CONECT NUMERICAL CONTROL LIMITED

UNIT 25 MOSSEDGE INDUSTRIAL ESTATE
LINWOOD
PAISLEY PA3 3HR

**********USER WARNING**********

The Operating System Software supplied with your Conect product is issued on a Site Licence Basis which means that you may make such copies as you wish of the Machine Operating Software and of the Manual providing it is only used within the premises to which the Machine was originally supplied.

The Software may not be copied or distributed for use off the Site to which the Machine was originally supplied and may not be sold or transferred in any form other than the original.

The acceptance and use of the Software and Manuals implies the acceptance and compliance with the above conditions.

CONECT LATHE OPERATING MANUAL

INDEX

INDEX

INDEX

CONECT CNC LATHE OPERATING MANUAL

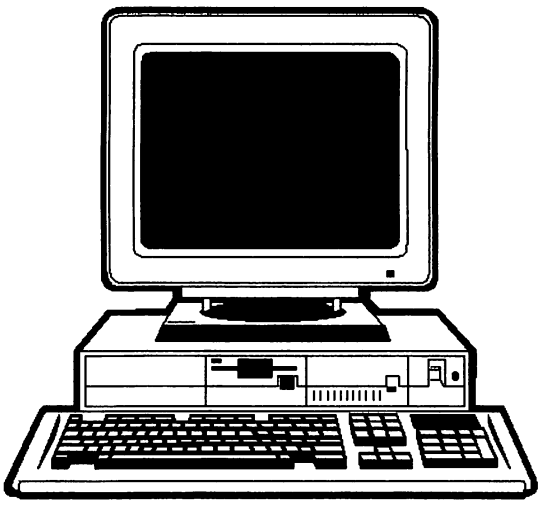
INSTALLATION

INSTALLATION

Computer Hardware

Minimum Hardware Requirements (For machine control only)

IBM pc or 100% compatible.
640Kb ram.
Single 720Kb floppy drive.
EGA/VGA colour graphics adaptor.
Mouse.
2 serial ports. (If machine link required and no mouse port).
Parallel Port.
Keyboard.



Important:-

If you intend to use the same computer for design and machine control then a hard drive must be fitted (40Mb).

Important:-

Before attempting to run the software, make a copy of the Conect Master disk.

Use the "DISKCOPY" command to format and copy all files to a disk with the same format ie 3.5" to 3.5" (1.44Mbyte).

If you are not sure about the copying commands checkout your Dos manual or ask for help.

Running From Floppy

If your computer does not have a hard drive, ignore the following installation procedure and run the software from floppy.

INSTALLATION

Installing Onto A Hard Drive

If you have a hard drive with at least 700Kb of free diskspace, follow the instructions below to Install from a master copy.

1 - Place the master copy into the floppy drive.

2 - Make sure the current drive is the correct drive for installation ie C:

3 - Type A:INSTALL (Enter)

4 - Follow the screen prompts to install.

Installing Onto A Floppy Drive ie - a: to b:

You can install the software onto a different disk format, but it must have at least 720Kb of free disk space.

1 - Format a blank floppy.

2 - Place the Master Disk into one drive and the blank disk into the other drive.

3 - Select the drive that has the blank disk. ie b:

4 - Type a:install

INSTALLATION

Directory Structure (Floppy or Hard Drive)

After the installation is completed the following directories and sub-directories will have been created.

\CONECTL	- Main executable files and data.	
\CONECTL\HELP\	- Helpfiles in DXF format.	[.DXF]
\CONECTL\TOOLS\	- Toolshapes in DXF format.	[.DXF]
\CONECTL\CNCFILES\	- Cnc programs.	[.CNL]
\CONECTL\DXF\	- Dxf profiles.	[.DXF]
\CONECTL\DEMOS\	- Demo files.	[.DEM]

Conectl Directory Filetypes

CONECTL.HLP	- Helpfiles.
CONECTL.TLS	- Tooloffsets for standard toolpost.
CONECTLR.TLS	- Tooloffsets for rotary toolpost.
CONECTL.SET	- Menus and errormessages.
CONECTL.TXT	- Screen layout and parameters.
CONECTL.EXE	- Main executable program.
MET_SHEL.EXE	- Graphics shell.
CONECTL.SKD	- Cad template.

Different Font types

SYSTEM00.FNT
SYSTEM08.FNT
SYSTEM16.FNT
SYSTEM24.FNT
SYSTEM32.FNT
SYSTEM48.FNT
SYSTEM72.FNT
ROMANSIM.FNT

ADDRESS.TXT	- Registration address.
PRINT.TXT	- Printer information.

Running The Software

To run from floppy or from the hard drive select the drive that contains the Conect software and type: "CONECTL".

INSTALLATION

Configuration

The first screen you will see allows for screen and mouse configuration. The system automatically detects the graphics card and the mouse in the computer.

You should be able to simply press Return to continue.

If there are problems you can select other graphics and mouse options. If the problem persists contact your dealer for help.

Changing Graphics Option

Press 'D' on the keyboard for a listing of the various graphics adaptors. Use the arrow keys to highlight your choice and press the return key to confirm your selection.

Note:- If you select an adaptor that is not compatible with the computer system you are using, you may hang the computer.

Changing the Mouse Option

Press 'I' on the keyboard for a listing of the various mouse options. Use the arrow keys to highlight your choice and press the return key to confirm your selection.

Conect Registration Screen

Following on from the configuration setup the registration screen is displayed. This contains the address of the suppliers and the customers name.

Important:-

Conect have a policy of supplying un-protected software under a licence agreement that restricts the use of the software to a single site.

CONECT CNC LATHE OPERATING MANUAL

MAINSCREEN

MAIN SCREEN

General Layout

After the software has been loaded for the first time, the main screen will be displayed. At the right-hand side of the screen you can see the programing and edit keys. The layout is typical of today's Cnc control systems and is commonly known as a MDI panel.

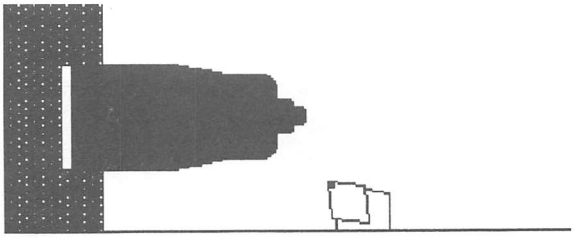
The Alpha-numeric keys are used for writing Cnc programs and the other keys for editing.

Note:-

MDI stands for Manual Data Input.

Under the MDI panel there is a small simulation screen for line by line simulation.

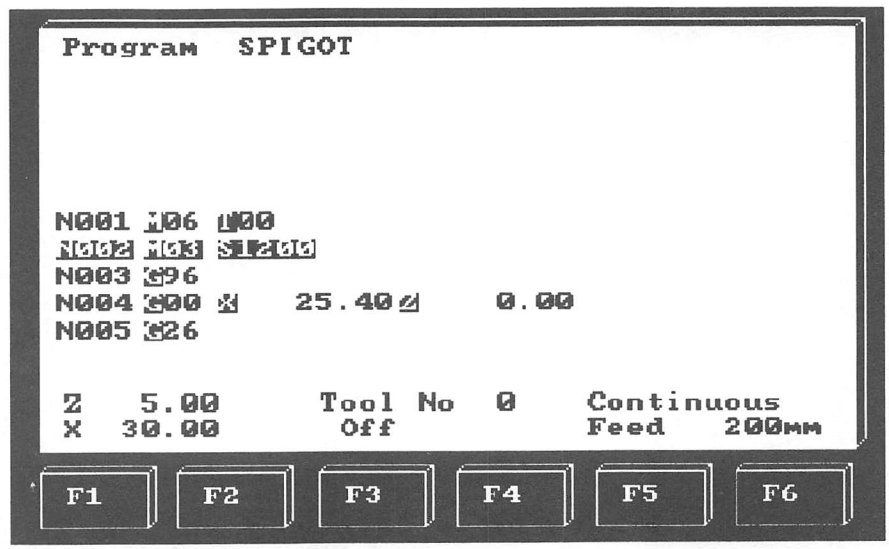
All tool movement and metal removal can be simulated in Auto or single step mode.



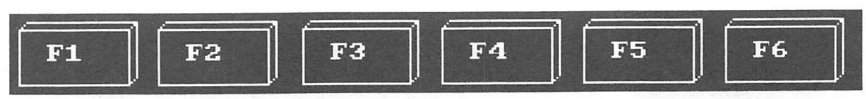
MAIN SCREEN

Editor Screen

On the left there is the editor where the Cnc blocks are displayed.



Just under the editor are the softkeys (numbered from F1 - F6).



