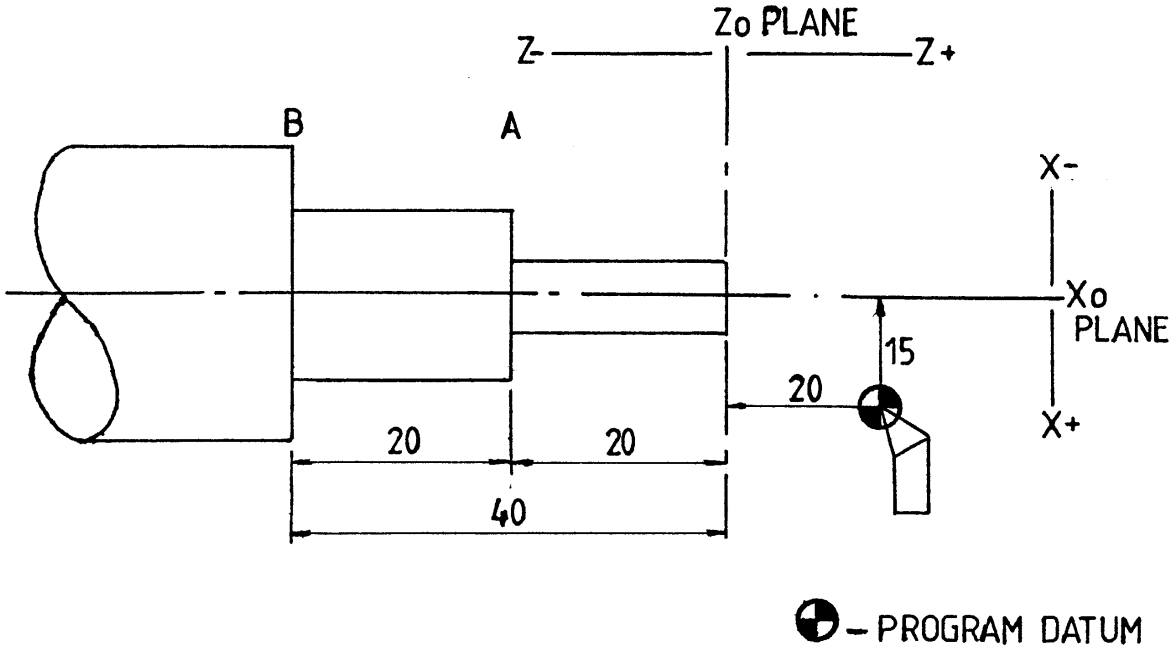
The code G71 siginifies that all the tool movements from this point onwards will be metric. The code can be input at any time in the program. No parameters are required for this code.

NOTE: The units being used within a program can be changed at any time, thus not restricting the user to one measuring format for any one programmed component.

4.11 ABSOLUTE G90, INCREMENTAL G91 FORMATS

Absolute and incremental programming is explained in the following section on basic principles.

4.12 BASIC PRINCIPLES



The above diagram illustrates the terms XO; ZO; Program Datum; Absolute and Incremental, as used in the writing of programs for CNC Machines.

ZO is taken as the end of the workplece.

XO is taken as the centre line of the spindle.

The Program Datum is shown 20mm from the ZO plane and 15mm from the XO plane. Its position is expressed as Z20; X15.

When programming in absolute units and using the program datum for reference, then the face "A" is positioned at Z,-20; the face is positioned at Z,-40.

NOTE: All Absolute Z measurements to the left of ZO are -VE.
All Absolute Z measurements to the right of ZO are +VE.
Similarly for dimensions on either side of XO plane, as shown.

When Incremental programming is used, the co-ordinates are as follows (still working from the program datum):-

Face "A" is positioned at Z,-40.
Face "B" is positioned at Z,-60.
All Incremental Z measurements towards the chuck are -VE.
All Incremental Z measurements away from the chuck are +VE.
All Incremental X measurements towards the centre line are -VE.
All Incremental X measurements away from the centre line are +VE.

4.13 MACHINE AXIS FORMAT

The diagrams below illustrate the plan view of STARTURN. The Z axis runs along the lenqth of the bed, and the X axis along the cross slide at 90° to the bed. The plus and minus signs indicate the direction of the tool.

