

SECTION 35

IBM PROGRAMMING

SECTION 36

INSTALLING THE SOFTWARE

When supplied the package will consist of a 5¼" floppy disc which must be installed onto the IBM computer being used. There are two methods of program installation depending on whether the software is to be run on a hard disc or 5¼" floppy disc drive system.

Installation on this disc system.

- i) Put the supplied STARTURN disc in Drive 'A'
- ii) Type A: Install PRESS ENTER
- iii) When prompted type 'C' to install on the hard disc.

To run the software simply type 'STARTURN' and providing the dongle is correctly situated the software will run.

- i) Prepare a formatted blank disc containing disc operating system (DOS). This is done by using a formatting utility disc and typing 'FORMAT A:/S
- ii) Insert the STARTURN master disc and type A: install. You will be prompted by the computer throughout the installation procedure.
- iii) Take a backup copy of the newly installed disc and dfile the original disc in a safe place.
- iv) To run the software type A: STARTURN (The drive letter may change depending on the disc drive being used).

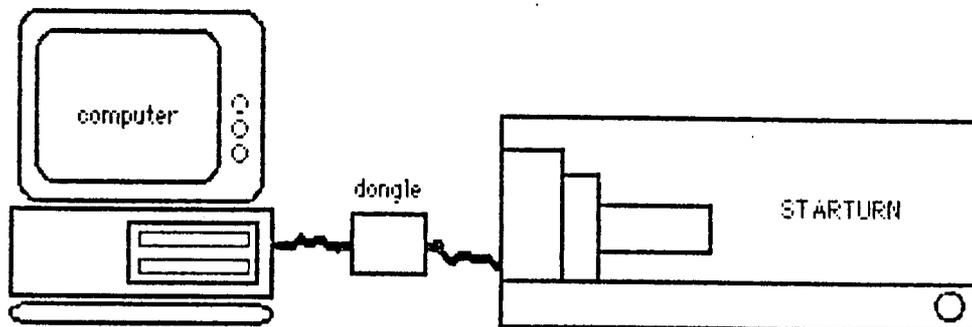
SYSTEM OPERATION

37.1 CONNECTING THE SYSTEM

Ensure all electrical supply is turned off.

The Starturn software has been designed to run on the IBM PC (XT or AT versions). The system requires that an IBM Asynchronous serial card configured for Port 0 (com1) be inserted in a free slot in the computer. An equivalent serial card can be used, and the switch settings to set the card to port 0/ will be found in the manual supplied with the serial card.

The Software Security Device (Dongle) is connected to the serial card via the ribbon cable, the ribbon cable from the other side of the security device connects to the cable supplied with the machine, this in turn plugs into the 'computer link' socket located at the back of the machine. (See Diagram).



SECTION 38

OPERATION

Insert the STARTURN disk into the computer disc drive (Drive A) and boot the system by pressing CTRL, ALT, DEL keys at the same time and the software will then automatically load, and display the main menu shown below.

<u>EDITOR</u>	<u>PROGRAM -----:</u>	<u>MAIN MENU</u>
		: Please select one of the
		: following:
		: Edit cnc program
		: Disc utilities
		: New program
		: Print program listing
		: Manual control
		: Simulation
		: Simulation settings
		: Exit Starturn system
		:
<u>N001</u>		: Press 'f1' for help

This screen is always the first to be displayed when the software is started, and it gives access to all of the STARTURN functions, each of which are displayed on the screen within the right hand window.

By pressing the UP or DOWN arrow keys on the numerical keypad, section of the computer keyboard the various functions are highlighted with a white cursor bar, and they can be selected by positioning the cursor on the appropriate function, and, pressing  enter key.

Each of the STARTURN'S FUNCTIONS will be explained in more detail under their individual headings. However, a brief explanation can be displayed on the screen by pressing the f1 key (HELP) found on the left hand side of the computer keyboard.

A list of the functions will be shown in the right hand window of the screen, this information can be scrolled up or down by use of the up and down arrow keys.

To return the display to the main menu, pres the 'ESC' (ESCAPE) key located at the top left of the alphanumeric section of the keyboard.

SECTION 39

ENTERING YOUR PROGRAM INTO THE COMPUTER

Once written the program can be entered into the computer, ready for simulation, execution or printing.

To input the program the 'Edit CNC Program' option from the main menu must be taken.

To select this move the cursor bar by means of the arrow keys \uparrow \downarrow located on the numerical keypad, section of the keyboard, until it rests on the 'Edit CNC Program' function, and press enter .

The screen will display all the relevant G codes in the right-hand window of the screen, along with the editing options at the bottom of the left of screen, as shown below.