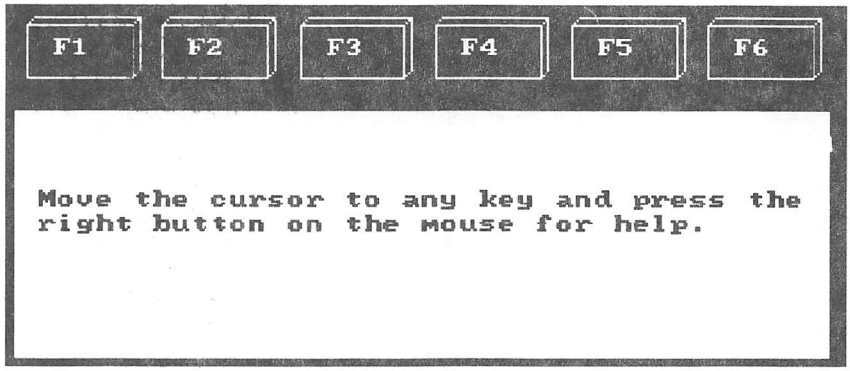
Note:-

Softkeys are used on Cnc control systems in the same way as function keys are used on the computer, to by-pass menu selections with a single keypress.

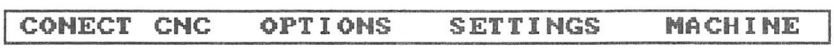
MAIN SCREEN

Display Screen

Under the softkeys you will find the help display screen.



Finally at the top of the screen there are three pulldown menus.



The 'Options' and 'Settings' menu can be selected from the Editor screen but the 'Machine' menu is only available from the Machine control screen.

MAIN SCREEN

HELP SYSTEM

Conect cnc lathe software has an advanced help system that works in several ways.

1 - Instant Help

For 'Instant help' on any of the buttons, menus or screen displays simply point with the mouse and press the right mouse button.

Help will appear in the help screen display window.

2 - More Help

To select more help click on the 'MORE HELP' button with the mouse or press **F10** on the computer keyboard.

After displaying all the available help pages for the selected topic the system returns to the first help page.

3 - Interactive Help

Interactive help appears automatically within the system. This option is commonly known as 'context sensitive help' and is mainly used during data entry to explain the G and M codes. For example **Fig 10** shows G02 diagramatic help which was drawn in AutoSketch and saved as a Dxf file.

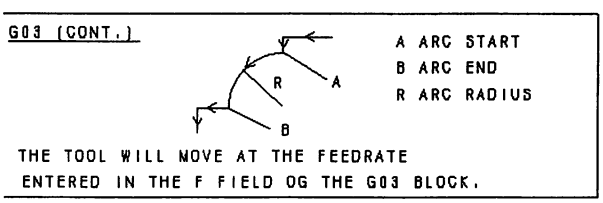


Fig 10

You can design your own help system very easily and integrate it into the Conect system.

MAIN SCREEN

TOOL LIST

Displays the machine tool offsets that are currently in memory.
You will be prompted to press the Esc key to clear the tool offset display.

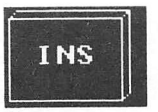
STEPMODE

Simulates the current line and moves to the next line of the program. Allows line by line simulation as the program is being written.You can start from any cnc block but do not start in the middle of a cycle.

CANCEL

The CANCEL key deletes the current block and restores the old value.

INSERT KEY



Inserts a block of information onto the current line in the CNC editor. If there is no block to insert a new line will be inserted in the CNC program.

DELETE KEY

Deletes the last character from the current block. If an Edit operation is not active the current line is deleted.

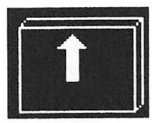
MAIN SCREEN

INPUT DXF

Reads in a DXF profile from Cad and automatically generates the Cnc part program.

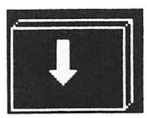
ARROWUP

Moves up one line in the editor. Use the mouse or the Left Arrow key on the computer keyboard.



ARROWDOWN

Moves down one line in the editor until the end of the program is reached. Use the mouse or the Right Arrow key on the computer keyboard.

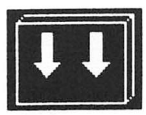


PAGEUP

Moves the editor up one page or to line number one, whichever is the nearest. Use the mouse or the Up Arrow key from the computer keyboard.

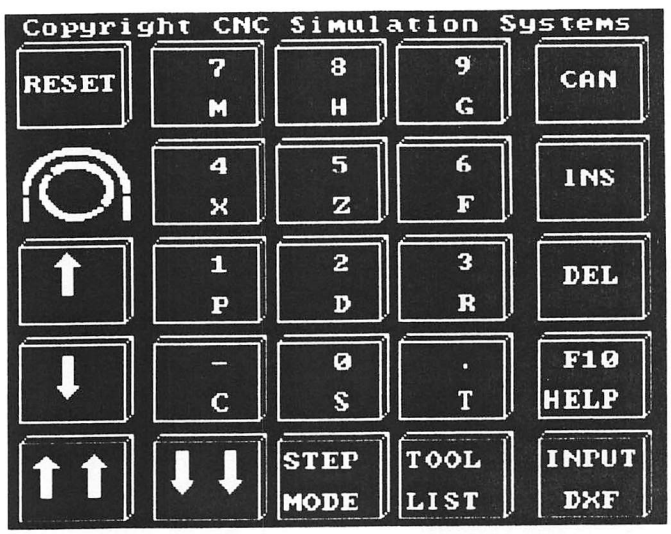
MOVE DOWN A PAGE

Moves the editor down one page or to the end of the program, whichever is the nearest. Use the mouse or the Down Arrow key on the computer keyboard.



ASCIIKEYS

All the alpha-numeric keys contain letters and numbers for part program generation. To write a program the keys must be used in a certain order. See writing cnc programs for more info.



MAIN SCREEN

RESET THE SYSTEM

Restores all the values to a software switch on condition. This option should be selected before starting to Record a demofile. If you use this option all your cnc program will be lost.

MORE HELP

Selects more help if more help is available. Using the right button on the mouse selects the first help page for that topic.
More Help will scroll from one help related page to the next.

CONECT CNC LATHE OPERATING MANUAL

SOFTKEYS

SOFTKEYS

To select any of the Softkeys you can point with the mouse and press the left mouse button or press the relevant function key on the computer keyboard.

F1 - SIMULATE PROGRAM

Starts the simulation from line one. To stop simulating press the Esc key.

If there are any tool changes you will be prompted for the Tool number. The tool shapes will be displayed in the 'Help Display Screen' to assist with your selection.

If the Tool No displayed is correct simply press Return to accept, or type in a different number.

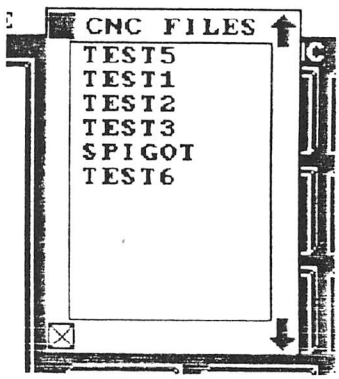
Note:-

If your system is configured for a Rotary Toolpost you will not be prompted for a Tool number.

SOFTKEYS

F2 - LOAD PROGRAM

A pull down menu showing the available files from the current directory will be displayed. To select, highlight a file using the mouse or the up and down arrow keys. Click with the left mouse button or press the Return key to confirm.



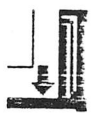
The selected file will be loaded from Disk and displayed in the editor.

The shape of the finished part will be displayed in the simulation screen. This will allow you to browse through the cnc programs till you find the part you want.

This is a useful feature if you can't remember the filename.

Page Swapping

To display more files click on the top right arrow or bottom right arrow with the mouse or press the PageUp or PageDn keys on the computer keyboard.



Changing Directory

To change the current Directory click on the top left black button with the mouse or press the 'D' key on the computer keyboard. You will be prompted for the Directory name.

Example:- c:\conectl\cncfiles

Clearing The Menu

To clear the menu click on the bottom left button with the mouse or press the Esc key.

SOFTKEYS

F3 - SAVE PROGRAM

Prompts for a filename and saves the program to Disk. A filename can only have 8 characters maximum in its name. The file will be saved in the current Directory. To change the current Directory use softkey F2 and select the 'change directory' option as described on the previous page.

F4 - RECORD KEYSTROKES

Records all the users keystrokes in a file on disk This file can later be selected to replay the students work.

Important

It is recommended to press the 'RESET' button before selecting this option. This will reset the system to its startup condition

You will be prompted for a filename (a maximum of 8 characters only may be used) after typing in the filename press the Return key to confirm. To cancel this option press the Esc key.