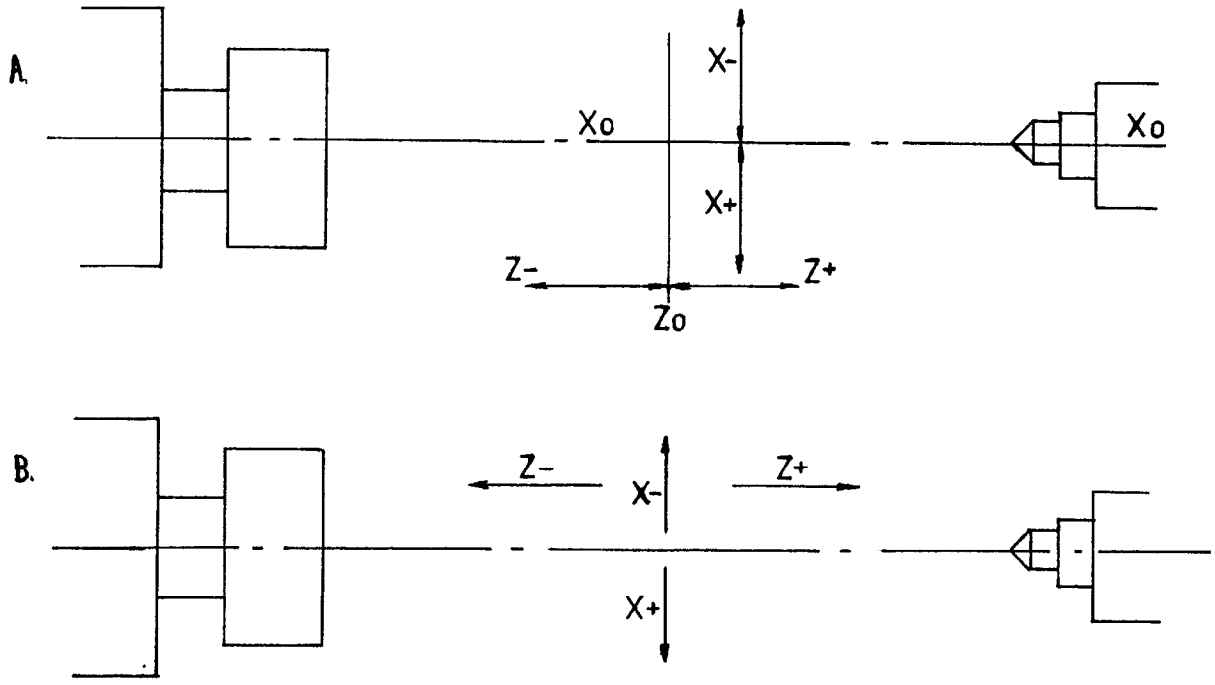
Fig. A. Absolute

Z Axis: to the left hand side of Z0 towards the chuck is Negative.
to the right hand side of Z0 away from the chuck is positive.

X Axis: X0 is on the centre line of the spindle.
away from X0 towards the splash guard the movement is neqative.
and towards the operator from X0 is positive.

Fig. B. Incremental

Z Axis: towards the chuck is negative.
away from the chuck is positive.
X Axis: away from the operator is neqative.
towards the operator is positive.



5. PROGRAM EDITING

5.1 EDITING

This section explains how a program may be edited: explanations will be given on how to add, delete and change blocks of a program.

For a program to be edited, it must first be in the computer's memory. A program that has previously been saved on to disk will have to be loaded - see LOAD FROM DISK section 7.1. Otherwise, if it is a program on which you are currently working, then take option 4 "EDIT THE PROGRAM" from the main menu.

After pressing "4" followed by return, you will be faced with the program input screen, along with the first 10 blocks of the program.
Block 1 of the program will be displayed at the bottom of the screen, between the two green bands. These green bands contain the active block of program, i.e. if a block has to be altered or removed from the program, then it must first be placed in this position. This is done by skipping through the program, using the cursor up/down keys on the computer keyboard. This will move the line pointer down through the program and display the block pointed to between the green bands.
The various programming functions are described in detail below.

5.2 INSERTING PROGRAM BLOCKS

It is often the case that blocks need to be inserted into the middle of a program. The procedure is as follows:

Ensure that the computer is in insert mode - this is denoted by the message "ADD" at the top of the screen. To change the mode simply press the f2-MODE key (red function key) - this is highlighted by the message at the bottom of the screen. Move the block pointer to the block below the one where the new line is to be inserted, by using the cursor up/down keys on the computer keyboard.
Now simply type the new program block and press the f0-ENTER function key. This will insert the block into the program body and the rest of the program will be re-numbered accordingly by the computer.

5.3 DELETING PROGRAM BLOCKS

Blocks can be deleted from programs by placing the line pointer alongside the block to be removed and pressing the f4-DELETE key. Confirm the entry by "Y" or "N" when prompted by the computer. The block of program pointed to will then be removed and all other blocks will be renumbered automatically by the computer.

5.4 AMENDING PROGRAM BLOCKS

Errors in individual blocks of program can be amended by adopting the following procedure:

Ensure that the current mode is the OVERWRITE mode - this is indicated by the message "ADD" or "OVER" at the top of the screen in this case it must read "OVER".
Place the line pointer to the block of program to be amended, by using the cursor up/down keys. The message "ENTER COLUMN" will now be shown.

Simply type the new column heading and the relevant information to be changed, then press the fO-ENTER function key. This will re-insert the amended block into the program in its new form.

in the example below, the value of X requires changing. The block will look like this:

N	G	X/I	Z	F	S	T	M
7	01	6.000	-25.000	30	1000	1	

To change the value of X from 6.000 to 7.000, enter X 7.000 and press RETURN. This will change the data. Pressing f0-ENTER will re-insert the block into the program.

6.0 GUIDELINES

Following these guidelines will assist the operator and reduce costly programming errors.

1. Start each program with the UNITS andlor the FORMAT on pages 1 & 2 of the Program and follow these with the Program Datum information.