EDITOR	PROGRAM	G CODE LIST
		G00 RAPID TRAVERSE
		G01 LINEAR
		G02 CIRCULAR CLW
		G03 CIRCULAR CCLW
		G04 DWELL
		G05 END SUBROUTINE
		G06 END DO LOOP
		G20 PROGRAM SCALE
		G28 SUBROUTINE START
		G33 THREADING REPEAT CYCLE
		G54 PROGRAM OFFSET
		G65 CALL SUBROUTINE
		G70 IMPERIAL UNITS
		G71 METRIC UNITS
		G73 START DO LOOP
		G84 CANNED TURNING CYCLE
		G90 ABSOLUTE FORMAT
		G91 INCREMENTAL FORMAT

N001

the function of the options displayed at the bottom of the screen is as follows.

UP \uparrow DOWN \downarrow

The UP DOWN arrow allows a program to be scrolled on the screen, this allows the program to be listed on the screen at the speed the operator desires. A program block which requires editing can be moved into position by using the arrow keys, these are located on the numeric keypad section of the keyboard.

S SEARCH

The S "SEARCH" function allows a particular block of program to be found and displayed at the bottom of the screen ready for editing.

On an entry of 'S' the operator will be prompted to input the desired block number of program to be found. When detected, the block will be shown at the bottom of the screen.

'ESC' QUIT

Pressing the 'ESC' key at the top left of the alphanumeric section of the keyboard will exit from the 'CNC PROGRAM EDITOR' and return the operator to the main menu. Any program present in the memory of the computer will not be erased during this process.

'INS' INSERT

Pressing the 'INS' key at the bottom right of the keyboard takes the computer into program insert mode, and it is this mode which allows for a program to be entered into the computer,.

On pressing 'INS' the following message will be displayed.

Enter a Gcode or Mcode
or(to put in a comment. 'ESC' to exit

It is now possible to input the CNC part program by entering a G or M code. If M followed by enter is input then a list of available M codes replaces the G code listing.

To make the program easier to understand during future inspection the facility has been built into the system to allow comments to be typed into the program as reminders as to what the functions of the program are doing. The comment facility can be used at the beginning of the program to state the length dia and type of material being cut, etc.

To enter the comment into the program, instead of typing G or M, type '(' bracket, this will allow for the text to be entered, followed by 'RETURN' to accept.

Pressing the 'ESC' key when prompted by the above message will take the operator back a stage to the editing options selection.

When inputting the G and M codes into the computer to build up the program, the screen prompts for the information it requires to complete the block, e.g. if G00 RAPID TRAVERSE is entered by typing G00 followed by ENTER then the computer prompts for the X and Z values. These values can be typed in any order. When complete type 'E' to end the block and go onto the next. The screen input for G00 RAPID TRAVERSE will look like this:-

```
EDITOR          PROGRAM -----:      G CODE LIST
: G00  RAPIDE TRAVERSE
: G01  LINEAR
: G02  CIRCULAR CLW
: G03  CIRCULAR CCLW
: G04  DWELL
: G05  END SUBROUTINE
: G06  END DO-LOOP
: G20  PROGRAM SCALE
: G28  SUBROUTINE START
: G33  THREADING REPEAT CYCLE
: G54  PROGRAM OFFSET
: G65  CALL SUBROUTINE
: G70  IMPERIAL UNITS
: G71  METRIC UNITS
: G73  START DO-LOOP
: G84  CANNED TURNING CYCLE
: G90  ABSOLUTE FORMAT
: G91  INCREMENTAL FORMAT
:
N001 G00 RAPID TRAVERSE
      X          Z
-----
Press one of X Z
E ends editing
```

With all the G & M codes, prompts will appear on the screen to assist in programming. The codes along with their respective prompts are listed below.

PROMPT

G00	RAPID TRAVERSE	X CO-ORDINATE	Z CO-ORDINATE		
G01	LINEAR	X CO-ORDINATE	Z CO-ORDINATE	F FEEDRATE	
G02	CIRCULAR CW	X CO-ORDINATE	Z CO-ORDINATE	F FEEDRATE	R RADIUS
G03	CIRCULAR CCW	X CO-ORDINATE	Z CO-ORDINATE	F FEEDRATE	R RADIUS
G04	DWELL	SECONDS (1 - 99 VALUE RANGE)			
G05	END SUBROUTINE	NO INPUT REQUIRED			
G06	END DO-LOOP	NO INPUT REQUIRED			
G20	PROGRAM SCALE	SCALE (0.01 TO 10 VALUE RANGE)			
G28	SUBROUTINE START	LABEL NUMBER			
G33	THREADING CYCLE	X CORE RADIUS	Z LENGTH	P PITCH	C NUMBER OF CUTS
G54	PROGRAM OFFSET	X OFFSET	Z OFFSET		
G65	CALL SUBROUTINE	LABEL NUMBER			
G70	IMPERIAL UNITS	NO INPUT REQUIRED			
G71	METRIC UNITS	NO INPUT REQUIRED			
G73	STAR DO-LOOP	NUMBER OF TIMES (1 TO 99 VALUE RANGE)			
G84	CANNED TURNING CYCLE	X FINISH RADIUS	Z LENGTH	F FEEDRATE	
G90	ABSOLUTE FORMAT	NO INPUT REQUIRED			
G91	INCREMENTAL FORMAT	NO INPUT REQUIRED			