After selecting a filename you will notice the message 'RECORDMODE' is displayed on the 'Menu Bar' at the top of the screen. This will remain until you click on the F4 softkey to close the file.

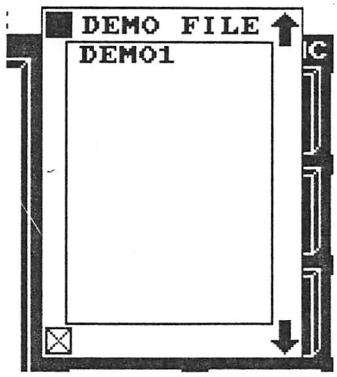
SOFTKEYS

F5 - REPLAY KEYSTROKES

Important

It is recommended to press the 'RESET' button before selecting this option. This will reset the system to its startup condition.

A pull down menu showing the available files from the current directory will be displayed. To select, highlight a file using the mouse or the up and down arrow keys. Click with the left mouse button or press the Return key to confirm.



Page Swapping

To display more files click on the top right arrow or bottom right arrow button with the mouse or press the PageUp or PageDn keys on the computer keyboard.

Changing Directory

To change the current Directory click on the top left black menu button with the mouse or press the 'D' key on the computer keyboard. You will be prompted for the Directory name.

Example:- C:\conectl\demos

Type in the Directory name and press return to confirm.If you want to cancel this operation press the Esc key.

Clearing The Menu

To clear the pull down menu, click on the bottom left menu button or press the Esc key on the computer keyboard.

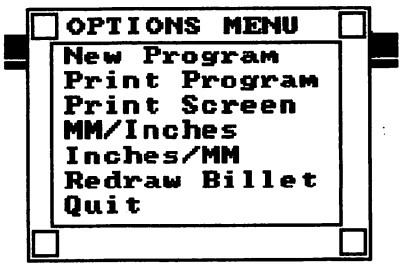
F6 - MACHINE SIMULATION

Switches to the machine simulation screen. If you want to return to the editor screen press the F6 softkey again.

CONECT CNC LATHE OPERATING MANUAL

OPTIONS MENU

OPTIONS MENU



Select the Options Menu by clicking on the Menu Bar or by pressing Ctrl-s on the computer keyboard.

New Program

Select this option to start a new part program. Be sure the current program in memory has been saved.
Type in the filename, upto a maximum of eight characters and press the return key.
The current program will be erased and the editor will be ready for input beginning on line one.

Print Program

Outputs the current part program to the printer if one is connected.

Example Of Printout

```
CONECT CNC TRAINING SYSTEM    Page  1
CNC PROGRAM  TEST1
Billet Length 50
Billet Diameter 25.4
N001 M06 T 00
N002 M03 S 2100
N003 G96
N004 G00 X    25.60 Z     0.00
N005 G26
N006 G01 X     4.00 Z     0.00 F   120
N007 G01 X     4.00 Z    -2.00 F
N008 G01 X     6.00 Z    -2.00 F
N009 G01 X     6.00 Z    -4.00 F
N010 G01 X     8.00 Z    -5.00 F
N011 G01 X     8.00 Z    -6.00 F
N012 G01 X    14.00 Z    -6.00 F
N013 G01 X    14.00 Z   -19.60 F
N014 G02 X    22.00 Z   -23.60 F       R   4.00
N015 G01 X    24.00 Z   -23.60 F
N016 G01 X    24.00 Z   -30.20 F
N017 G01 X    25.60 Z   -30.20 F
N018 G01 X    25.60 Z          F
N019 G28
N020 G97
N021 G92
N022 M05
N023 M30
```

OPTIONS MENU

Print Screen (F9)

Print screen outputs the displayed screen image to the printer.
Select the correct driver from the Settings Menu if this has not been set.
Pressing the F9 key on the computer keyboard, at anytime in the software, will also output the screen image.

You will be prompted to continue Y/N after menu selection.

Note:-

All printer output is through the parallel port.

MM/Inches

Converts the current part program from millimetres to inches. To convert the system from Metric to Imperial see the Settings Menu.

Inches/MM

Converts the current part program from inches to millimetres. To convert the system from Imperial to Metric see the Settings Menu.

Note:-

If you change the units of measure you may have to change the billet dimensions to suit.

Redraw Billet

Redraws the billet and chuck. Useful for refreshing the screen after a Tool collision.

Quit

Prompts the user Y/N before ending program execution.

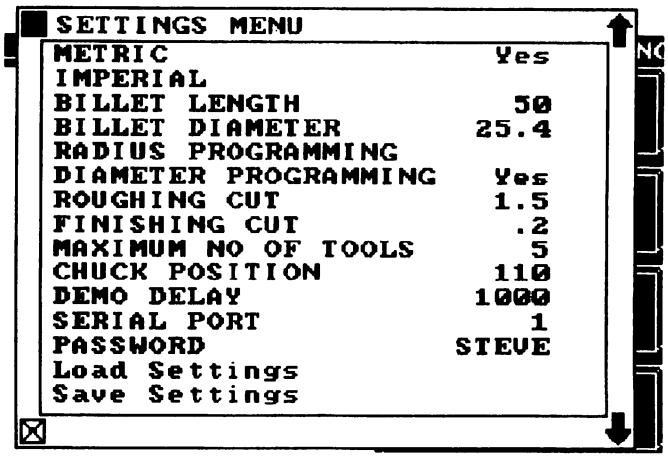
CONECT CNC LATHE OPERATING MANUAL

SETTINGS MENU

SETTINGS MENU

SETTINGS MENU

How To Change The Settings



Click on the Settings Menu Bar or press 'Alt S' on the computer keyboard.

Select any of the settings options by moving the mouse and clicking with the left button or using the cursor keys and pressing the Return key.

Click on the top right arrow button or press the PgUp key to display the last settings page.

Click on the bottom right arrow button or press the PgDn key to display the next settings page.

Metric

To program in Metric select this option. Always check to see if the other settings parameters are in Metric ie Billet Length. All axes positions will be shown to two decimal places of accuracy.

Feed will be shown in mm/min.

Imperial

To program in Imperial select this option. Always check to see if the other settings parameters are in Imperial ie Billet Length. All axes positions will be shown to four decimal places of accuracy.

Feed will be shown in ins/min.

SETTINGS MENU

Billet Length

After selection Type in the length of Billet and press the Return key.

The new value will be displayed in the Settings Menu. The length is the amount of material from the chuck jaw to the end of the bar.
If the length is too large for the screen it will be automatically scaled down.
Any change in length will be automatically added to the tool offsets.

Billet Diameter

After selection type in the diameter value and press the Return key.

The new value will be displayed in the Settings Menu. The billet on the screen will be re-drawn with the new values.

Radius Programming

All 'X' axis moves are programmed as radii dimensions.

Example:-

G00 X10.00 Z-10.00
would move the Tool to a position 10mm from the centre of the bar and -10mm on the 'Z' axis.

Diameter Programming

All 'X' axis moves are programmed as Diameter positions.

Example:-

G00 X10.00 Z-10.00
would move to a position 5mm from the centre of the bar and -10mm on the 'Z' axis.

SETTINGS MENU

Roughing Cut

This value is used in the automatic roughing cycle for the amount of metal that will be removed from the radius of the bar for each roughing pass of the tool.

Finishing Cut

This value is used in the automatic roughing cycle as the amount of metal that will be left for a final finishing pass after the roughing operation is complete.

Maximum No Of Tools

This value specifies the no of tools to be made available within the software system. The maximum is 99.

Chuck Position

Sets the position of the chuck up and down the screen. May be required at a future date for different tooling set ups.

Demo Delay

Demo programs are a series of keystrokes or mouse actions that are saved to disk. When these are replayed the delay in between each operation can be set. This will assist students to set the speed of the demos to suit their own needs.

Serial Port

The default serial port for linking to the Cadet lathe is Port 2.This can be set to any value between 1-4. You must ensure that your computer does have the selected serial port number.

SETTINGS MENU

Password

Not implemented.