2. Do-loops and Subroutines should usually be programmed incrementally.

3. Remember to return to the format used for the rest of the program at the end of a Do-loop or Subroutine.

4. Subroutines must be entered after the end of the program.

5. Subroutines can be contained within Subroutines or Do-loops (i.e. nested).

6. Do-loops can be nested within Do-loops or Subroutines.

7. Ensure that the feedrate programmed is compatible with the units selected.

8. The minimum feedrate for circular interpolation should be 40m/min or 1.5 inches/min.

9. Any minor arc (less than 180°) either CW or CCW can be programmed in one page. Major arcs (greater than 1800) should be constructed using two minor arcs.

10. Do not program a radius which is less than the value to join the X Z points specified.

11. Units and/or format may be changed during the pages of a program.

12. Metric ballscrews are used - therefore the prime units are mm.

13. Too high a spindle speed, or too coarse a pitch in the threading cycle, will have an erratic effect on the program execution, causing the program to be re-executed from the start.

14. To ensure accurate repetitive re-execution of a program, Reference Tool 0 should be programmed in the last point to point block, as this carries no tool offsets on the X & Z axes.

15. No G00 G01 block should be programmed without a movement on X or Z e.g. in absolute format, if already at X15 Z10, the following page cannot contain both these values.

16. Taper turning on STARTURN is done using the Point to Point function. The maximum ratio between the 2 axes is 35:1 - i.e. 1mm on X to 35mm on Z and vice versa.

17. Write the program down and up-date the written copy as you are editing.

18. Program the tools in sequence, always starting at number 1. Keep the written program and record the tool description on it.

19. THINK!

7. MENU OPTIONS

7.1 OPTION I)- LOAD PROGRAM FROM DISK

This option will load a program previously saved on to the disk, back into the computer's memory. By pressing "1" followed by return, the following message will be displayed:-

ENTER PROGRAM NAME-?

Input the name of the program to be loaded. If the name does not exist, then you will be informed by the message "FILE DOES NOT EXIST". Press RETURN to display the main menu, or enter the correct file name.
When the chosen program has been successfully loaded, the PROGRAM INFORMATION BLOCK will be displayed - see section 7.2 for more information.
Should the program to be loaded be stored on a different disk, then enter "1" to load from disk. When requested to enter the program name, insert the disk into the drive and type the program name to be loaded. Once loaded, the master disk must be re-inserted to allow normal operation to continue.

7.2 OPTION 3)- SAVE PROGRAM TO DISK

This option will save a program currently in the computer's memory on to a disk for permanent storage, from where it can be recalled at any time by using Option 1)- LOAD PROGRAM FROM DISK. On entry of "3" the following message will be displayed:

ENTER PROGRAM NAME-?

Type in the name of your choice, up to a maximum of 7 characters. You will then be requested to fill in the data on the PROGRAM INFORMATION BLOCK. An example is shown below:-

```
PART NO:        12345...................
DESCRIPTION:    SCREWED SHAFT ..........
                BILLET LEN 8OMM ........
                BILLET DIA 20MM ........
                ........................
AUTHOR:         J. KNOWLES .............
DATE:           03/12/84 ...............
VERSION:        1.11 ...................

  USE CURSOR KEYS, RETURN AND DELETE TO
     EDIT PROGRAM INFORMATION BLOCK
      PRESS 'ESCAPE' WHEN FINISHED
```

The information entered on to the PROGRAM INFORMATION BLOCK, can be anything of your choice. In the example on the previous page, it has been used to record the information relating to the billet to be used when running the program. This information is always displayed when the program is recalled, by using Option 1)- LOAD PROGRAM FROM DISK.
Should an attempt be made to save programs when there are none in the computer's memory, then the following message will be displayed

NO PROGRAM CURRENTLY ACTIVE
PRESS ANY KEY TO CONTINUE

Should a program require saving on to a disk other than the master disk, enter "3", and when requested to enter a program name, insert the desired disk on which the program is to be stored. When the program has been successfully saved, re-insert the master disk, so normal operation can continue. NOTE: When saving programs on to a disk, the error message "DISK ERROR 198" may be displayed. This only means that the disk is full and no more programs may be stored.

7.3 OPTION 5)- CATALOGUE OF PROGRAMS

The CATALOGUE OF PROGRAMS function will display on the screen all the programs stored on any one disk. on an entry of "5" the programs will be displayed in the format shown below:

P.TEST	001312	001312	000264	OAE
P.SHAFT2	001312	001312	000240	OAB
P.AC1002	001312	001312	00032A	09E

In the above example it can be seen that 3 programs are stored on the disk in question, and the program names are TEST, SHAFT2 and AC1002. The prefix P which appears before all program names, is allocated by the computer for its own use and does not in any way affect the operator.
At the bottom of the screen will be displayed the message:

DELETE A PROGRAM Y/N?

This option will now allow selective programs to be removed from the disk by entering "Y", whereupon you will be requested to enter the name of the program to be removed. Once this has been done, an up-dated catalogue of programs will be displayed, omitting the file which has just been removed. Continue in this manner, until all the desired files have been removed.
Should no files need deleting from the disk, simply enter "N", and the STARTURN Main Menu will be displayed on the screen. To obtain a catalogue of programs from a different disk, enter "5" for the catalogue option, take out the curent disk and replace it with the one to be checked and re-execute the catalogue command. When this has been done, the master disk must be re-inserted into the drive, to enable normal operation to continue.

