STARTURN SIMULATOR

Setting Up

When running the Starturn Simulator for the first time select the following:-

a) Press 'D' to select a program DRIVE. If you don't select a Drive number a message will remind you to do so.

b) Press 'N' to give the CNC program a name. If you don't select a Cnc name a message will remind you to do so.

c) Press 'W' to select the workpeice size.

SET WORKPEICE SIZE

Press 'W' from the Main Menu to enter this section as shown below.

STARTURN - MACHINE SIMULATION

Select one of these operations
L ... Length of Workpeice
D ... Diameter of Workpeice
S ... Save Settings
R ... Read Settings
Q ... Quit
Key in your choice:
Length ? Diameter ?

LENGTH
Press 'L' key to enter a suitable Length.

DIAMETER
Press 'D' key to enter a suitable Diameter.

SAVE SETTINGS

To save the Datum settings select 'S'. The data is saved automatically under the same filename as the workpeice but using a different letter prefix.

READ SETTINGS
Select this option to load Datum information that has already been saved at an earlier date.

QUIT
Press the Esc key to quit. Any of the information can be changed by returning to this section. Remember if you change anything to Save it to Disk.

Entering Cnc programs.

Once the Disk Drive number, component name, Datum position and component dimensions have been entered a CNC program can be written.

Press the 'C' key to Create a CNC program.

Note

If you are using a Data Disk in Drive 0 you will be prompted when to place it in the Disk Drive.

HOW TO ENTER PROGRAMS

The Starturn like most CNC turning machines is programmed using G and M codes . Because of the learning problems associated with CNC programming languages we have incorporated single line entry with on screen simulation. This is particularly useful for beginner's to CNC programming.

TYPING A CNC LINE

All lines are typed at the prompt "=>", with a flashing cursor indicating the computer is waiting for a key to be pressed.

The following keys can be used to edit the current line:-

Delete	to erase previous characters.
Arrow keys	to move cursor to new edit position.
Esc or Ctrl U	to delete the current line.
Return	to confirm.

Correct Format

All lines can be typed in lower or upper case. Spaces can be used to pad out the lines. G codes can be typed as "G01" or as "G1". Where certain information has been missed off a line you will be told. Information on a line can be in any order.

For example :-

```
G1 X22.3 Z12.3 F122  or
g1 z12.3 f122 x22.3
```

All lines are checked for validity and a related error message will appear on the screen. If the line is valid you will be asked to confirm whether you want to accept the line or not.

Command Functions.

To assist in the learning of CNC programming there are several command functions to allow the writing of interactive lessons.

1- Comments Eg (This is a comment

Comments can be typed only after a left bracket has been keyed in as the first character on a new line. This is useful for describing what is happening in the program. The programmer's name, component information and tool sizes can also be typed into the program as comments.

2- Scrolling Messages Eg !This is a message

Scrolling Messages can be typed only after an exclamation mark has been keyed in as the first character on a new line. The message will be displayed at the bottom of the screen. Useful as an instruction to the student. Eg To tell him to write down what type of cycle is to be executed next.

3- Static messages ?This is a static message

These are similar to scrolling messages but wait for a key to be pressed before execution of the program continues. The first character for Static messages is a question mark.

COMMANDS

There are several commands that affect the simulation and aid in the development of CNC lessons.

All commands are entered after a left hand square bracket Eg [TOOLPATHON. **Note** - the '[' will appear as a '<-' character when in the editor mode.

DRILL DIAMETER
Prior to a Toolchange this command is used to tell the Simulation what Drill diameter is required by the next Tool. Only becomes active after a Toolchange.
Format:- **[DRILLDIA 8** will select an 8mm Tool.

TOOLWIDTH
Prior to a Toolchange this command is used to tell the Simulation what Tool width the grooving Tool has.
Format: **[TOOLWIDTH 1.5**

SINGLE STEP SIMULATION.

This command allows single step or continuous execution of the program. May be selected anywhere within the program and implemented as many times as required.
Format :- **[STEP** - halts between blocks
[NOSTEP - continuous execution

CLEARING MESSAGES.

The messages that appear at the bottom of the screen scroll up when mew messages are printed but do stay on the screen. This command clears all messages from the message window
Format :- **[CLEAR**

HELP

Typing "HELP" at the command line will list the G and M codes with a description at the bottom of the screen. Use the arrow keys to scroll up and down and the Esc key to quit.

PRINT OPTIONS

There are two Help files that can be printed out from the Edit Command Line.
Type PRINT1 for a listing and description of all the G and M codes.
Type PRINT2 to send file HELP2 to the printer. This file is user definable. Use any word processor to create your own Help notes. Save the file as "HELP2" onto the Disk that is currently used for saving CNC programs.

ESCAPE

During Single Line Simulation mode you must program a M02 to quit. You can then Return to the Main Menu by pressing the Esc key.

SIMULATION

To simulate a CNC program in full select the Simulation option from the Main Menu.

All axis moves are simulated on the screen with the metal being removed at the bottom and a Tool path drawn at the top of the component.
Fast Traverse is shown as a dotted line with all feed movement in solid. Any moves outside the screen are clipped and not visible. If the STEP command is used you will have to press a key after each line has been simulated.
After the full CNC program has been simulated you may press the Esc key to Return to the Main Menu.

Starturn Editor

Select option 'E' from the Main Menu to enter the full screen Text Editor.
Once a student is coversant with CNC programming through use of the single line simulation he can move on to a full screen cnc text editor. The whole program can be typed using powerful edit routines like Copy, Move, Delete, Insert, Merge.