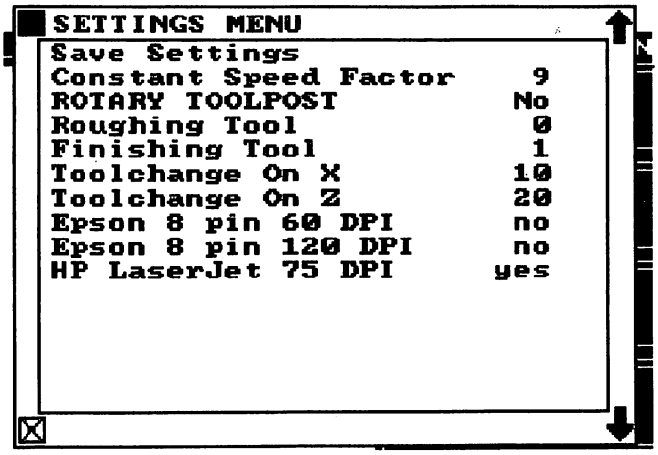
Load Settings/Save Settings

To load or save settings select either of these options.You will be prompted for a filename. The default filename on switch on is CONECTL. Save under this name if you want the settings to be loaded when you run the software.

SETTINGS MENU

Constant Speed Factor

This value can be set to any number between 1 - 9. The higher the value the higher the spindle speeds. To take advantage of this feature you must use G96 to switch CSS on. G97 switches it off. Gcode help will give you more info.



Rotary Toolpost Yes/No

Toggles between Rotary Toolpost and Standard Tooling.

Toolchange On X

Sets the tool change position on the 'X' axis. This parameter is a diameter value.

Toolchange On Z

Sets the tool change position on the 'Z' axis. This value should be well clear of the billet position.

Roughing Tool

Sets the Roughing Tool number that will be used when importing Dxf profiles.

Finishing Tool

Sets the Finishing Tool number that will be used when importing Dxf profiles.

The following printer options can be selected by highlighting with the mouse or the arrow keys.

Epson 9 Pin

Epson 24 Pin

Laserjet 70 DPI

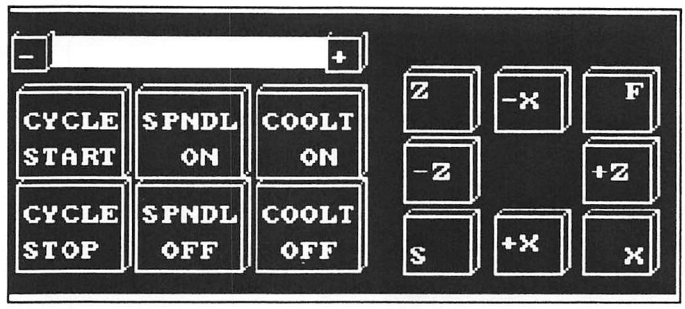
Laserjet 150 DPI

Laserjet 300 DPI

SIMULATION SCREEN

SIMULATION SCREEN

To enter the simulation screen press the F6 softkey from within the editor screen.
At the bottom right you will see the control panel for operating the software and the Conect lathe.

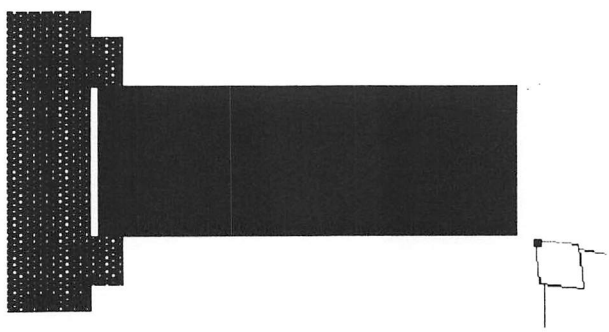


To the left the help screen works exactly the same as the main screen.

CYCLE START..... F1

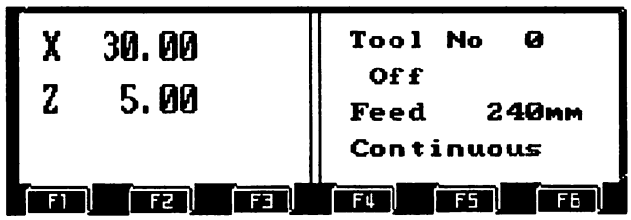
Starts the simulation from line one to the end of the program. To stop the simulation click on Cycle Stop or press the F1 key again.

At the top left the chuck and the billet are shown ready for the simulation of the Cnc part program.

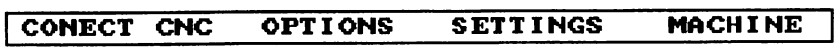


SIMULATION SCREEN

To the right all the positional display and the softkeys. This display is the same for the simulation and the machine.



Both the main screen and the simulation screen have the same menu bar at the top of the screen. You can only access the Machine Menu from within the simulation screen.

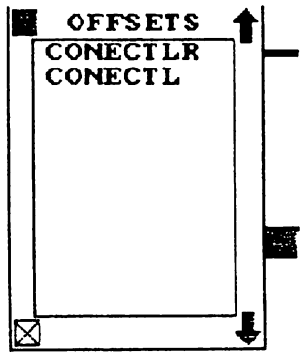


SIMULATION SCREEN

AUTO SIMULATE..... F1

Behaves like the Cycle Start option.

LOAD TOOL OFFSETS..... F2



Lists all the Tool Offset Files in a pull down window. Point with the mouse or use the Up and Down arrow keys to select a file. Click on the bottom or top arrow to select more files. Click on the top left black button to change Directory.

To quit click on the bottom left button or press the Esc key.

AUTO ZOOM..... F3

Sets the zoom factor to one of five settings.
If the length or the diameter is too large for the screen it will be automatically scaled down.

SELECTS TOOL..... F4

Use this option to change tools. Ensure the screen tool has been moved clear of the billet to avoid collisions.

Standard Tooling

Prompts for a new Tool number. Enter the number and press Return. The old tool shape will be erased and the new shape displayed. The Tool number cannot be larger than the 'Maximum No Of Tools' value from the Settings Menu.

Rotary Tooling

Automatically moves onto the next Tool. The old tool shape will be erased and the new shape displayed.

If the next Tool number is greater than the 'Maximum No Of Tools' value in the Settings Menu then Tool No 0 will be selected.

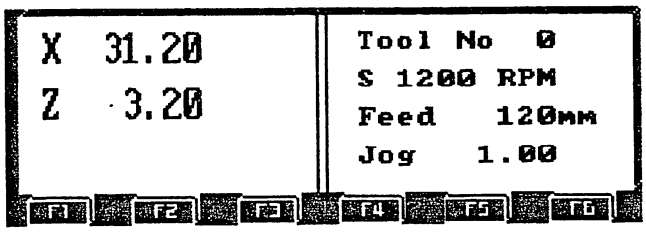
SIMULATION SCREEN

CONTINUOUS OR JOG MODE.... F5

Selects continuous or jogmode.

Continuous Mode

In continuous the screen tool travels at the set feedrate until the mouse button or the keys on the computer are released.



Jog Mode

Jog mode can be any of three values:-

1.0mm

0.10mm

0.01mm

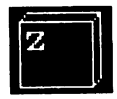
Each keypress or mouse action will jog the Tool the set value.

EDITOR SCREEN..... F6

Switches the software back to the first editor screen. No program information or tool positional information is lost by switching screens.

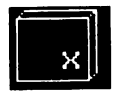
SIMULATION SCREEN

DATUM Z AXIS..... Z



When the Tool is at the correct position on the Z axis, normally at the end of the bar, this option will zero the Z position and record a new Tool Offset for the current Tool.

DATUM X AXIS..... X



When the Tool is at the correct position on the X axis, normally on the Outside Diameter of the billet this option will set the 'X' position to the diameter value specified in the Settings Menu and record a new Tool Offset for the current Tool.

The system will display the Diameter value and record a new Tooloffset for the current Tool.

When To Datum The Screen Tools

All the Tool Offsets for the screen tools are preset and do not require setting unless:-

1 - You are teaching how to datum tools in the simulation.

2 - You have designed your own tools and want to set their offsets.

3 - Someone has changed the offsets by accident.

Note:-

If you have not selected 'Save Tooloffsets' from the 'Machine Menu' then loading the offsets (F2) will reset all the offsets to their original values.

Important:-

Always check that the correct tool has been selected before setting the 'tool datum' of any tool.

Changing the billet length or the diameter will automatically adjust the tool offsets.

SIMULATION SCREEN

ZMINUS MOVING IN NEGATIVE Z.... Left Arrow

If you are in 'Continuous Mode' keeping the left button on the mouse pressed will move the Tool along the 'Z' axis in a negative direction. If you are in 'jog mode' the Tool will move the set jog amount for each mouse click or keypress.

ZPLUS MOVING IN POSITIVE Z .... Right Arrow

If you are in 'Continuous Mode' keeping the left button on the mouse pressed will move the Tool along the 'Z' axis in a positive direction.If you are in 'Jog Mode' the Tool will move the set amount for each mouse click or keypress.