7.4 OPTION 7)- GRAPHIC SIMULATION

This option allows a program to be proved by running it on the computer, before actually using the machine. After pressing "7" then return, the framework for the Graphic Simulation will be displayed on the screen, as shown below:

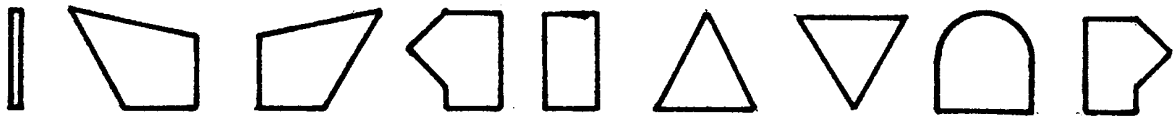
```
-------------------------------------------------------------
     TOOL PATH SIMULATION
-------------------------------------------------------------
     ENTER BILLET DIMENSIONS
     LENGTH (MM)?




-------------------------------------------------------------
     KEY 0      KEY 1      KEY 2      KEY 3      KEY 4
     STEP       <SCALE     >SCALE     AGAIN      ABORT
-------------------------------------------------------------
     KEY 5      KEY 6      KEY 7      KEY 8      KEY 9
     PRINT      START
-------------------------------------------------------------
```

The computer is now requesting the input for the length of the billet to be used in the Graphic Simulation. This figure is entered in metric. The diameter in the next item of data to be entered - again, in metric units. After the billet parameters have been set and accepted by the computer, the tool library screen shown below will be displayed.

```
ASSIGN TOOL NUMBERS
TO REOUIRED SHAPES
PRESS:-
ESCAPE - WHEN FINISHED
SPACE BAR TO CLEAR
RETURN - MOVE TO NEXT
```



The tool numbers represented in the program are now input. The cursor is automatically placed under the first tool, which is Tool 0, or the reference tool - see section 8.1.
The rest of the tools can now be given a number by moving the cursor under the appropriate tool profile, using the space bar and entering the number.
When all the tools have been selected, the ESCAPE key must be pressed to exit the tool selection process.
The screen will now show a representation of the billet in yellow and the chuck jaws in red.
The grid gives an indication as to the length of the billet, the size of which is given at the top of the screen.
Also shown are the X0 and Z0 relative to the end of the billet.