The Main functions of the Editor are listed below:-

CNC EDITOR

F0 - Help
Use the arrow keys to page up or down through the Help Screens. Press the Esc key to quit.

F1 - G and M code listing
Use the arrow keys to page up or down or press the ESC key to Quit.

F2 - Info
Program information on free memory available ,Denford's address and telephone number and information about marked lines.

F3 - Menu

L ..... Load Cnc Program
S ..... Save Cnc Program
N ..... Begin New Program
R ..... Renumber/Strip N Prefixes
P ..... Print Cnc Program
H ..... Print Help Notes

Select an option

Load Cnc Program

a) You will be prompted to accept the default filename or type in a new filename. If you type in a new filename the default filename will still be active when you return to the Main Menu.
b) If the editor has a program resident in memory you will be prompted as to whether you wish to merge the program on Disk with the active program or not.
c) If you don't want to merge programs and the active program has been changed since the last save to Disk you will be warned and given the chance to Save the program.
d) If the program being loaded is too large for the Editor you will be told so and the file will be truncated.

CNC EDITOR

Saving Cnc Program

a) When saving you will be prompted to accept the current filename or not. Either press Return to accept or type in the new name.

b) If the filename already exists on Disk you will be warned and prompted to overwrite or not.

c) If part of the program is currently marked you will be offered the choice of saving just the marked area or all of the program.

d) If during the save the Disk becomes full you will be told. Either Compact the disc on another computer and then try again or format a Data Disk and select save again.

Begin A New Program

Select this option to wipe the current program from memory. If the current program has been changed since the last save you will be told so and given the chance to save it.

Renumber/Strip N Numbers

The N block numbers are line numbers and quite often you will want to renumber them. If you want to miss them out altogether during editing you can. They can be added at the end of programming prior to a save. If there is a marked area you will be asked whether to change the marked area or to change the whole program.

a) You will be asked if you want to add N block numbers or strip them off.

b) If you selected add N block numbers you must select the start value and an increment value.

After one of the above operations control returns to the main edit screen.

Print Cnc Program

You will be prompted for Line feeds (Y/N)?
If there is a marked area you will be asked whether to print the marked area or to print the whole program.

CNC EDITOR

Print Help Notes

You will be given a choice of two Help files:

Help File Number 1

This file contains information on the G and M codes and how the Editor works. The name of the file is EDHELP1.

Help File Number 2

This file can be filled out by the software user with any wordprocessor that saves files in ascii format. The name of the File is EDHELP2.

Remember

This file is empty until you create your own Help Screen and save it to Disk under the name EDHELP2.

CNC EDIT COMMANDS

Cursor Position

The flashing cursor always points to the current edit line.

Cursor Keys

Left Arrow	Moves one space to the left.
Right Arrow	Moves one space to the right.
Ctrl/Left Arrow	Moves to start of the line.
Ctrl/Right Arrow	Moves to the end of the line.
Up Arrow	Moves up one line.
Down Arrow	Moves down one line.
Shift/Up/Arrow	Moves up one page.
Shift/Down/Arrow	Moves down one page.

CNC EDITOR

CNC EDIT COMMANDS

Ctrl/Up/Arrow Moves to the first line.

Ctrl/Down/Arrow Moves to the last line.

Delete Keys

Delete Deletes character before cursor position.

Copy Deletes character at cursor position.

Control Keys

Ctrl & N Makes room for new blank line.

Ctrl & U Deletes line at cursor position.

Ctrl & R Restores current line to pre edit status.

Function Keys

F0 Displays help screens

F1 Displays G and M code's.

F2 Displays program information.

F3 Selects utilities menu.

Ctrl & F4 Block Copy

Ctrl & F5 Block Move

Ctrl & F6 Block Delete

F7 Begin marking block.

F8 End marking block.

F9 Toggle between insert and overwrite.

SECTION TWO

CONNECTING THE CABLE

On the BBC, the cable connects to the port labelled 'RS423' with the screw on the connector facing away from the power ON/OFF switch. On the STARTURN the cable connects to the port labelled 'RS232' with the screw on the connector facing the top.

Please note

You do not need the cable linked to run the Simulation and Editor, only for sending or receiving information between the computer and the Starturn in MANUAL CONTROL and EXECUTE modes.

IMPORTANT

Make a backup copy of your STARTURN software before running the program. You will also need to prepare a blank formatted disc to store your CNC programs. Your Disk Drive manual will tell you how to copy and format Disks.

CONTROL PANEL FUNCTIONS

EMERGENCY STOP
The red emergency stop button is positioned to the far right of the Control Panel. Pressing this button will stop any axis and spindle movement dead. Use the key to unlock the machine for furthur use.

CONTROL PANEL FUNCTIONS

STARTURN CONTROLLER

The control software for the Starturn machine is supplied on a seperate Disk to the Starturn Simulator. This means somemone can program and simulate away from the machine while another student can be using the Starturn machine.

Boot the software up in the usual way by pressing the Shift and Break key.

Note:-

The Starturn control and execution software will run on any model of BBC.

STARTURN CONTROL

The Main Menu is displayed as below:-

STARTURN - MACHINE CONTROL

Select one of these operations:

M ... Manual control of Starturn
E ... Execute CNC program
C ... Configure Machine
D ... Select CNC program drive
N ... Give CNC program a name
Q ... Quit

Key in your choice:

CNC program drive not known
CNC program is not named

Before selecting the Main options set the Drive number and program name.