XMINUS MOVING IN NEGATIVE X.... Up Arrow

If you are in 'Continuous Mode' keeping the left button on the mouse pressed will move the Tool along the 'X' axis in a negative direction. If you are in 'Jog Mode' the Tool will move the set amount for each mouse click or keypress.

XPLUS MOVING IN POSITIVE X.... Down Arrow

If you are in 'Continuous Mode' keeping the left button on the mouse pressed will move the Tool along the 'X' axis in a positive direction. If you are in 'Jog Mode' the Tool will move the set amount for each mouse click or keypress.

SIMULATION SCREEN

TURN SPINDLE ON.

Switches the spindle on and displays the current spindle speed.

TURN SPINDLE OFF

Switches the spindle off and the spindle speed display is set to 'Off'.

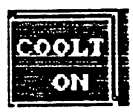
SPINDLE SPEED DOWN

If the spindle has been switched on this option will reduce the spindle speed. Release the mouse button to cancel.

SPINDLE SPEED UP

If the spindle has been switched on this option will increase the spindle speed to a maximum of 2500. Release the mouse button to cancel.

COOLANT ON



Switches the Coolant On.

COOLANT OFF

Switches the Coolant Off.

SIMULATION SCREEN

CYCLE START..... F1

Starts the simulation from line one to the end of the program.
To stop the simulation click on Cycle Stop or press the Esc key.

CYCLE STOP..... Esc

Stops the simulation. You cannot restart the simulation rom the line you stopped at. Cycle Start will reset to line one. Use Step Mode if you want to simulate from other start lines.

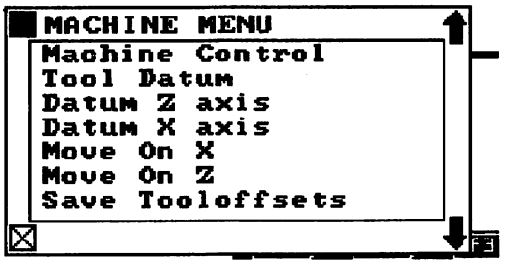
FEEDRATE UP..... +

Increases the Feedrate to a maximum of 2000mm/min.
The new feedrate is displayed.

FEEDRATE DOWN.... -

Decreases the Feedrate to a minimum of 100mm/min.
The new Feedrate is displayed.

MACHINE MENU



To select the Machine Menu you need to be in the Simulation Screen. Press F6 to switch between the Main Screen and the Simulation Screen.

Machine Control Off/On

Switches communications between the lathe and the computer to On or Off.

Machine Control On

If you are switching to Machine Control On for the first time since the Conect lathe was powered up, you will be prompted to datum the 'X' and the 'Z' axes.
When the Conect lathe is switched Off it loses it's position and will only find it again when both axes have been sent to datum.

Note:-

Switching the computer Off and On will not result in any positional errors.

Communication Errors

When Machine Control is On the software constantly checks the communications for any data corruption. If you are having problems switch the machine and the computer off and check the following:-

a) Check that all cabling is secure.
b) Check the Com port is correctly configured.
c) If you are still having problems get the cable checked.
d) Try a different computer.
e) If all else fails contact your Dealer.

MACHINE MENU

Tool Datum

This only applies to machines with a Rotary Toolpost.

Procedure:-

This procedure only needs to be done when the machine has been switched on.

1) Set Machine control on and index the rotary toolpost to tool position zero using the F4 key.

2) Set Machine control off and set the screen tool to zero with the F4 key.

3) Set Machine control on and select the Tool Datum option from the Machine menu.

Note:-

If you leave the machine set to position Tool zero before switching it off the Tool Datum will be set the next time you use the machine.

Datum Z axis/Datum Xaxis

If you are linking to the machine for the first time since the machine was switched on, you will be forced to datum both axes before any machine control can take place. The first axis must be the 'X' axis.

When To Datum

Although you are forced to datum at 'switch on' you may need to datum at other times:-

a) If the emergency button has been pressed in.
b) If excessive force has been used during the machining operations.
c) If the Tool seems to be out of position.
d) If the machined part has measurement errors.

Note:-

If you press the emergency button always remember to release it before trying to communicate again.

MACHINE MENU

Move On X

This option is only available if the machine is connected and has been sent to datum.

Type in the absolute position you want the machine to move to. Remember to type in a diameter if you configured for diameter programming and a radius value if you are configured for radius programming.

To stop the machine press the esc key on the computer keyboard or the emergency button on the Conect lathe.

Move On Z

This option is only available if the machine is connected and has been sent to datum.

Type in the absolute position you want the machine to move to.

To stop the machine press the esc key on the computer keyboard or the emergency button on the Conect lathe.

Save Tool Offsets

Offsets are needed for each individual Tool. There are two different sets of offsets.

Simulation Offsets

All the tools that are supplied with the Conect lathe software have their offsets stored on disk. These are preset and only need changing if the tool drawings are changed or you add some more tool drawings to the system.

MACHINE MENU

Machine Tool Offsets

Machine tool offsets need to be set before any machining is started.
We suggest all the tools are set when the machine arrives.

Standard Tooling

If you are using standard tooling (no rotary toolpost) then the tooloffsets are stored in the file 'Conectl.tls'. These offsets will be automatically loaded when you run the Conect lathe software.

Rotary Tooling

If you are using standard tooling (no rotary toolpost) then the tool offsets are stored in the file 'Conectlr.tls'. These offsets will be automatically loaded when you run the Conect lathe software.

Remember

If you save your tooloffsets as 'Conectl' or as 'Conectlr' they will be automatically loaded into memory the next time you run the software.
Both the tooloffsets for the simulation tools and the machine tools are stored in the same file.

MACHINE OPERATION

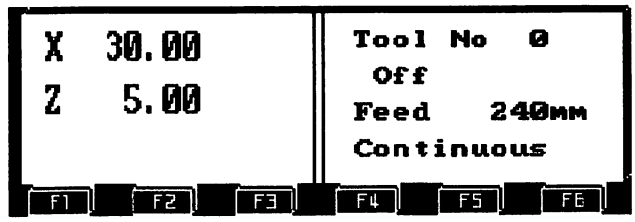
Introduction

This section describes the machine operation, from the computer and at the machine.

Note:-

To move the tool on the machine you must use the machines own axis buttons.

The axis buttons in the software control the movement of the tool on the screen. They cannot be used for controlling the machine.!



At the top right the display shows the axes positions, tool number, spindle status, feed and feedmode.

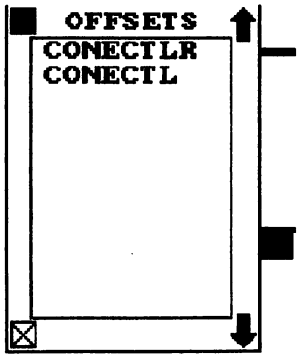
MACHINE OPERATION

SOFTKEYS

AUTO SIMULATE..... F1

Behaves like the Cycle Start option.

LOAD TOOL OFFSETS..... F2



Lists all the Tool Offset Files in a pull down window. Point with the mouse or use the Up and Down arrow keys to select a file. Click on the bottom or top arrow to select more files. Click on the top left black button to change Directory.

To quit click on the bottom left button or press the Esc key.

AUTO ZOOM..... F3

Sets the zoom factor to one of five settings.
If the length or the diameter is too large for the screen it will be automatically scaled down.

SELECTS TOOL..... F4

Use this option to change tools. Ensure the tool has been moved clear of the component to avoid collisions.

Standard Tooling

If you select a different tool on the machine use this option to select a similar tool for the simulation.

Rotary Tooling

Automatically rotates to the next tool position on the machine and on the screen.

If the next Tool number is greater than the 'Maximum No Of Tools' value in the Settings Menu then Tool No 0 will be selected.

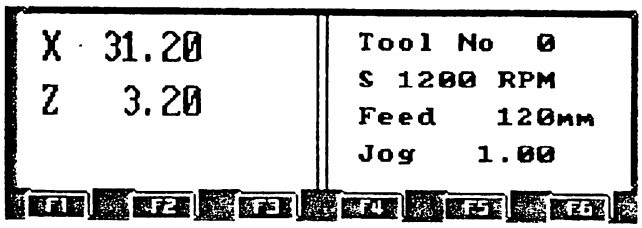
SOFTKEYS

CONTINUOUS OR JOG MODE.... F5

Selects continuous or jogmode.

Continuous Mode

In continuous the machine will move at the set feedrate until the axis button on the machine is released.



Jog Mode

Jog mode can be any of three values:-

1.0mm

0.10mm

0.01mm

Pressing any of the axis buttons on the machine will jog the Tool the set value.

EDITOR SCREEN..... F6

Switches the software back to the first editor screen. No program information or tool positional information is lost by switching screens. Machine control can still be active from the 'editor screen' and the machine can still be moved around with the axis buttons.