When a program has been input into the computer the screen will look like the one displayed below:-

EDITOR	PROGRAM DMITEST:	G CODE LIST
N006 M06 TOOL CHANGE	:	G00 RAPID TRAVERSE
TOOL 1	:	G01 LINEAR
N007 G65 CALL SUBROUTINE	:	G02 CIRCULAR CLW
LABEL 2	:	G03 CIRCULAR CCLW
N008 600 RAPID TRAVERSE	:	G04 DWELL
X 0.000 Z-0.500	:	G05 END SUBROUTINE
N009 G65 CALL SUBROUTINE	:	G06 END DO-LOOP
LABEL 2	:	G20 PROGRAM SCALE
(canned turning cycle in a loop	:	G28 SUBROUTINE START
N010 G90 ABSOLUTE FORMAT	:	G33 THREADING REPEAT CYCLE
N011 G00 RAPID TRAVERSE	:	G54 PROGRAM OFFSET
X 10.000 Z 0.000	:	G65 CALL SUBROUTINE
N012 G73 START DO-LOOP	:	G70 IMPERIAL UNITS
3 TIMES	:	G71 METRIC UNITS
N013 G91 INCREMENTAL FORMAT	:	G73 START DO-LOOP
N014 G84 CANNED TURNING CYCLE	:	G84 CANNED TURNING CYCLE
X-0 5.00 Z-25.000 F 78.00	:	G90 ABSOLUTE FORMAT
N015 G00 RAPID TRAVERSE	:	G91 INCREMENTAL FORMAT
X-0 500 Z	:	
N016 G06 END DO-LOOP	:	

Up Down S Search ESC Quit
 INS insert DEL Delete E Edit

DEL DELETE

The delete function allows blocks of program to be erased from the memory. To delete a specific block of program it must first be positioned at the bottom of the screen by use of the up and down arrow keys. Once in position pressing the DEL key in the bottom right hand corner of the keyboard, will erase the block. The rest of the program will however be renumbered automatically from that point onwards by the computer.

NOTE:

Use this function carefully as once a program block has been deleted it is not possible for it to be retrieved. The whole block would have to be re-inserted.

E EDIT

Pressing E at the program editing stage allows for individual values within blocks of program to be changed. e.g. should the Z value in block 20 of a program require amendment, it must first be positioned at the bottom of the screen by either using the up and down arrow keys or by using the search facility.

Once the desired block is in position at the bottom of the screen, pressing 'E' will prompt for the values to be changed. When the block has been amended press E again to return to program edit mode.

SECTION 40

GRAPHIC SIMULATION

When the program has been typed into the computer it can be simulated on the screen by use of the graphic simulation option.

If it is desired the simulation and the machine can be run at the same time, this shows exactly what is being executed by the machine.

To run the graphic simulation adopt the following procedure.

40.1 SIMULATION SETTINGS

From the STARTURN main menu take the 'SIMULATION SETTINGS' option by moving the white cursor bar by means of the up and down arrow keys located on the numeric keypad section of the keyboard, press RETURN when the cursor is in the correct position.

The screen will now look like this:-

EDITOR	PROGRAM DMTEST:	SETTINGS
N007 G65 CALL SUBROUTINE		:Please select one of the following:
LABEL 2		:
N008 G00 RAPID TRAVERSE		: Set length 70.000
X 0.000 Z-0.500		: Set diameter 20.000
N009 G65 CALL SUBROUTINE		: Set scale 2.000
LABEL 2		: Bore diameter 0.000
(canned turning cycle in a loop		: Imperial/metric Metric
N010 G90 ABSOLUTE FORMAT		: Machine control No
N011 G00 RAPID TRAVERSE		:
X 10.000 Z 0.000		:
N012 G73 START DO-LOOP		:
3 TIMES		:
N013 G91 INCREMENTAL FORMAT		:
N014 G84 CANNED TURNING CYCLE		:
X-0 5.00 Z-25.000 F 78.00		:
N015 G00 RAPID TRAVERSE		:
X-0 500 Z		:
N016 G06 END DO-LOOP		:
N017 G90 ABSOLUTE FORMAT		:
		:
		:Press 'f1' for help 'ESC' to exit

A brief explanation of the simulation setting functions can be gained by pressing the f1 key on the left of the keyboard. However they are explained in detail on the following pages.