Manual Control of Starturn

X Y Z	Axis moves
F	Alter Feedrate
S	Alter spindle speed
T	Tool change and Offsets
M	Manual Control
D	Datum machine
U	Change units
E	Execute CNC program

CONFIGURATION OF STARTURN MACHINE

Before the software will drive the Starturn machine the disc must be configured for the different options available. The settings are written to disc so there is only a need to do this once. The options are :

A) Automatic spindle speed control (programmable from the computer) **or** Manual spindle speed control (set from a dial on the machine).

B) Automatic tool post **or** Manual tool change system.

Select option **C** on the main menu to configure the disc.

STARTURN CONTROL

The following facilities will be available in this mode of operation.

Move X/Z Axis

Allows a single axis absolute move in the X/Z axis to be executed, at the current feedrate.
All moves must be within the parameters displayed within brackets.

Feed rate

Changes the current feedrate at which all movements in manual mode take place.

Spindle

Spindle speed can be changed within the stated parameters.

Tool change and Offsets

Displays a the menu below:

C	Change tool
X	Sets X offset
Z	Sets Z offset
E	Edit tool offsets
L	Load offsets from disk
S	Save offsets to disk

Change tool

Selects a Tool and calls up it's offset if the Offsets are set.

Set X offset

Sets the 'X' datum position on the screen after promting for the diameter of the bar to be entered.

Set Z offset

Zeroes the 'Z' datum position on the screen. All future moves will be from this set Datum position unless changed at a later date.

STARTURN CONTROL

All programming is from a set Datum position and the Starturn needs to know where this Datum is. By moving the Tool to the end and the Diameter of the component you are in the correct place to Set the Datum at this position.

Tool offsets

Tool Offsets need to be set for all the Tools you are currently going to use.
Be sure to set the first Tool as Tool 0 by touching on the end of the component with the Tool and datuming the 'Z' axis. Then move to the Diameter of the workpeice, after taking a small cut measure the Diameter and Datum the 'X' axis by typing in the measured Diameter.

Edit

The 10 tool length offsets allowed can be entered or altered simply by selecting the tool number, from 0 to 9 and entering the new offset.

Load Tool Offsets from disk

Type in the Tool Offset filename and press Return to load information.

Save Tool Offsets to Disk

Type in the Tool Offset filename and press Return to save the current offsets to Disk.

Once the Tool Offsets have been fixed you can execute the program.

PROGRAM EXECUTION

During program execution the machine uses several commands as documented in the Simulation section.
[STEP [NOSTEP [CLEAR
Comments and questions are also used allowing lessons to be produced.

Escape Key.

If during execution you want to restart the program from the beginning or to quit altogether, pressing the Escape key will give you these two options.

Emergency Button

The red emergency button can be used to stop execution at anytime. Using this button will cut off power to the drives and switch the spindle off. You can restart the program from the beginning by releasing the emergency button.

Important

The Escape key is not to be used in an emergency, it does not stop the spindle or the drives and should only be used under controlled conditions.

After the program has completed execution, with a M02 code the spindle switches itself off.

Programs may be repeated any number of times. Remember to load your Tool Offsets before executing an old program.

CAM DESIGNER

Introduction

The Cam Designer module allows the student to build a component from shapes using a cursor and ruler to set the size. Any shape can be edited and new shapes can be inserted to the left or right of any existing shapes. After the Design stage the Cnc Part Program is created automatically by the computer with the correct Tool information, Speeds, Feeds and depth of cut for the Starturn 4 or Starturn 8.

Select the 'B... Build' option at the main menu display to select this module.

Command Keys

Cursor Left and Right

Selects the current object shape for editing (yellow) from the component displayed. The object shape can then be edited or deleted.

Space Bar

Selects the next shape to the right, from the shapes displayed at he bottom of the screen, to be the active shape.

F0

Inserts the active shape to the left of the current object shape.

F1

Inserts the active shape to the right of the current object shape.

F9

Selects the Main Menu

Del

Deletes the current object shape.

Return

Selects the current object shape for editing.

Building A Shape

After inserting a new shape you can set the size by moving the cursor along the rulers to the correct position using the arrow keys. Press the Return key to confirm and the object shape will be redrawn to it's new size.

The size of any object shape can be changed by making it active and pressing Return to show the cursor. Press the Return key again to confirm.

Important Rules

Each shape has to adhere to certain rules and the software checks to make sure the designed component can actually be machined.

Sizes

This rule applies to all the shapes except grooves:-

The Diameter of a shape cannot be smaller than the previous shape. Therefore all shapes moving from right to left must be the same Diameter or bigger.

The cursors can only be moved to a position that is allowed. The cursor stops moving when:-

Trying to define an arc that is not 90 degrees.

Trying to define a diameter or length that is out of range.

Trying to add a smaller Diameter object to the left.

Roughing and Finishing

Tool No 1 will be placed in the CNC program for all roughing and finishing work, so be sure to fit the Tool in the correct position if you have a Programmable Toolpost. The depth of cut for roughing is worked out automatically with a change in spindle speeds at certain Diameters. All feed rates are calculated by the software. All shapes except the groove and thread are for roughing and finishing.

Grooving

The rightmost shape defines a groove and is the width of our standard grooving and parting off Tool (1.5mm).
You can insert any number of grooves.

Tool number 5 will be placed in the CNC program for all grooving work,so be sure to fit the Grooving Tool into the correct position if you have a Programmable Toolpost.
All grooves will be machined after the roughing and finishing profiles but before the threading.