DATUM Z AXIS..... Z (Software button)

When the Tool is at the correct position on the Z axis, normally after facing the end of the bar, this option will zero the Z position and record a new Tool Offset for the current Tool.

Action:-

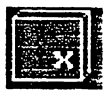
1) Move the tool along the 'Z' axis till it is near the end of the bar.

2) Set the Jog option to 1.0mm and move closer to the end of the bar.

3) Set the jog option to .01mm and move along the 'Z' axis until the tool just touches the end of the bar.

4) Datum Tool.

DATUM X AXIS..... X (Software button)



When the Tool is at the correct position on the X axis, normally after taking a cut on the Outside Diameter of the billet this option will prompt for the diameter and record a new Tool Offset for the current Tool.

Action:-

1) Move the tool along the 'Z' axis till it is near the end of the bar.

2) Move the tool along the 'X' axis until it is just below the diameter.

3) Set the feedrate to a sensible cutting feed and move along the 'Z' axis taking cut along the diameter.

4) Datum Tool.

MACHINE OPERATION

When To Datum Tools

1 - You are teaching how to datum tools on the machine.

2 - You have fitted some new tools that have no offsets.

3 - Someone has changed the offsets by accident.

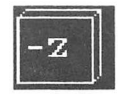
Note:-

If you have not selected 'Save Tooloffsets' from the 'Machine Menu' then loading the offsets (F2) will reset all the offsets to their original values.

Important:-

Always check that the correct tool has been selected before setting the 'tool datum' of any tool.

ZMINUS MOVING IN NEGATIVE Z



If you are in 'Continuous Mode' keeping the button on the machine pressed will move the Tool along the 'Z' axis in a negative direction. If you are in 'jog mode' the Tool will move the set jog amount for each button press.

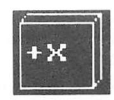
ZPLUS MOVING IN POSITIVE Z

If you are in 'Continuous Mode' keeping the button on the machine pressed will move the Tool along the 'Z' axis in a positive direction.If you are in 'Jog Mode' the Tool will move the set jog amount for each button press.

XMINUS MOVING IN NEGATIVE X

If you are in 'Continuous Mode' keeping the button on the machine pressed will move the Tool along the 'X' axis in a negative direction. If you are in 'Jog Mode' the Tool will move the set jog amount for button press.

XPLUS MOVING IN POSITIVE X



If you are in 'Continuous Mode' keeping the button on the machine pressed will move the Tool along the 'X' axis in a positive direction. If you are in 'Jog Mode' the Tool will move the set jog amount for each button press.

Fast Traverse

To fast traverse the machine press the middle of the five buttons and then any of the axes buttons. Do not fast traverse when the tool is near to the component.

TURN SPINDLE ON.

Switches the spindle on and displays the current spindle speed. You may have to wait for several seconds before the programed spindle speed has been reached.

TURN SPINDLE OFF

Switches the spindle off and the spindle speed display is set to 'Off'.

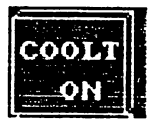
SPINDLE SPEED DOWN

If the spindle has been switched on this option will reduce the spindle speed. Release the mouse button to cancel.

SPINDLE SPEED UP

If the spindle has been switched on this option will increase the spindle speed to a maximum of 2500rpm. Release the mouse button to cancel.

COOLANT ON



Switches the Coolant On.

COOLANT OFF

Switches the Coolant Off.

MACHINE OPERATION

CYCLE START..... F1

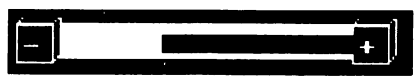
Starts the simulation and the machine operation from line one to the end of the program.
The simulation is always two blocks in front of the machine operation.
To stop the simulation click on Cycle Stop or press the Esc key.
Use the emergency button if required.

CYCLE STOP.....F1

Stops the machine operation. You cannot restart the machining from the line you stopped at. Cycle Start will reset to line one.

FEEDRATE UP..... +

Increases the Feedrate to a maximum of 2000mm/min.
The new feedrate is displayed.



FEEDRATE DOWN.... -

Decreases the Feedrate to a minimum of 10mm/min.
The new Feedrate is displayed.

MACHINE OPERATION

EMERGENCY, STOP

Be prepared to use the emergency stop if required. It will stop the spindle and all machine movement.
Lifting the guard up during a machine operation will also cause an emergency stop.

Note:-

After an emergency stop:-

a) Switch off the machine.

b) Release the emergency stop.

c) Switch the machine control off in the software.

d) Switch the machine on.

e) Switch the machine control on in the software.

f) Datum the axes.

Guard Override

For setting tools you can use the key supplied with the Conect lathe to override the guard.

Remember to switch back to override off when you have finished.

CONECT CNC LATHE OPERATING MANUAL

DEFINING TOOLS

DEFINING TOOLS

DXF TOOLSHAPES

Tools can be defined in Cad and passed to the system through the DXF option.

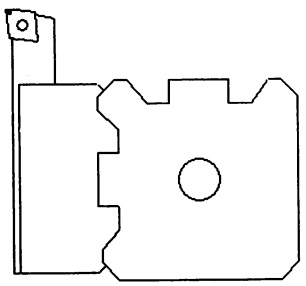
Note:-

You will see below some of the tool shapes that are supplied with the system. Use these existing Dxf shapes as a template to design your own toolshapes.

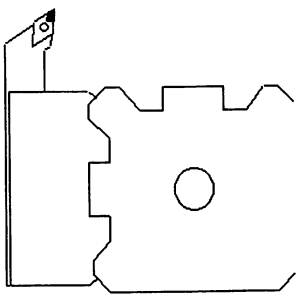
Important:-

When defining Toolshapes use only the line command and restrict yourself to a maximum of 100 entities maximum. Remember the more entities you draw the slower the animation when simulating tool paths.

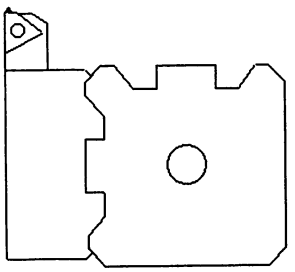
Standard Tooling



Tool No 0



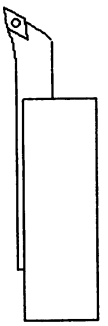
Tool No 3



Tool No 5

Rotary Tooling

For Rotary toolpost user's we supply the same number of pre-drawn toolshapes. If you select the Rotary option in the Settings Menu the Rotary toolshapes will automatically be selcted.



Tool No 1

DEFINING TOOLS

Saving Toolshapes To Disk

The toolshapes supplied with the system are stored as dxf files with the name 'TOOL' plus the number.

Example Tool number 1 is 'TOOL1.DXF'.

If you define a new tool give it a name above 'TOOL5.DXF' or you could overwrite existing toolshapes.

Important:-

Ensure the toolshapes are stored in the \CONECTL\TOOLS\ directory or the system will not be able to find them.

Change the Maximum number of tools value in the settings menu to the updated number.

Important:-

Rotary Toolpost Users.

If you are configured for a Rotary Toolpost the toolshapes have different names.

Tool number 1 - 'TOOLR1.DXF'

Tool number 5 - 'TOOLR5.DXF'.

DXF PROFILES

DXF PROFILES

Cad Profiles

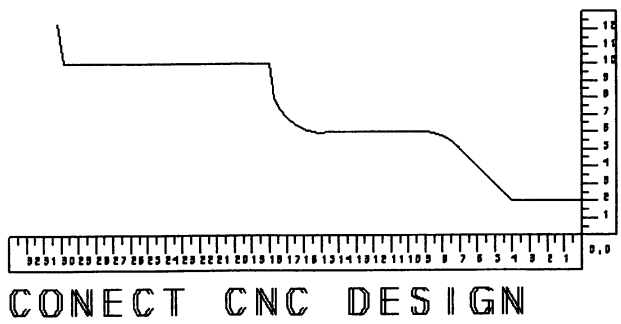
Profiles defined in Cad can be output to the Conect lathe system through the DXF link.
We support all Cad systems that have this option.

Note:-

DXF stands for Data Exchange Format and is available with the majority of Cad systems. AutoSketch is the Cad system used in this manual to describe how the link works.

Drawing Template

We supply a drawing template with the Conect lathe software that can be loaded into Autosketch as a drawing. Templates are not required to define profiles but can assist in the process.



The drawing template is called 'CONECTL.SKD' and can be automatically loaded when you run the AutoSketch program by typing 'SKETCH CONECTL'.

Note:-

Copy the template drawing into the 'SKETCH' directory before issuing the above command.