SET LENGTH

The set length option allows for the length of the billet to be used, to be input into the computer. the length entered represents the total length of the billet and not just the amount protruding from the chuck. (allow 15mm for chuck grip).

Once the billet length has been entered press ENTER to record the figure.

SET DIAMETER

The set diameter option allows for the diameter of the billet to be used, to be input into the computer.

Once the billet diameter has been entered press ENTER to record the figure.

SET SCALE

The set scale option allows for the simulation scale to be increased, or decreased until it is acceptable on the screen. Once run the simulation scale may be changed to give a more acceptable picture. Should too big a value be input then the simulation will be executed outside the limits of the screen and it will not be possible to see the result.

BORE DIA

The bore diameter option allows the billet to be used to be 'PRE DRILLED' read for a boring operation. The bore will be for the whole length of the component at the required dia.

IMPERIAL/METRIC

The Imperial/Metric function allows the dimension for the length of diameter of the billet, specified to be converted from metric to English and vice versa.

This conversion process does not affect the program in the memory of the computer.

To execute the change place the white cursor on the option by using the UP and DOWN arrow keys, and press ENTER . This will change the value either.

Imperial or Metric to its opposite. It can be noted that as the option for the units changes then the length and diameter will be converted.

40.2 MACHINE CONTROL

The machine control (YES/NO) option will, when set to 'YES' allow for the simulation process, to be executed at the same time as the machine.

If option 'YES' is selected but the tool offsets have not been set the message

DATUM TOOL ZERO FIRST

will be displayed. If this occurs, then the tool offsets must be set using the toolsetting facility (see page 60)

Once the tools are set the simulation will run and request the spindle speed to be set to the one programmed. When a toolchange is detected in the program the operator will be requested to fit the correct tool number corresponding with the program.

Should the machine control option be set to 'NO' then only the screen simulation will be run. In this case no spindle speed or tool change is requested by the program.

When all the simulation settings have been correctly set press 'ESC' to return to the main STARTURN menu.

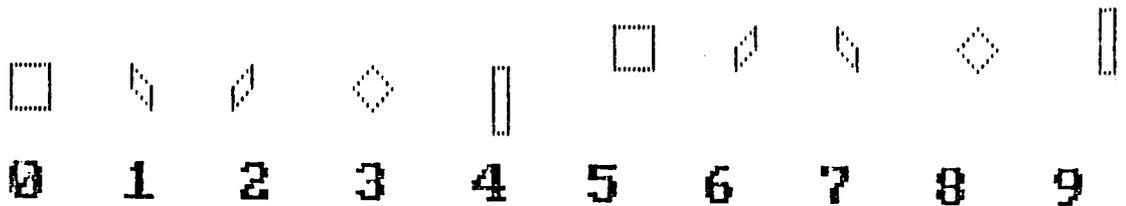
40.3 EXECUTING THE SIMULATION

To run the toolpath simulation on the computer monitor, take the 'SIMULATION' option from the STARTURN main menu.

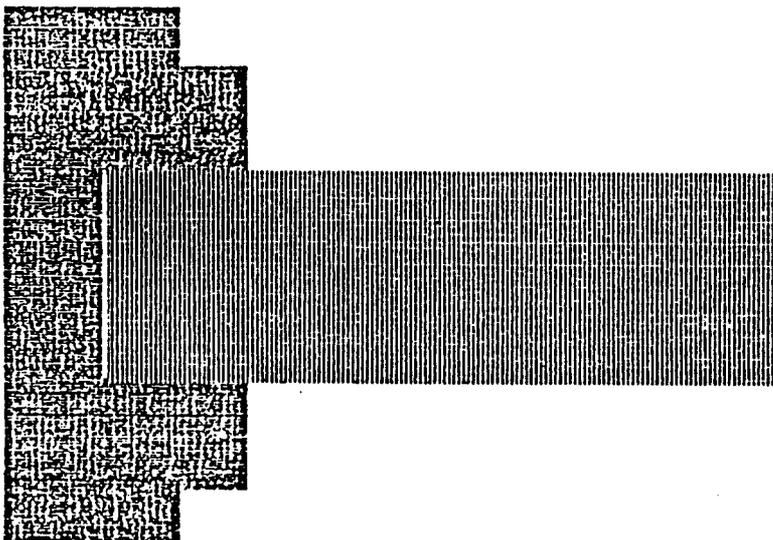
The chuck and billet will be drawn on the screen (chuck in yellow, billet in red). The simulation will be executed block by block, with the block being executed displayed on the screen above the simulation.

When a tool change is detected in the program the simulation will stop, display the available tools and prompt for which tool is, to be used. At this point the screen will display the following:-

TOOL SELECTION



Which tool?