Threading

Shape no 2 is the Threading shape, defined by two colors. You can have any number of threads and the pitch, root diameter, speed and feed are all automatically added.
The start of Threading will be 3mm to the right of the defined thread and if there is a groove at the end of the thread .5mm will be added allowing the threading tool to run into the groove.
Tool number 3 will be placed in the CNC program for all grooving work,so be sure to fit the Grooving Tool into the correct position if you have a Programmable Toolpost.

Facing and Parting

Press the F9 key for the Menu Options and set the Facing and Parting options to 'yes' to activate or 'no' to deactivate.

Facing, if selected will be the first operation and Tool 1 will be used.

Parting Off, if selected will be the last operation and Tool no 5 will be used.

Disk Operations

Press the F9 key for the Load and Save options. The current File name will be used for both Loading and Saving. If you are on a single Disk Drive and not using the System Disk for Data place your Data Disk in the Drive.

Important

If you use the system Disk for saving programs you will soon run out of space. You may have to use the *Compact command to clear the Disk. It is far better to either use a Dual Disk Drive or to use a Data Disk for storing your Cnc Programs.

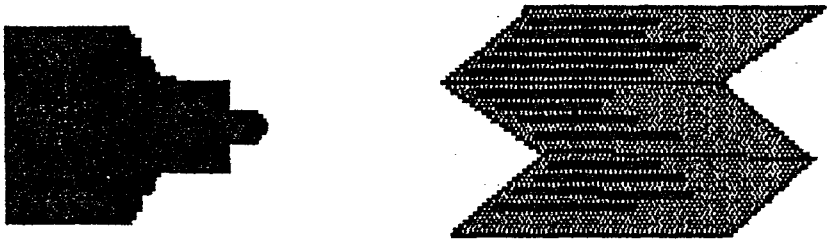
Building the CNC program.

Press F9 to select the Build option. A Cnc program for the current program will be written to Disk and the cuts will be shown on the screen. After completion the Simulator will be loaded and the program can be tested.

Note:

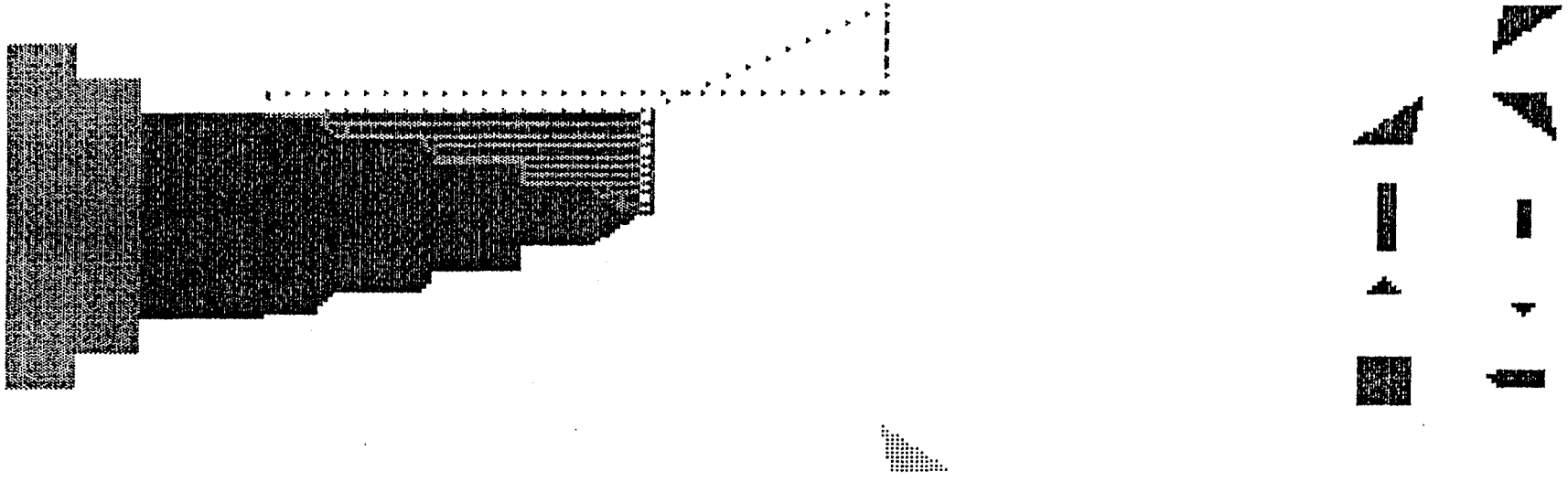
Any component shapes can be re-loaded into the Design Module for editing.

EXAMPLE PROGRAMS



DENFORD STARTURN READING TURN1

```
N155  G01  Z-16.50
N160  G02  X7.50 Z-18.75 R2.25
N165  G01  Z-23.75
N170  G03  X9.50 Z-25.75 R2.00
N175  G01  Z-30.25
N180  G00  X11.50
N185  G00  Z20
N190  M05
N195  M02
```