DXF PROFILES

Design Rules

To successfully pass your design from the Cad system to the Conect lathe software, you must adhere to certain rules.

a) Check the settings menu for

i Roughing Tool Number

ii Finishing Tool Number

iii Roughing Cut

iv Finishing Cut

Note:-

Save the settings to disk if you want the new values to be automatically loaded when running the software.

b) Defining profiles in Cad

i Use only lines and arcs.

ii Use the snap or attach to connect entities.

iii Do not edit the template.

iv Valid profiles must have a bigger diameter from left to right.

v Do not use more than 99 entities for a single profile.

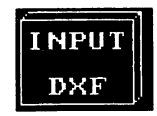
c) Saving Dxf files

i Make a Dxf file to save the profile.

ii Save the Dxf file in the correct directory. The default directory for the dxf files is \conectl\dxf.

DXF PROFILES

Reading Dxf Files



When you select the 'INPUT DXF' option from the MDI panel you will be prompted for a filename. If you have stored the DXF profile from your Cad system in the default directory it will be listed in the pull down window.

If you have selected a different directory you can click on the top left black button of the pull down menu or press the 'D' key on the computer to select another directory.

Speeds and Feeds

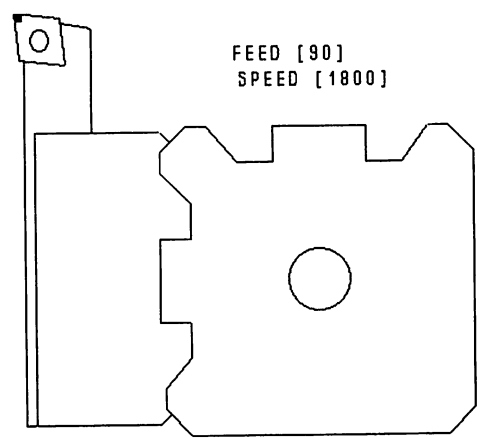
Speeds and feeds are automatically inserted into the part program. The values are default values or your own values specified in the Dxf file that relates to the tool number you have selected.

Below you can see Tool1 which is stored on disk as 'Tool1.dxf' in the \tools directory.

By selecting the 'text option' in your cad system the values for speed and feed can be set for each tool. These values will be selected by the Conect software and inserted into the part program for you.

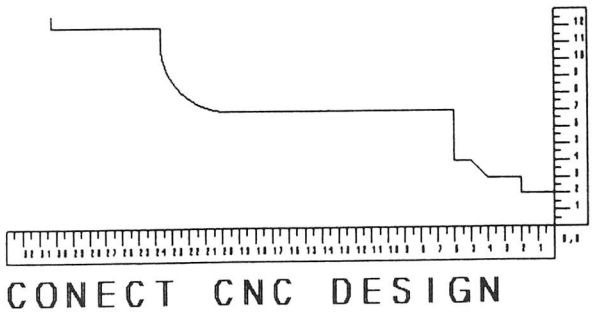
Note:-

The position and size of the text does not matter.



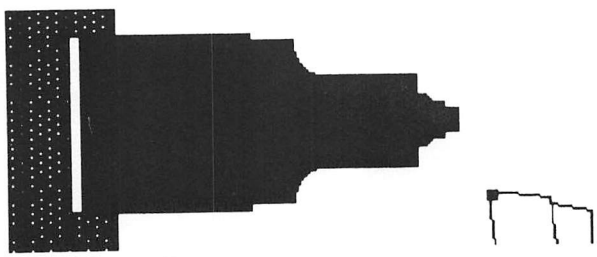
DXF PROFILES

Example Test1



Test1.dxf

CONECT CNC TRAINING SYSTEM
CNC PROGRAM TEST1 Billet
Length 50
Billet Diameter 25.4

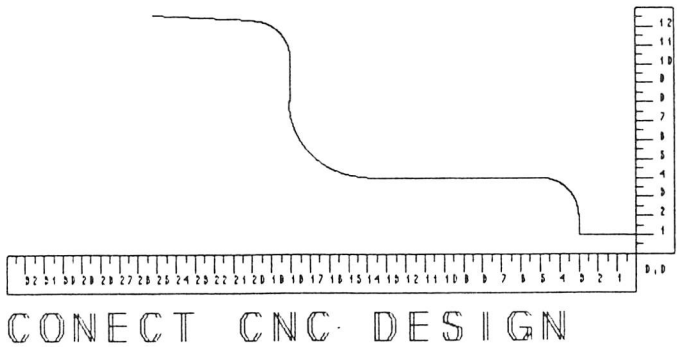


Test1.cnl

```
N001 M06 T 00
N002 M03 S 2100
N003 G96
N004 G00 X    25.60 Z     0.00
N005 G26
N006 G01 X     4.00 Z     0.00 F   120
N007 G01 X     4.00 Z    -2.00 F
N008 G01 X     6.00 Z    -2.00 F
N009 G01 X     6.00 Z    -4.00 F
N010 G01 X     8.00 Z    -5.00 F
N011 G01 X     8.00 Z    -6.00 F
N012 G01 X    14.00 Z    -6.00 F
N013 G01 X    14.00 Z   -19.60 F
N014 G02 X    22.00 Z   -23.60 F       R   4.00
N015 G01 X    24.00 Z   -23.60 F
N016 G01 X    24.00 Z   -30.20 F
N017 G01 X    25.60 Z   -30.20 F
N018 G01 X    25.60 Z          F
N019 G28
N020 G97
N021 G92
N022 M05
N023 M30
```

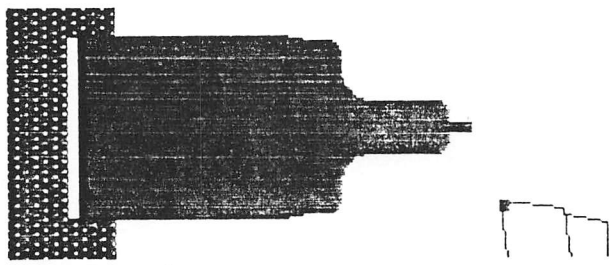
DXF PROFILES

Example Test2



Test2.dxf

CONECT CNC TRAINING SYSTEM CNC PROGRAM TEST2
Billet Length 50
Billet Diameter 25.4

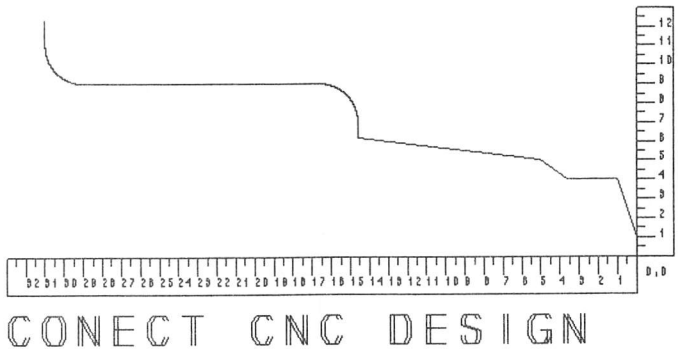


Test2.cnl

```
N001 M06 T 00
N002 M03 S 21000
N003 G96
N004 G00 X    25.40 Z
0.00
N005 G26
N006 G01 X     2.00 Z     0.00 F   120
N007 G01 X     2.00 Z    -3.00 F
N008 G01 X     4.00 Z    -3.00 F
N009 G03 X     8.00 Z    -5.00 F       R   2.00
N010 G01 X     8.00 Z   -14.00 F
N011 G02 X    16.00 Z   -18.00 F       R   4.00
N012 G01 X    20.62 Z   -18.00 F
N013 G03 X    24.60 Z   -19.89 F       R   2.00
N014 G01 X    25.20 Z   -25.20 F
N015 G01 X    25.40 Z          F
N016 G28
N017 G97
N018 G92
N019 M05
N020 M30
```

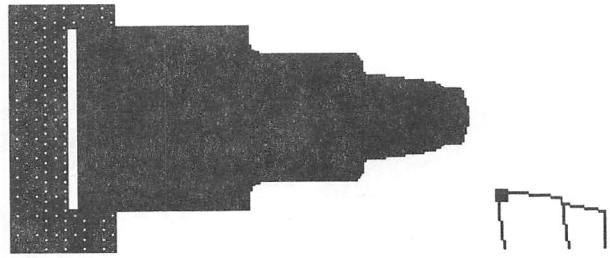
DXF PROFILES

Example Test3



Test3.dxf

CONECT CNC TRAINING SYSTEM CNC PROGRAM TEST3
Billet Length 50
Billet Diameter 25.4



Test3.cnl

```
N001 M06 T 00
N002 M03 S 2100
N003 G96
N004 G00 X    25.40 Z     0.00
N005 G26
N006 G01 X     2.00 Z     0.00 F   120
N007 G01 X     8.00 Z    -1.00 F
N008 G01 X     8.00 Z    -3.60 F
N009 G01 X    10.00 Z    -5.00 F
N010 G01 X    12.40 Z   -14.60 F
N011 G01 X    14.00 Z   -14.60 F
N012 G03 X    18.00 Z   -16.60 F       R   2.00
N013 G01 X    18.00 Z   -29.00 F
N014 G02 X    22.00 Z   -31.00 F       R   2.00
N015 G01 X    24.80 Z   -31.00 F
N016 G01 X    25.40 Z          F
N017 G28
N018 G97
N019 G92
N020 M05
N021 M30
```

CONECT CNC LATHE OPERATING MANUAL

CNC PROGRAMMING

CNC PROGRAMMING

Cnc stands for computerised numerical control. For a cnc control to be able to do anything you need to write a 'part program' in the language the control will understand.