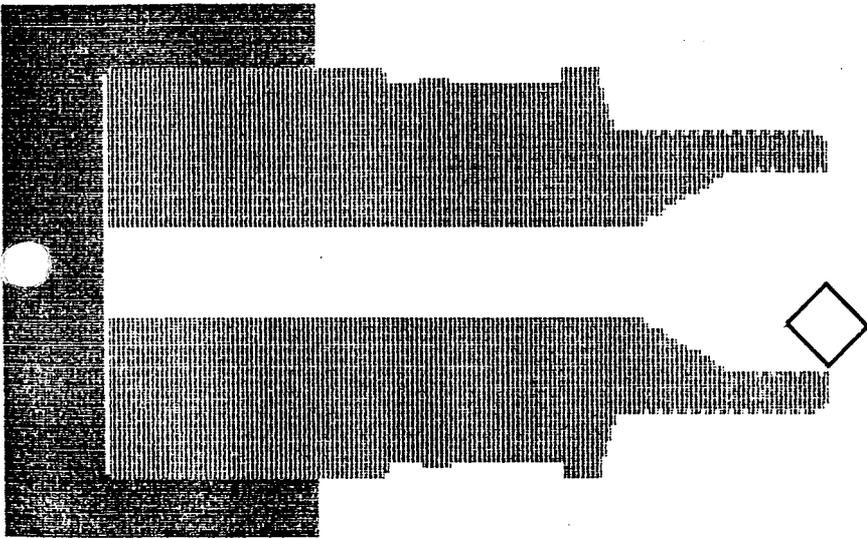
Tool number 0 represents the reference tool (see page 60)

Tools 1 to 4 represent the available external tools whilst Tools 5 to 9 represent available internal tools used for boring work.

when the correct tool has been specified the simulation will run with that tool until the program detects a tool change, whereupon the tools will be displayed again and a new tool may be selected.

During the simulation process the screen will show the following.

```
N065 G00 RAPID TRAVERSE
      X 6.000      Z 0.914
N066 G33 THREADING REPEAT CYCLE
      X 6.473      Z-5.000      P 1.500
      C 16
N067 G00 RAPID TRAVERSE
      X 0.000      Z 0.914
```

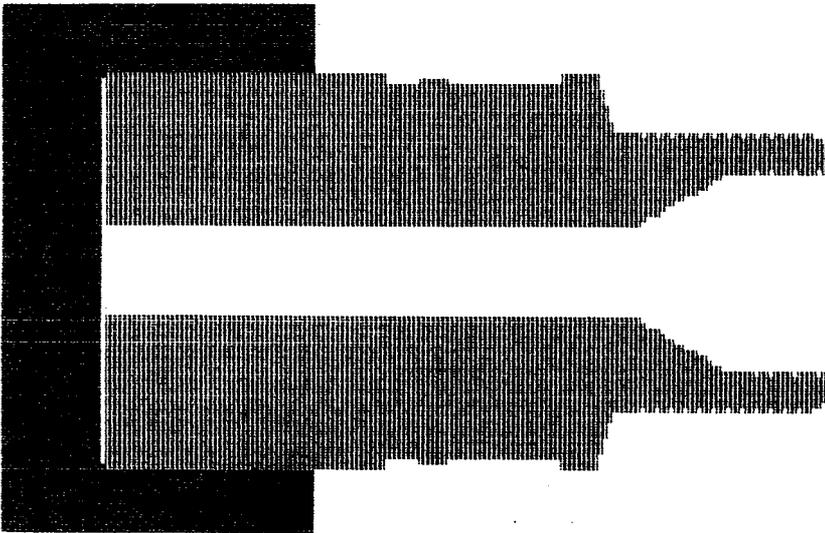


The Program Listing can be 'turned off' by pressing the ENTER  key, this will make the simulation run faster. The listing can be re-displayed on the screen by pressing the enter  key again.

As the simulation is running it can be stopped by pressing the 'ESC' key. This will halt the graphics after completion of the current block being executed.