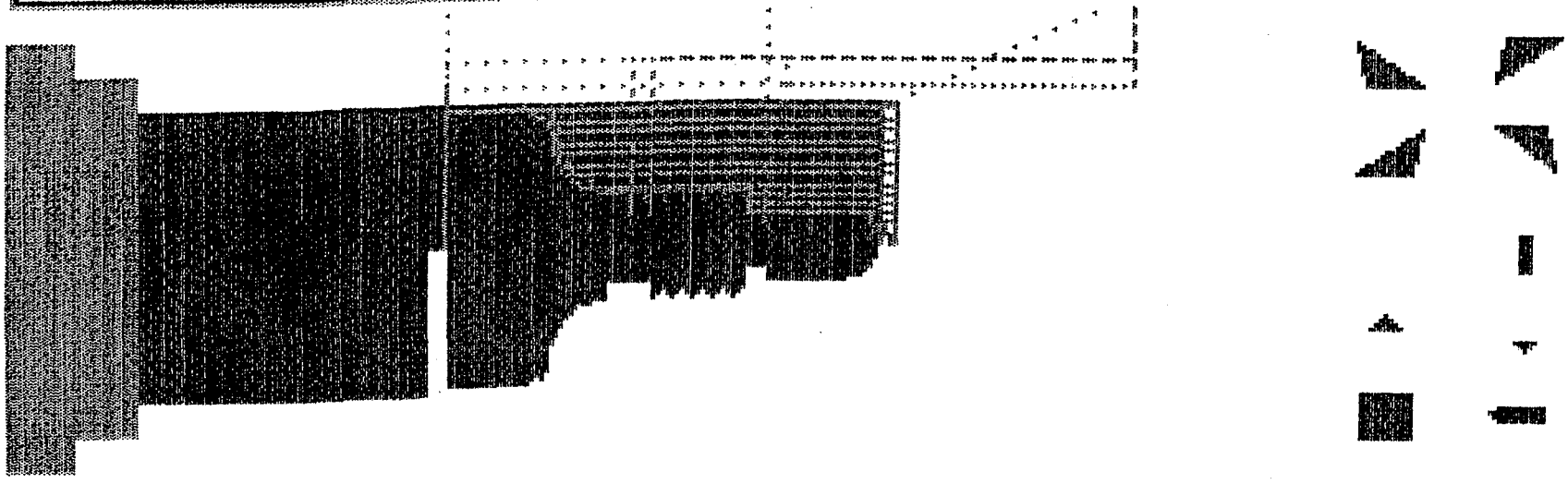


Roughing cuts.
Finishing cuts.

```
@toolwidth 1.5
!Roughing cuts.
N005 G00 X14.00
N010 G00 Z20
N015 M06 T1
N020 M03 S1800
N025 G00 X10.00 Z1
N030 G01 Z1 F120
N035 G84 X8.60 Z-25.39
N040 G01 X8.60
N045 G84 X7.60 Z-23.81
N050 G01 X7.60
N055 G84 X6.60 Z-16.78
N060 G01 X6.60
N065 G84 X5.60 Z-16.41
N070 G01 X5.60
N075 G84 X5.35 Z-16.40
N080 G01 X5.35
N085 G82 X3.10 Z-9.40 C3
N090 G01 X3.10
N095 G84 X2.10 Z-2.57
N100 G01 X2.10
N105 G84 X1.10 Z-1.24
N110 G01 X1.10
N115 G84 X0.10 Z0.09
N120 G01 X0.10
!Finishing cuts.
N125 M03 S2000
N130 G00 X0 Z1
N135 G01 Z0 F120
N140 G01 X3.00 Z-4.00
N145 G01 Z-9.50
N150 G01 X5.25
N155 G01 Z-16.50
N160 G02 X7.50 Z-18.75 R2.25
N165 G01 Z-23.75
N170 G03 X9.50 Z-25.75 R2.00
N175 G01 Z-30.25
N180 G00 X11.50
N185 G00 Z20
N190 M05
N195 M02
```

DENFORD STARTURN READING TURN2

```
N385  M06  T5
N390  M03  S1600
N395  G00  Z-35.25
N400  G00  X14.75
N405  G01  X0  F90
N410  G00  X14.75
N415  G00  Z20
N420  M05
N425  M02
```