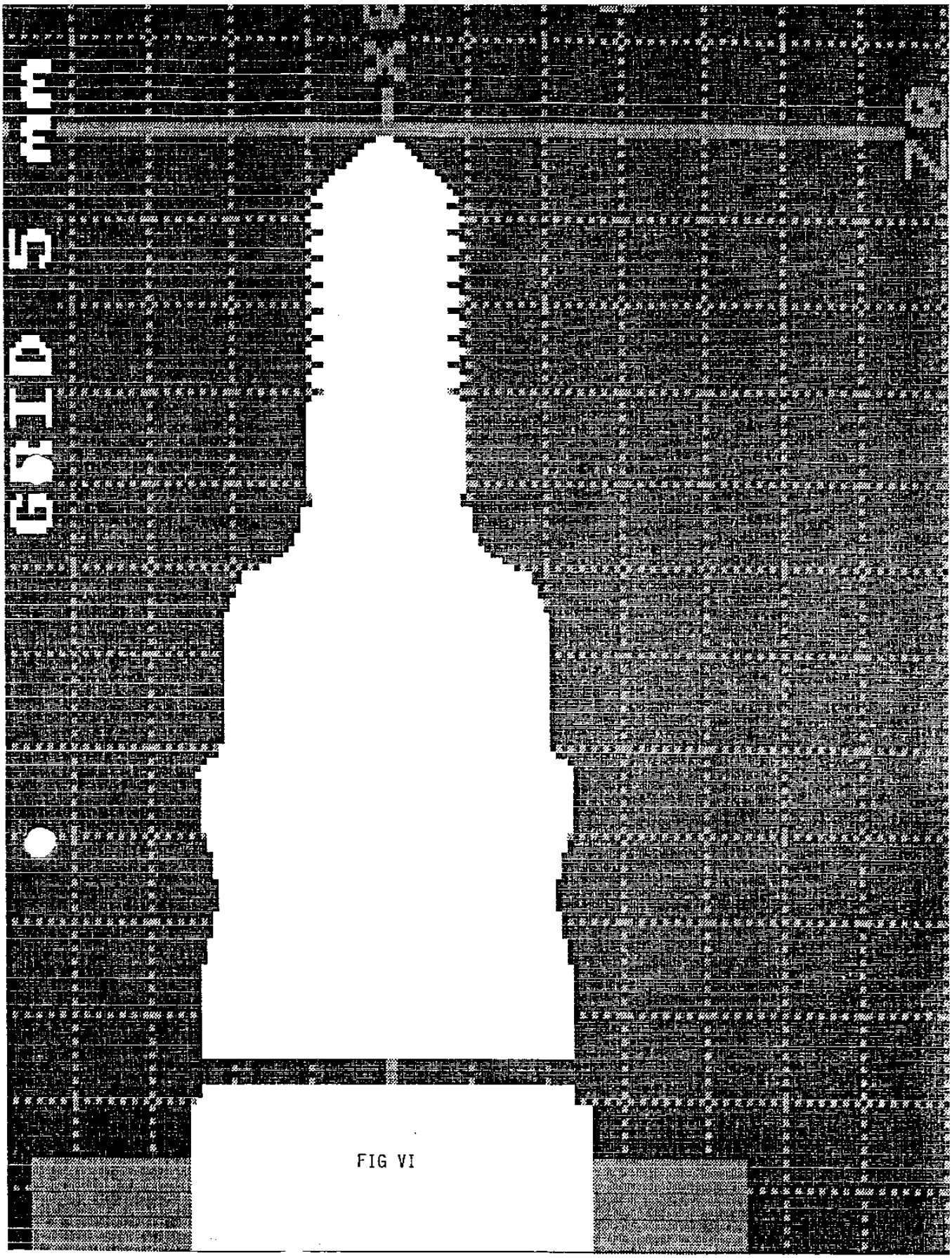


FIG VI

7.5 FUNCTION KEYS

The bottom section of the screen contains 10 boxes, representing the 10 red function keys on the computer keyboard. Their usage is described below.

"f0" - When pressed, this key will allow for the program to be executed in single step mode. This will be denoted by the message 'STEP MOW' at the bottom of the right hand screen window. Pressing the key once more will revert back to the continuous mode. When using the single step mode, each operation can be executed by pressing the space bar.

"fl" - When pressed, this key allows for the Graphic Simulation to be scaled down. Each time the key is pressed, the billet is reduced in size and the dimension of the grid is increased. If the fl key is pressed during the running of the simulation, then the process will be restarted at the new scale.

"f2" - The f2 function key acts as function key fl; however, in the case of f2, the Graphic Simulation is SCALED UP.

"f3" - This function key will halt the running of the Graphic Simulation and allow for new billet length and diameter parameters to be entered. The procedure can then continue in the normal manner, as described earlier.

"f4" - The f4 key will abort the simulation process and return control to the Starturn Main Menu.

"f5" - Key f5 will produce a screen dump of the component after the Simulation has been run. An example of the screen dump is given overleaf. As the print is being produced, it can be stopped by pressing the "ESCAPE" key, which will return you to the start of the Graphic Simulation process.

"f6" - Pressing the f6 function key will start the Graphic Simulation process. Once pressed, the machine datum position will be displayed at the correct location on the screen, together with the profile of the tool beinq used first. The block by block information on the right hand screen window will be constantly up-dated as the simulation progresses. This data can be examined more closely by going into the single step mode by pressing the function key f0. The format for the data displayed on the screen is shown on page 23.

THE RIGHT HAND WINDOW OF THE SCREEN DISPLAYS THE FOLLOWING:-

N = REPRESENTS THE CURRENT BLOCK NUMBER

G = REPRESENTS THE G CODE BEING USED IN THE BLOCK

X = REPRESENTS ANY X CO-ORDINATE

Z = REPRESENTS ANY Z CO-ORDINATE

I = REPRESENTS ANY I CO-ORDINATE

F = REPRESENTS THE FEEDRATE ADOPTED FOR THE BLOCK

S = REPRESENTS THE SPINDLE SPEED FOR THE BLOCK

T = REPRESENTS THE TOOL NUMBER FOR THE BLOCK

M = REPRESENTS ANY M CODE

FORMAT = REPRESENTS THE FORMAT (ABSOLUTE OR INCREMENTAL)

UNITS = REPRESENTS THE UNITS OF MEASUREMENT FOR THE PROGRAM

TOOL REPRESENTS THE TOOL NUMBER

FEED = REPRESENTS THE FEEDRATE

SPEED = REPRESENTS THE SPINDLE SPEED

THE NEXT LINE OF INFORMATION GIVES A BRIEF DESCRIPTION OF THE OPERATION BEING PERFORMED - E.G.

P TO P (POINT TO POINT)
THREADING
LINEAR
C CIRCLE (CLOCKWISE CIRCULAR)

Keys f7, f8 and f9 have no use at this point and are left blank for future development.

When the Graphic Simulation has been completed, it is a useful exercise to re-run it, using different billet parameters -e.g. try entering a larger diameter bar or longer length of billet to see what results are obtained. This will stress to students the importance of using the correct billet sizes, when the component is actually being manufactured.

7.6 OPTION 8)- PRINT PROGRAM

This option from the Starturn Main Menu will produce a hard copy printout of the program in the computer's memory. The program and the program information block will be printed out and will also appear on the screen. An example is shown overleaf.

NOTE: This function can only be executed if a printer is attached to the system, otherwise the program will 'hang'; and the system can only be 're-booted' by pressing SHIFT and BREAK simultaneously on the computer keyboard.

7.7 OPTION 9)- ASSIGN WORKING DRIVE

This option from the Starturn Main Menu is used when two disc drives are connected to the system. One drive can be used for the master disk, and the programs can be saved and loaded from the other drive - i.e. the master disk can be used in drive 0 and programs stored and loaded from drives 1,2, or 3.
The working drive assigned by Option 9 is the drive containing the programs and not the master disk.
If an attempt is made to save programs to the wrong drive, then the message "DRIVE FAULT 14 AT 00/C" will be displayed, together with the message "PRESS SPACE BAR TO CONTINUE".
The correct drive can now be assigned and the system will operate normally.