The finished simulation will be as follows:-

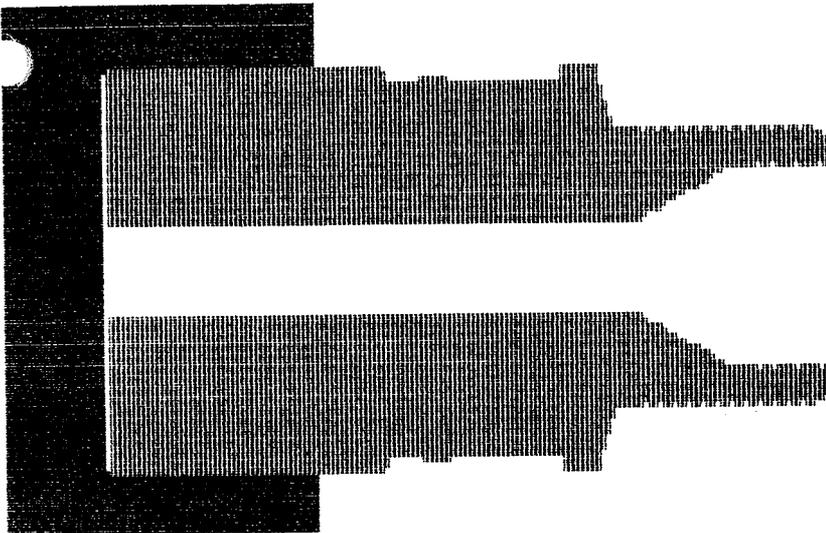
```
N068 X 0.000 Z 0.914  
G00 RAPID TRAVERSE  
N069 X 0.000 Z 0.000  
G00 RAPID TRAVERSE  
N070 X 20.000 Z 20.000  
M05 SPINDLE STOP  
N071 M02 END PROGRAM
```



Press ESC to exit

When a component has been programmed with boring operations and the simulation settings state a bore diameter to be shown on the graphics, then the original graphics picture to be drawn will be as follows:-

```
N065 G00 RAPID TRAVERSE  
X 6.000 Z 0.914  
N066 G33 THREADING REPEAT CYCLE  
X 6.473 Z-5.000 P 1.500  
C 16  
N067 G00 RAPID TRAVERSE  
X 0.000 Z 0.914
```



A graphics screen dump may be taken on a printer with centronics parallel interfacing capabilities simply by presing SHIFT/PRISC keys.

This print screen routine is available for a screen dump or any of the STARTURN'S displays and may be executed by adopting the above procedure.

To exit from the disk directory mode back to the disk utilities menu, press the 'ESC' key.

DELETE PROGRAM

The DELETE program option allows for the removal of a program from the disk. When the option is selected the computer will request the name of the program to be deleted.

NOTE: Use this function carefully as once a program has been deleted from the disk it cannot be retrieved. The whole program must be re-entered.

RENAME PROGRAM

The RENAME program option allows for a program currently stored on a disk to have its name changed on entry of the option, the original program name will be prompted for, when input the next prompt will be for the programs new name. When the process is complete the change can be checked by executing the disk directory option from the disk utilities menu.

CHANGE PATH

The CHANGE PATH allows for programs to be loaded from or stored to a specific area of the Disk previously created using the 'MAKE NEW PATH' option from the disk utilities menu.

MAKE NEW PATH

The MAKE NEW path option allows for an area of disk to be allocated for example to each student in a class. The area on disk could be named after the student and would contain all of his or her programs. This function makes the management of the disk a lot easier as the data is grouped in specific areas.

To make a new path select the option from the menu and type the name of the path when requested by the computer.

e.g. to make a path on the disk called 'FRED' type FRED when requested by the computer.

A path may however be created from a path e.g. a path called 'CLASS1' may be made and the names of the students in Class 1 created, within it thus breaking down the usage of the disk further.

SECTION 42

NEW PROGRAM

The new program option selected from the STATURN main menu allows for the program currently in the memory of the computer to be deleted, thus clearing the memory ready for the next program entry procedure.

The new program option does not save the program to disk, this must be done via the disk utilities menu described on page 51 of the manual, thus is this option is selected the program will be permanently destroyed, and cannot be recovered.

SECTION 43

PRINT PROGRAM LISTING

The PRINT program listing option on the STARTURN main menu allows for the program currently in the memory of the computer to be printed with centronics parallel interface capabilities.

For the PRINT program facility to operate correctly there must be a centronics parallel interface card in position in the computer.

The program printout will be in the format shown below.

DENFORD MACHINE TOOLS LTD STARTURN IMB

PROGRAM DMITEST LISTING

```
(TEST PROGRAM FOR STARTURN..
(This program should be run
(on arrival of your new
(Starturn Lathe .....
(Please ensure that you hae
(everything connected right.
(Your computer should have
(a serial card linked to the
(Starturn lathe connected by
(the cable and security device
(supplied with the machine.
N001 G90 ABSOLUTE FORMAT
N002 G71 METRIC UNITS
N003 G20 PROGRAM SCALE
SCALE 1.000
N004 G00 RAPID TRAVERSE
X 11.000 Z 0.000
(call facing cycle twice..
(use tool 1 in simulation.
N005 M03 SPINDLE FORWARD
650 RPM
N006 M06 TOOL CHANGE
TOOL 1
N007 G65 CALL SUBROUTINE
LABEL 2
N008 G00 RAPID TRAVERSE
X 0.000 Z-0.500
N009 G65 CALL SUBROUTINE
LABEL 2
(canned turning cycle in a loop.
N010 G90 ABSOLUTE FORMAT
N011 G00 RAPID TRAVERSE
X 10.000 Z 0.000
N012 G73 START DO-LOOP
3 TIMES
N013 G91 INCREMENTAL FORMAT
```

SECTION 44

MANUAL CONTROL OF PROGRAM EXECUTION

The manual control option which can be selected from the STARTURN main menu, by moving the white cursor bar by means of the ↑ up and ↓ down arrow keys on the numeric keypad section of the computer keyboard and pressing ENTER .