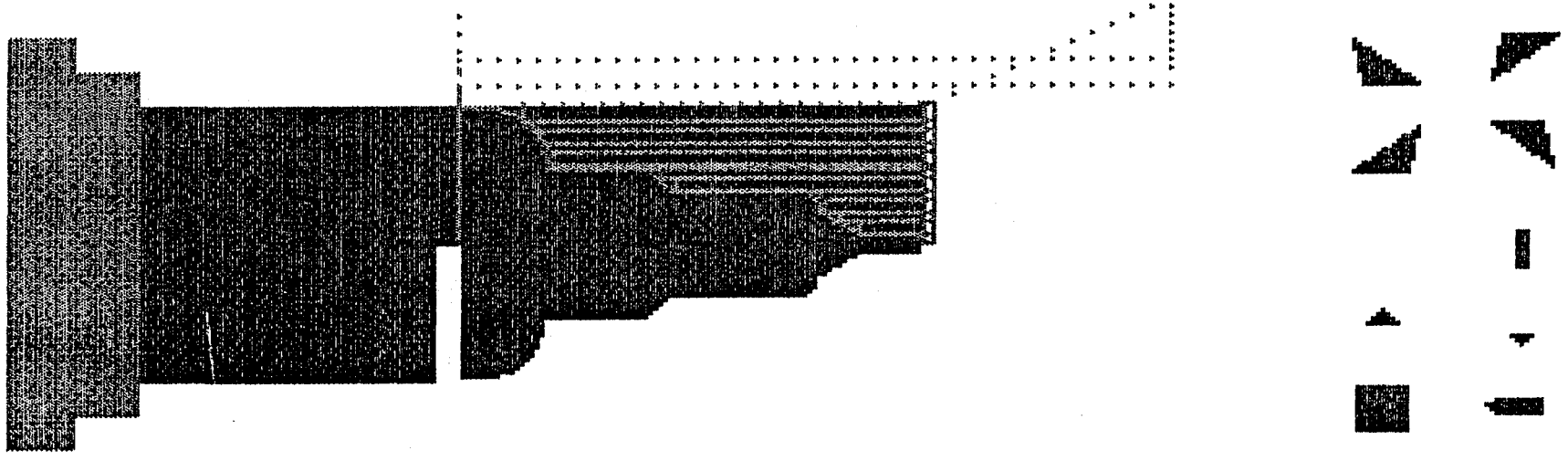
Grooving cuts.
Threading cuts.
Parting off.

```
@toolwidth 1.5
!Roughing cuts.
N005 G00 X17.00
N010 G00 Z20
N015 M06 T1
N020 M03 S1600
N025 G00 X13.25 Z1
N030 G01 Z1 F120
N035 G83 X0.00 Z0
N040 G84 X12.50 Z-29.15
N045 G01 X12.50
N050 G84 X11.50 Z-26.75
N055 G01 X11.50
N060 G84 X10.50 Z-26.30
N065 G01 X10.50
N070 M03 S1800
N075 G84 X9.50 Z-26.22
N080 G01 X9.50
N085 G84 X8.50 Z-26.02
N090 G01 X8.50
N095 G84 X7.50 Z-25.58
N100 G01 X7.50
N105 G84 X6.50 Z-24.82
N110 G01 X6.50
N115 G84 X5.50 Z-23.43
N120 G01 X5.50
N125 G82 X3.25 Z-10.65 C3
N130 G01 X3.25
N135 G84 X2.25 Z-0.36
N140 G01 X2.25
N145 G84 X1.25 Z0.31
N150 G01 X1.25
N155 G84 X0.25 Z0.50
N160 G01 X0.25
N165 G84 X0.00 Z0.50
N170 G01 X0.00
!Finishing cuts.
N175 M03 S2000
N180 G00 X0 Z1
N185 G01 Z0 F120
N190 G01 X0.25
N195 G02 X3.00 Z-2.75 R2.75
N200 G01 X3.25
N205 G01 Z-9.25
N210 G01 Z-10.75
N215 G01 X5.00
N220 G01 Z-18.75
N225 G01 Z-20.25
N230 G01 Z-21.75
N235 G03 X10.00 Z-26.75 R5.00
N240 G02 X12.50 Z-29.25 R2.50
N245 G01 X12.75
N250 G01 Z-35.25
!Grooving cuts.
N255 G00 X17.00
N260 G00 Z20
N265 M06 T5
N270 M03 S1600
N275 G00 Z-9.25
N280 G00 X5.36
N285 G01 X2.25 F90
N290 G00 X17.00
```

```
N295 G00 Z-18.75
N300 G00 X7.00
N305 G01 X3.25 F90
N310 G00 X17.00
N315 G00 Z-20.25
N320 G00 X7.00
N325 G01 X3.25 F90
N330 G00 X17.00
!Threading cuts.
N335 G00 X17.00
N340 G00 Z20
N345 M06 T3
N350 G00 X14.75
N355 G00 Z-7.75
N360 G00 X5.00
N365 M03 S235.00
N370 G33 X4.08 Z-20.00 C31 P1.50
!Parting off.
N375 G00 X17.00
N380 G00 Z20
N385 M06 T5
N390 M03 S1600
N395 G00 Z-35.25
N400 G00 X14.75
N405 G01 X0 F90
N410 G00 X14.75
N415 G00 Z20
N420 M05
N425 M02
```

DENFORD STARTURN READING TURN3

```
N240  M06  T5
N245  M03  S1600
N250  G00  Z-37.75
N255  G00  X14.50
N260  G01  X0  F90
N265  G00  X14.50
N270  G00  Z20
N275  M05
N280  M02
```