7.8 OPTION 6)- EXECUTE PROGRAM

Once the program has been written and proved to be correct by using the Graphic Simulation (see section 7.4), it can then be executed on the machine itself.

NOTE: The Execute Program option can only be entered if the computer and machine are connected; if they are not, then control is returned to the Starturn Main Menu.

After pressing "6" followed by return, you will be presented with the MANUAL OPERATION screen shown on Page 25.

The X and Z figures shown on the screen do not, at this point, relate to the actual position of the tool - this will be set during the toolsetting procedure.

Information is displayed on the screen as to the feedrate of the tool, the tool number, the units being used and the type of jog mode active at any time.

The red function keys at the top of the computer keyboard are represented in the bottom half of the screen, and are explained in section 7.9.

```
        PART NO: TEST I ..............
    DESCRIPTION: TEST PIECE ONE ......
                 BAR DIA.19mm ........
                 BAR LEN 60mm ........
                 FREE CUTTING M/S ....
          AUTHOR: NEIL SMITH ..........
            DATE: 6/3/85 ..............
         VERSION: 1 ...................
```

FIG V11

N 1	G 71	X		Z		I		F	S	T	M
N 2	G 90	X		Z		I		F	S	T	M
N 3	G 50	X	12.000	Z	5.000	I		F	S	T	M
N 4	G 0	X	9.500	Z	0.200	I		F	S	T	M
N 5	G 91	X		Z		I		F	S	T	M
N 6	G 73	X	3.000	Z		I		F	S	T	M
N 7	G 0	X	-1.000	Z	0.000	I		F	S	T	M
N 8	G 1	X	0.000	Z	-15.200	I		F 40	S 500	T	M
N 9	G 0	X	0.000	Z	15.200	I		F	S	T	M
N 10	G 6	X		Z		I		F	S	T	M
N 11	G 90	X		Z		I		F	S	T	M
N 12	G 0	X	5.000	Z	0.200	I		F	S	T	M
N 13	G 1	X	6.200	Z	-1.000	I		F 40	S 500	T	M
N 14	G 1	X	5.000	Z	0.200	I		F 500	S	T	M
N 15	G 1	X	4.500	Z	0.200	I		F 40	S	T	M
N 16	G 1	X	6.000	Z	-1.500	I		F	S	T	M
N 17	G 1	X	6.000	Z	-15.000	I		F	S	T	M
N 18	G 2	X	9.000	Z	-18.000	I	3.000	F	S	T	M
N 19	G 1	X	9.000	Z	-20.000	I		F	S	T	M
N 20	G 0	X	10.000	Z	-20.000	I		F	S	T	M
N 21	G 50	X	12.000	Z	5.000	I		F	S	T 2	M
N 22	G 0	X	8.000	Z	0.500	I		F	S	T	M
N 23	G 73	X	44.000	Z		I		F	S	T	M
N 24	G 33	X	-2.000	Z	-10.000	I	1.750	F	S 150	T	M
N 25	G 91	X		Z		I		F	S	T	M
N 26	G 0	X	-0.040	Z	0.000	I		F	S	T	M
N 27	G 90	X		Z		I		F	S	T	M
N 28	G 6	X		Z		I		F	S	T	M
N 29	G 50	X	12.000	Z	5.000	I		F	S	T 3	M
N 30	G 0	X	12.000	Z	-22.000	I		F	S	T	M
N 31	G 0	X	9.500	Z	-22.000	I		F	S	T	M
N 32	G 1	X	-0.200	Z	-22.000	I		F 20	S 500	T	M
N 33	G 0	X	12.000	Z	-22.000	I		F	S	T	M
N 34	G 50	X	12.000	Z	5.000	I		F	S	T 0	M 2

7.9 OPTION 6)- EXECUTE PROGRAM (cont/d)
7.10

MANUAL OPERATION SCREEN

```
      STARTURN                                DRIVE 0 PROGRAM=TEST3
                      MANUAL CONTROL
         X             Z                        SPINDLE SPEED
       12.342      -23.456                           000
     ------------------------------------------------------------------
      FEED RATE = 400 MM/MIN                        TOOL 0
      MM UNITS                                CONTINUOUS JOG
      FO=JOG MODE                             Fl=IMPERIAL/METRIC
      F2=FEED<                                F3=FEED>
      F4=EXIT MANUAL                          F5=EXECUTE PROGRAM
      F6=TOOL OFFSET                          F7=TOOL NO
      F8=SET ZO                               F9=SET X0
                 USE CURSOR KEYS TO POSITION TOOL
```

7.11 FUNCTION KEYS

"f0" - JOG MODE. Pressing this key will alternate the jog between CONTINUOUS JOG and SINGLE STEP JOG. In continuous mode, when the computer cursor control keys are pressed, the slides will move until the key is released. In single step mode, as the cursor key is pressed, the slide will only move 0.005mm per step, no matter how long the key is depressed.