In this option the computer controls the machine, and the operator is able to set the tool length offsets, and execute the program.

When selected the following menu will be displayed.

MANUAL CONTROL	PROGRAM DMTEST:	PANEL CONTROL
STARTURN CONTROL PANEL X		:use these function keys:
		: f1 Change jog to continuous
		: f2 Change units of measure
X Z SPINDLE SPEED		: f3 Feed <
		: f4 Feed >
7.92 -12.59 0 RPM		: f5 Finish manual control
		: f6 Execute program
		: f7 Tool offsets
		: f8 Set tool number
FEED = 380 M.M. /MIN TOOL O		: f9 Datum X
		: f10 Datum Z
		: Space bar
METRIC UNITS CONTINUOUS FEED		: Use these cursor keys for X and Z
		:feeding:
		: UP -X
		: DOWN +X
		: LEFT -Z
		: RIGHT +Z

The window on the left hand side of the screen displays the X and Z position of the slides. Until the tool offsets have been correctly set the figures are meaningless. The spindle speed is also displayed. This will display the correct spindle speed as the speed is adjusted by use of the spindle thyristor knob.

The feedrate value is also displayed, the initial value is set to 380mm/min. The method of changing the feedrate is explained in the following section.

The current tool is shown, this automatically registers as tool 0.

Finally the units of measurement are displayed. The default value is 'METRIC', and the mode of JOG is shown, this defaults to CONTINUOUS - explained in the next section.

44.1 THE 'f' FUNCTION KEYS

On the right hand side of the screen, is a window displaying functions used by the 'f' keys on the left hand side of the computer keyboard. Their usage within the STARTURN software is described below.

f1 CHANGE JOG TO CONTINUOUS

The 'f1' function key changes the JOG mode of the machine. When first turned on the job mode is displayed on the screen as 'CONTINUOUS FEED'. This means that when the arrow keys on the numeric keypad section are pressed the tool will move in that direction until the key is released.

By changing the feed to JOG mode by pressing the 'f1' key the tool will jog at a feed of 10mm/min.

This is to allow the tool to be moved slowly to the component to avoid a tool collision, when setting the tool offsets the mode can be changed back to continuous by pressing 'f1' again.

f2 CHANGE UNITS OF MEASURE

When the manual control function is taken the machine units default to metric. The units can however be changed by pressing 'f2'. When pressed the screen will display imperial units, and the X and Z figures on the screen will be converted.

The units must be set to inch or metric according to the units programmed.

f3 FEED <

The 'f3' key changes the manual feedrate by decreasing it. When pressed the feedrate value displayed on the screen will be reduced. This is to allow for a slower rate of feed to be used when turning the diameter of facing the billet during the toolsetting procedure.

When first selected the feedrate defaults to 380mm/min.

The minimum feed in manual mode is 10mm/min.

f4 FEED >

The 'f4' function key acts in the opposite manner to 'f3' by increasing the manual feedrate. The maximum the feed can be increased to is 1200 mm/min.

f5 FINISH MANUAL CONTROL

The finished manual control function quits the manual control function and returns the operator to the main menu.

f6 EXECUTE PROGRAM

Upon pressing the 'f6' function key the program will be executed on the machine. The computer will request that the correct tool be placed in the toolpost at the appropriate time, when a M06 tool change command is detected in the program.

The same procedure will occur for a M03 spindle forward command, in this case the computer, will prompt for the spindle speed on the machine to be set to the same value as programmed.

In both the above cases i.e. when the tool has been changed or, spindle speed set the computer requests that the return key be pressed, to enable the rest of the program to be executed.

NOTE: Before the program is executed the tool offsets must be set to enable the program to run correctly. (see Tool Offsets section). Should any problem occur during the program, a tool breakage for example, and the execution of the program needs to be stopped immediately. Then press the red emergency, stop button on the right hand front of the machine.

Should the program execution require halting under a non emergency situation then press the 'ESC' key. This will revert the machine to manual mode and stop the program.

If pressed the emergency stop button can be unlocked using the key.