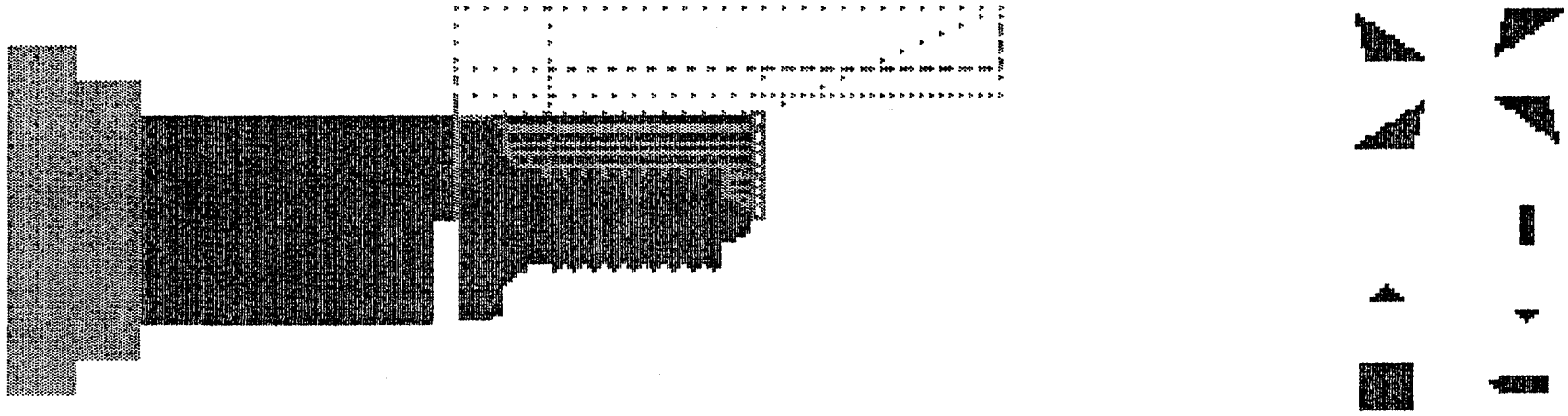
Roughing cuts.
Finishing cuts.
Parting off.

```
@toolwidth 1.5
!Roughing cuts.
N005 G00 X17.00
N010 G00 Z20
N015 M06 T1
N020 M03 S1800
N025 G00 X13.00 Z1
N030 G01 Z1 F120
N035 G83 X0 Z0
N040 G84 X11.60 Z-32.50
N045 G01 X11.60
N050 G84 X10.60 Z-31.41
N055 G01 X10.60
N060 G84 X9.60 Z-30.75
N065 G01 X9.60
N070 G84 X8.60 Z-30.36
N075 G01 X8.60
N080 G84 X7.60 Z-30.17
N085 G01 X7.60
N090 G84 X7.10 Z-30.15
N095 G01 X7.10
N100 G84 X6.10 Z-21.28
N105 G01 X6.10
N110 G84 X5.10 Z-20.16
N115 G01 X5.10
N120 G84 X4.10 Z-8.42
N125 G01 X4.10
N130 G84 X3.10 Z-8.15
N135 G01 X3.10
N140 G84 X2.10 Z-6.66
N145 G01 X2.10
N150 G84 X1.10 Z-5.16
N155 G01 X1.10
N160 G84 X0.10 Z0.10
N165 G01 X0.10
!Finishing cuts.
N170 M03 S2000
N175 G00 X0 Z1
N180 G01 Z0 F120
N185 G02 X1.00 Z-1.00 R1.00
N190 G01 Z-5.25
N195 G01 X3.00 Z-8.25
N200 G02 X5.00 Z-10.25 R2.00
N205 G01 Z-20.25
N210 G01 X7.00 Z-22.50
N215 G01 Z-30.25
N220 G02 X12.50 Z-35.75 R5.50
N225 G01 Z-37.75
!Parting off.
N230 G00 X17.00
N235 G00 Z20
N240 M06 T5
N245 M03 S1600
N250 G00 Z-37.75
```

```
N255 G00 X14.50
N260 G01 X0 F90
N265 G00 X14.50
N270 G00 Z20
N275 M05
N280 M02
```

DENFORD STARTURN READING TURN4

```
N275  M06  T5
N280  M03  S1600
N285  G00  Z-24.00
N290  G00  X11.50
N295  G01  X0  F90
N300  G00  X11.50
N305  G00  Z20
N310  M05
N315  M02
```



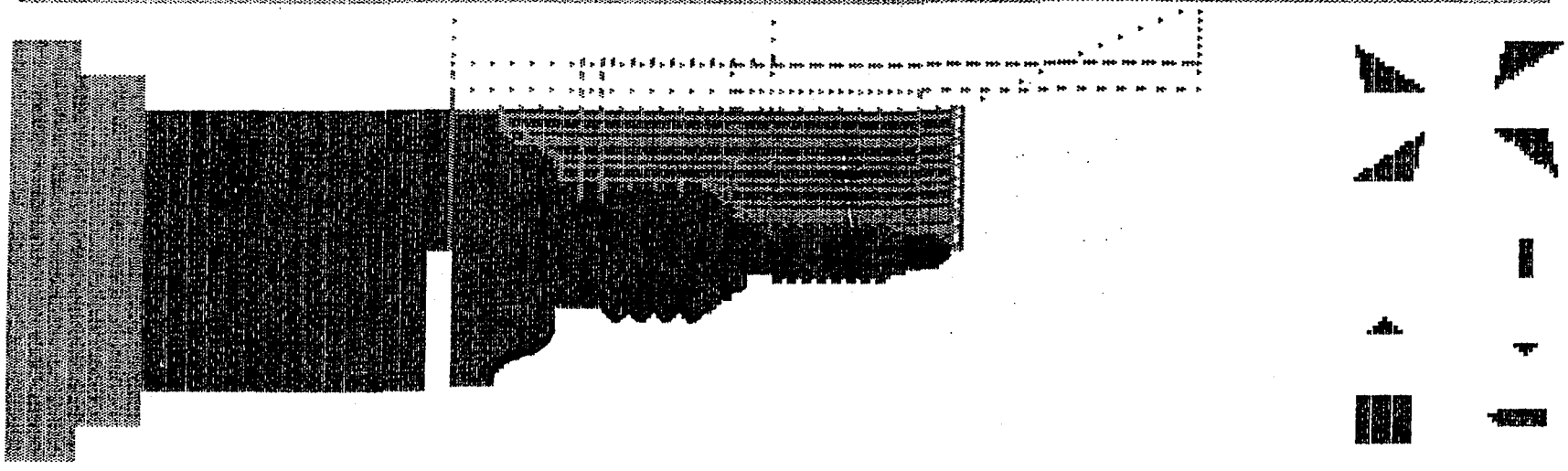
Grooving cuts.
Threading cuts.
Parting off.

```
@toolwidth 1.5
!Roughing cuts.
N005 G00 X14.00
N010 G00 Z20
N015 M06 T1
N020 M03 S1800
N025 G00 X10.00 Z1
N030 G01 Z1 F120
N035 G83 X0 Z0
N040 G84 X8.60 Z-19.99
N045 G01 X8.60
N050 G84 X8.10 Z-19.90
N055 G01 X8.10
N060 G84 X7.10 Z-19.90
N065 G01 X7.10
N070 G84 X6.10 Z-19.64
N075 G01 X6.10
N080 G84 X5.10 Z-18.06
N085 G01 X5.10
N090 G82 X2.35 Z-2.15 C4
N095 G01 X2.35
N100 G84 X1.35 Z-0.28
N105 G01 X1.35
N110 G84 X0.35 Z0.09
N115 G01 X0.35
N120 G84 X0.10 Z0.10
N125 G01 X0.10
!Finishing cuts.
N130 M03 S2000
N135 G00 X0 Z1
N140 G01 Z0 F120
N145 G02 X2.25 Z-2.25 R2.25
N150 G01 X5.00
N155 G01 Z-16.50
N160 G01 Z-18.00
N165 G03 X7.00 Z-20.00 R2.00
N170 G01 X8.00
N175 G02 X9.50 Z-21.50 R1.50
N180 G01 Z-24.00
!Grooving cuts.
N185 G00 X14.00
N190 G00 Z20
N195 M06 T5
N200 M03 S1600
N205 G00 Z-16.50
N210 G00 X7.00
N215 G01 X4.25 F90
N220 G00 X14.50
!Threading cuts.
N225 G00 X14.00
N230 G00 Z20
N235 M06 T3
N240 G00 X11.50
N245 G00 Z0.75
```

```
N250 G00 X5.00
N255 M03 S235
N260 G33 X4.08 Z-17.00 C31 P1.50
!Parting off.
N265 G00 X14.00
N270 G00 Z20
N275 M06 T5
N280 M03 S1600
N285 G00 Z-24.00
N290 G00 X11.50
N295 G01 X0 F90
N300 G00 X11.50
N305 G00 Z20
N310 M05
N315 M02
```