"fl" - IMPERIAL/METRIC. Pressing the f1 function key will alternate the units between metric and inches. When pressed, the relevant units of measurement will be displayed along with the corresponding values, under the X and Z columns.

"f2" - FEED< . This function is used in conjunction with the CONTINUOUS JOG MODE - i.e. it changes the feedrate from fast to slow and is infinitely variable from the two extremes of feed. As the key is depressed, the constantly-changing feedrate will be displayed on the screen. Release the key at the desired feedrate.

"f3" - FEED> . This function is the opposite of "f2" FEED< , in that it alters the feedrate from slow to a faster rate. As the key is depressed, the changing feedrate can be seen on the screen. Again release the key at the desired feedrate.

"f4" - EXIT MANUAL. Pressing this key will exit from the manual section of the process and return to the Starturn Main Menu. In effect, It aborts the procedure.

7.9 FUNCTION KEYS (cont/d)

"f5" - EXECUTE PROGRAM. This function key will exit the Manual section of the process and enters the Program Execution section itself.This will be described in section 8.3.

"f6" - TOOL OFFSET. This function will display the tool offsets for the tools selected in the program. After the program has been run, these offsets can be altered to provide fine dimensional adjustments to ensure that the second component turned is the correct size. This function is described in detail in Section 8.

"f7" - TOOL NO. The f7 function allows for the various tools selected in the program to be set for their individual X and Z zeros.- see Section 8.2.

"f8" - SET ZO. This function sets the ZO (taken at the face of billet) for the tool selected in the f7 Tool No Function.

"f9" - SET XO. This function sets the XO (takenj at the diamete of the bar) for the tool selected in the f7 Tool No Function.

8. OFFSETS

8.1 TOOL OFFSETS

Before the program can be run, each tool to be used in the program has to be set for its own X Zero and Z Zero. This is because each tool varies in the length and position when situated in the toolpost.

The first tool to be set is always tool 0. This acts as a reference tool and theoretically should not be used for cutting, as it may get broken and the offsets for the other tools will therefore become useless.

The same tool can, however, be used as tool 0 and tool 1, as both carry the same offset. Therefore, tool 0 can be set; and, as the same tool is to be used as tool 1, the latter does not require the toolset procedure to be executed.

8.2 SETTING THE OFFSETS

To set the tool offsets, the following procedure must be adopted.

Ensure the billet is secure in the chuck and that the tools are secure in their toolholders.

Select the first tool to be set - tool 0 - by pressing the f7 Tool NO. function key. Instructions will be given on the screen to fit the tool in the toolpost and press "C" to continue.

The Z Zero plane now needs to be set by moving the tool up to the end of the bar, using the cursor control keys. In the CONTINUOUS MODE, this process can be carried out at a fast feedrate.

As the tool nears the billet, the feedrate will need to be reduced to avoid tool collision. As the tool is almost in contact with the billet, the JOG MODE can be changed to SINGLE STEP JOG by pressing the fO JOG MODE key.

It is recommended that a pass is taken across the face of the billet. This process ensures a square reference face. Press the red function key f8 on the computer keyboard to set that plane as zero. At this point, the Z dimension on the screen will display "000".

The next process is to take a skim along the outside diameter of the billet, again by using the computer cursor control keys. At this stage, press the f9 SET XO function key, whereupon you will be requested to input the diameter of the newly-turned section of the billet. Therefore, an accurate measurement must be taken and input into the computer. The X dimension will now display half of the value entered, as STARTURN operates on a radial measurement system. Tool 0 and tool 1 are now set in relation to the outside diameter and face of the billet.

To set the next tool, press the f7 TOOL NO. function key on the computer keyboard. Again you will be requested to fit the chosen tool in the toolpost and press "C" to continue. The above tool-setting process can be repeated for all the tools used in the program.

8.3 EXECUTING THE PROGRAM

After all the tools have been set by using option f7, as previously described, the program can be executed by pressing the f5 function key (EXECUTE PROGRAM). The following will then be displayed on the screen:

```
STARTURN                                  DRIVE 0 PROGRAM = XXXXX
                    PROGRAM EXECUTION
     X                    Z                       SPINDLE SPEED
   999.9                999.9                           000
          DESTINATION                               PROGRAMMED
   999.9                999.9                           500
-------------------------------------------------------------
FEED RATE = 0 MM/MIN TOOL 0

MM UNITS

ABSOLUTE MODE

BLOCK 3 G50 - PROGRAM DATUM

              ESCAPE ABORTS fO = STEP/CONTINUOUS
```

This is the data related to the program being executed. The message shown on the last line means that pressing the "ESCAPE" key on the computer keyboard at this stage will abort the Execution procedure and return to the Starturn Main Menu.

The key fO will now alternate the operation between SINGLE STEP and CONTINUOUS MODE. When the mode of operation has been selected, press "C" to continue.

The computer will look for the first tool number in the program and prompt the message "FIT TOOL 1 'TYPE C TO CONTINUE". Fit the correct tool number in the toolpost and press "C" to continue with the remainder of the procedure.*

The next message to be displayed at the bottom of the screen will prompt you to set the spindle speed to the programmed value and press "C" to continue.