NOTE: Pressing emergency stop will effectively counteract the tool offsets to reset the tools only. Tool zero needs to be reset. (see Tool Offsets page 60).

f7 TOOL OFFSETS

The tool offsets function allows for the amendment of the tools that have been set. A component may have the length and diameter slightly altered to allow for tool wear compensation, or to make length and diameter changes without affecting the actual CNC Part Program. The method of tool adjustment is described in the 'SECOND OFF' section of the manual.

f8 SET TOOL NUMBER

The 'f8' set tool number function allows for the different tools to be used in the program to be selected for setting. This procedure is described in detail under the 'OFFSETS' section of the manual.

f9 DATUM X

The DATUM X function allows the diameter of the tool to be input as each of the tools to be used are touched onto the workpiece, when the tool tip is touched onto the diameter and 'f9' pressed the screen will register the radius. The full procedure is explained under the 'OFFSETS' section of the manual.

f10 DATUM Z

The 'f10' DATUM Z function allows the Z zero (the face of the billet) to be logged for each of the tools used in the program. When the tool is touched onto the end of the billet and the 'f10' key pressed the screen will display Z0 for that tool.

The full procedure is explained under the 'OFFSETS' section of the manual.

The final message displayed shows the direction the tool will move when the appropriate arrow keys on the keyboard are pressed.

TOOL OFFSETS

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

The first tool to be set is always Tool 0. This acts as a reference tool as all other tool offsets will be taken from this tool. Theoretically the zero reference tool should not be used for turning as it may get broken and the other offsets for other tools set will become redundant.

In a one tool program the same tool however can be used as Tool 0 and Tool 1 and both carry the same offset. Therefore if Tool 0 is to be used as Tool 1 it is only necessary to set the offsets for tool zero.

45.1 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 tool holders.

Select the first tool to be set - "TOOL 0" - by pressing the 'f8' set tool number option from the menu, a prompt will inform that Tool 0 must be inserted and 'RETURN' pressed.

The Z zero plane (end of billet) now needs to be set by moving the tool up to the end of the bar using the arrow keys. In the 'CONTINUOUS MODE' this 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 work the 'JOG MODE' can be changed to 'SINGLE STEP' by pressing the 'f1' key.

It is recommended that a pass is taken across the face of the billet. This process ensures a square reference face. Press the 'f10 DATUM Z' key to set the plane as zero. At this point the Z dimension on the screen will display 0.00.

The next process is to take a skim along the outside diameter of the billet, again by using the computers' arrow keys. At this stage press 'f9' set X DATUM' key, whereupon the computer will request the diameter of the newly turned section of the billet. An accurate measurement of the turned section must be taken, either with a micrometer or verniers. Enter the measured diameter into the computer. The X dimension on the screen will display half the value entered, as STARTURN operates on a radial system of measurement.

To set the new tool, press the 'f8 SET TOOL NUMBER' function key on the left of the computer keyboard. Again the computer will request that the tool be inserted in the toolpost and 'RETURN' pressed to acknowledge the tool has been inserted. The above tool setting procedure can be repeated for all the tools to be set in the program.

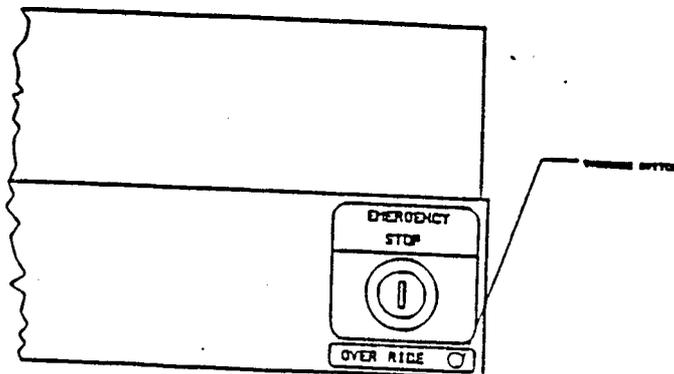
45.2 EXECUTING THE PROGRAM

After all the tools have been set the program can be executed by pressing the 'f6 EXECUTE PROGRAM' function key.

The computer will prompt at the correct times in the program for the correct tool to be placed in the tool post and for the spindle speed to be adjusted to the one programmed. The program can be run one block at a time by pressing the space bar. Press again to run the next block. This method of execution allows for closer inspection of the program as the execution is slower and non continuous.

45.3 AXIS LIMIT SWITCH OVERRIDE

NOTE: Should the slides hit the axis overtravel limit switch in the Z axis (the stop is adjustable along the bed) then simultaneously press the small reset button located under the emergency stop button and move the slide away under manual control.



AXIS LIMIT SWITCH OVERRIDE

46.2 RELATION OF TOOL OFFSETS FOR PREVIOUSLY DESCRIBED EXAMPLE

TOOL OFFSET REPRESENTATION