DENFORD STARTURN READING TURN5

```
N455  M06  T5
N460  M03  S1600
N465  G00  Z-40.50
N470  G00  X14.50
N475  G01  X0  F90
N480  G00  X14.50
N485  G00  Z20
N490  M05
N495  M02
```



Grooving cuts.
Threading cuts.
Parting off.

```
@toolwidth 1.5
!Roughing cuts.
N005 G00 X17.00
N010 G00 Z20
N015 M06 T1
N020 M03 S1800
N025 G00 X13.00 Z1
N030 G01 Z1 F120
N035 G83 X0 Z0
N040 G84 X12.10 Z-36.40
N045 G01 X12.10
N050 G84 X11.10 Z-36.14
N055 G01 X11.10
N060 G84 X10.10 Z-34.56
N065 G01 X10.10
N070 G84 X9.85 Z-34.40
N075 G01 X9.85
N080 G84 X8.85 Z-32.17
N085 G01 X8.85
N090 G84 X7.85 Z-31.57
N095 G01 X7.85
N100 G84 X6.85 Z-31.40
N105 G01 X6.85
N110 G84 X5.85 Z-19.39
N115 G01 X5.85
N120 G84 X4.85 Z-17.81
N125 G01 X4.85
N130 G84 X3.85 Z-17.27
N135 G01 X3.85
N140 G84 X3.35 Z-16.29
N145 G01 X3.35
N150 G84 X3.10 Z-16.15
N155 G01 X3.10
N160 G84 X2.35 Z-4.66
N165 G01 X2.35
N170 G84 X1.85 Z-1.69
N175 G01 X1.85
N180 G84 X0.85 Z-0.07
N185 G01 X0.85
N190 G84 X0.10 Z0.10
N195 G01 X0.10
!Finishing cuts.
N200 M03 S2000
N205 G00 X0 Z1
N210 G01 Z0 F120
N215 G02 X1.75 Z-1.75 R1.75
N220 G01 X2.25 Z-4.75
N225 G01 X3.00 Z-5.75
N230 G01 Z-14.75
N235 G01 Z-16.25
N240 G01 X3.25
N245 G03 X4.75 Z-17.75 R1.50
N250 G03 X6.75 Z-19.75 R2.00
N255 G01 X6.75 Z-20.75
```

```
N260 G01 Z-28.50
N265 G01 Z-30.00
N270 G01 Z-31.50
N275 G02 X9.75 Z-34.50 R3.00
N280 G01 X10.00
N285 G03 X12.00 Z-36.50 R2.00
N290 G01 X12.50
N295 G01 Z-40.50
!Grooving cuts.
N300 G00 X17.00
N305 G00 Z20
N310 M06 T5
N315 M03 S1600
N320 G00 Z-14.75
N325 G00 X5.25
N330 G01 X2.00 F90
N335 G00 X17.50
N340 G00 Z-28.50
N345 G00 X8.75
N350 G01 X5.25 F90
N355 G00 X17.50
N360 G00 Z-30.00
N365 G00 X8.75
N370 G01 X5.25 F90
N375 G00 X17.50
!Threading cuts.
N380 G00 X17.00
N385 G00 Z20
N390 M06 T3
N395 G00 X14.50
N400 G00 Z-2.75
N405 G00 X3.00
N410 M03 S235
N415 G33 X2.39 Z-15.25 C21 P1.00
N420 G00 X14.50
N425 G00 Z-17.75
N430 G00 X6.75
N435 M03 S205
N440 G33 X5.77 Z-29.00 C33 P2.00
!Parting off.
N445 G00 X17.00
N450 G00 Z20
N455 M06 T5
N460 M03 S1600
N465 G00 Z-40.50
N470 G00 X14.50
N475 G01 X0 F90
N480 G00 X14.50
N485 G00 Z20
N490 M05
N495 M02
```