The programmed spindle speed is shown at the top of the screen. start the spindle by pressing the green spindle start key on the front of the STARTURN cabinet and adjust the speed to the programmed speed by use of the variable speed control knob located on the front of the cabinet (next to the start/stop controls). As the speed is increasing, the actual speed will be constantly displayed over the programmed speed. When the two are equal, press "C" , and the program will execute.

*Each time a tool change is encountered, the above tool change procedure will need to be repeated.

8.4 TOOL OFFSETS/SECOND OFF

After the first component has been produced and inspected, any diametrical discrepancies can be adjusted for any other components to be produced by adjusting the tool offsets.

To inspect the current offsets, press "6" - EXECUTE PROGRAM from the Starturn Main Menu. This will display the screen shown in section 7.8. The offsets can be listed on the screen by taking the F6 - TOOL OFFSET function. The following screen will be displayed:-

```
STARTURN                                DRIVE 0 PROGRAM XXXX

                    TOOL OFFSETS
                    1)- LOAD TOOL OFFSETS
                    2)- SAVE TOOL OFFSETS
                    3)- EDIT TOOL OFFSETS
                    4)- DIRECTORY OF OFFSET FILES
```

The above options on the menu are explained in detail below.

1)- LOAD TOOL OFFSETS

This option will allow you to load offsets, previously saved on disk, back into the computer. Thus if the same tooling and billet are used to execute the program at a later time, the tool offsets need not be set, but re-loaded from disk. Press 1 and Return.

2)- SAVE TOOL OFFSETS

This option will allow you to save the offsets set before the program execution, on to the disk. Therefore, providing the same tooling and billet are used to re-execute the program at later time, the tool offsets do not require re-setting. When saving the tool offsets, you will be requested to enter a file name under which the offsets can be saved.

3)- EDIT TOOL OFFSETS

Depressing key 3 will display the X and Z tool offsets. Below is a typical example:-

```
              UNITS - MM
TOOL          X           Z
0             000.00      0000
1             -002.35     0.10
2             10.15       -20.75
3             1.00        0.10
4             0.000       0.000
5             0.000       0.000
6             0.000       0.000
7             0.000       0.000
8             0.000       0.000
9             0.000       0.000
        ENTER TOOL NUMBER
```

8.5 TOOL OFFSETS/SECOND OFF (cont/d)

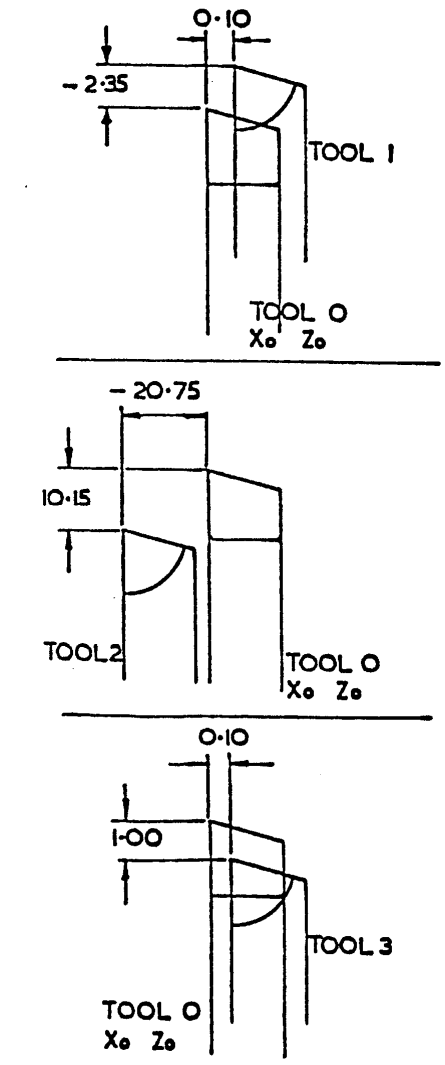
If, after producing several components, a diameter is incorrect, a tool offset can be edited to compensate for tool wear. For example if tool 2 (see example on previous page) is used to finish the component, and it is found to be lmm oversize, then the offset value for X must be INCREASED by 0.5mm, giving a new value of 10.65mm. Enter the tool number as "2" - this will then highlight the existing tool offsets by changing the data to blue and flashinq characters. The new value can now be input for the X offset.

The editing procedure explained on Page 29 can be used to alter lengths of components by editing the Z offset.

Once the offsets have been amended, return to the menu by pressing "RETURN".

The new offsets can be saved to disk by use of option 2)- SAVE TOOL OFFSETS.

TOOL OFFSET REPRESENTATION



9. G & M CODE LISTINGS

G00	POSITIONING (RAPID TRAVERSE)
G01	LINEAR INTERPOLATION
G02	CIRCULAR INTERPOLATION C'WISE
G03	CIRCULAR INTERPOLATION C C'WISE
G04	DWELL
G05	END SUBROUTINE
G06	END REPEAT LOOP
G28	SUBROUTINE
G33	THREADING
G50	PROGRAM DATUM
G65	SUBROUTINE CALL
G70	IMPERIAL UNITS
G71	METRIC UNITS
G73	REPEAT LOOP
G90	ABSOLUTE
G91	INCREMENTAL
MOO	PROGRAM STOP
M02	END OF PROGRAM