G and M codes

The standard language for most Cnc controls is known as G and M code programming. A program to machine the component or the machine part is known as a 'part program'.

The following section will describe in detail all the G and M codes for the Conect lathe.

Programming Mode.

Before starting to program you need to be aware of the different programming modes that can be selected.

Metric/Imperial

Select Metric or Imperial programming from the Settings Menu. You cannot mix units within the same part program.

Diameter/Radius Programming

Select Diameter or Radius programming from the Settings Menu. You cannot mix these two modes of programming.

Diameter programming simply means all moves in the 'X' axis will be diameter moves.

Radius programming simply means all moves in the 'X' axis will be radii moves.

Absolute/Incremental

Absolute programming will move the tool to the programmed distance from the datum position.

Incremental programming will move the tool the programmed distance.

CNC PROGRAMMING

X and Z AXES

The Conect lathe has two axes:-

Cross Axis

The cross axis moves the tool along the face of the billet and is known as the 'X' axis for progaming purposes.

Longitudinal Axis

The longitudinal axis moves the tool along the length of the billet and is known as the 'Z' axis for programing purposes.

Nblock Numbers

Nblock numbers are equivalent to line numbers and they are automatically inserted by the Conect system.

Linear Interpolation

The Conect lathe is capable of straight line moves at any angle. These moves are known as Linear moves or Linear Interpolation.

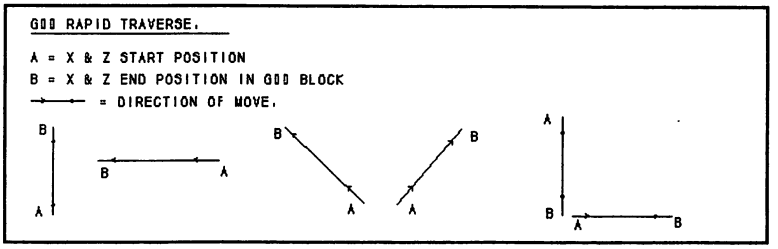
Circular Interpolation

The Conect lathe is capable of circular moves from 0 to 180 degrees at the specified radii. These moves are known as Circular Interpolation.

G CODES

G00 Fast Traverse Ex G00 X26.45 Z-12.45

Moves at the highest possible Feedrate to the programmed position. Use this command when the tool is well clear of the billet.

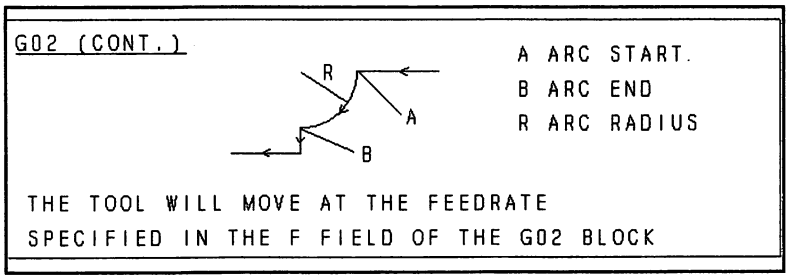


G01 Linear Traverse Ex G01 X26.45 Z-12.45 F 120

Moves at the programmed Feedrate to the programmed position. Use this command when the tool is required to cut metal.

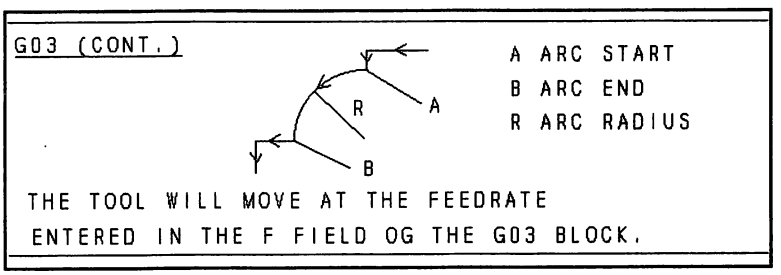
G02 Clockwise Circular Interpolation
Ex G02 X25.46 Z-12.56 R10 F123

Moves in a circular direction at the programmed feedrate to machine the programmed radius.



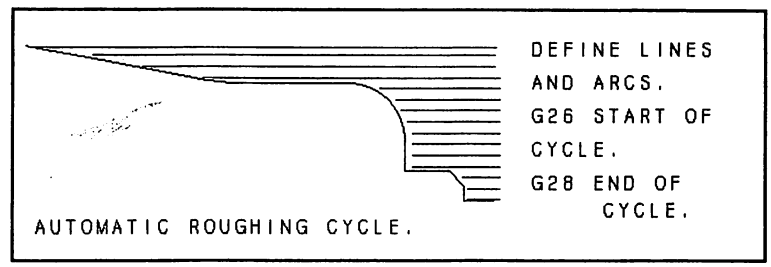
G03 Anti clockwise Circular Interpolation
Ex G02 X25.46 Z-12.56 R10 F123

Moves in a circular direction at the programmed feedrate to machine the programmed radius.



G26 Start of Automatic Roughing Cycle Ex G26

All the programmed moves after a G26 and upto a G28 End of Cycle define a finished profile. You can use lines and arcs.
All the roughing cuts will be automatically calculated by the system. A final finishing cut with the option to use a different tool will finish off the cycle.



AUTOMATIC ROUGHING CYCLE.

How It Works.

All profiles have to go from left to right and from a small diameter to a larger diameter.

Start Position

The start position is very important.

Z axis :- Start at the beginning of the profile.

X axis :- Start at the end of the profile.

Check the settings for :-

a) Roughing Tool Number

b) Finishing Tool Number

c) Roughing Cut

d) Finishing cut

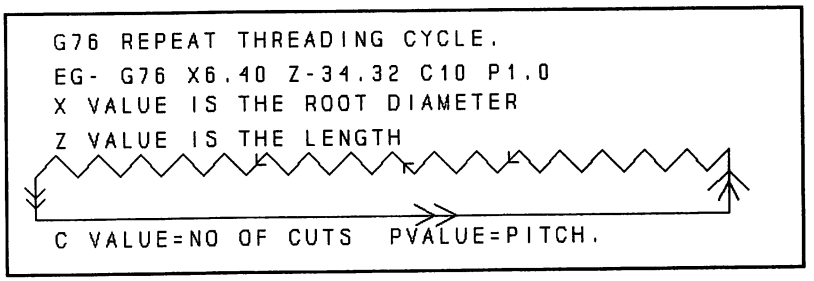
The Tool numbers for the roughing and finishing are inserted automatically. They may need changing to suit standard tooling or rotary tooling.

G28 End of Cycle Ex G28

Must follow a G26.

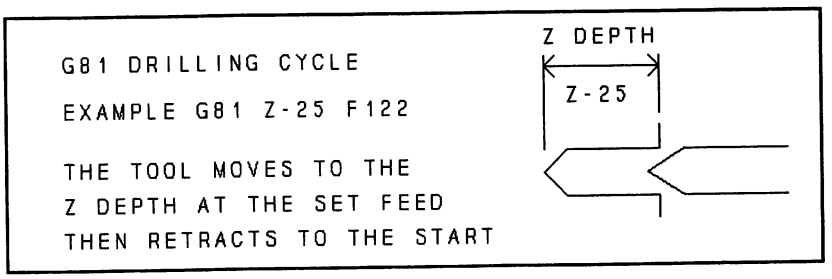
G76 Repeat Threading Cycle Ex G76 X5.12 Z-23.6 C10 P1.2

Cuts a thread at the specified root diameter and pitch. The 'C' field sets the no passes. The system will do 3 spring passes automatically at the end.



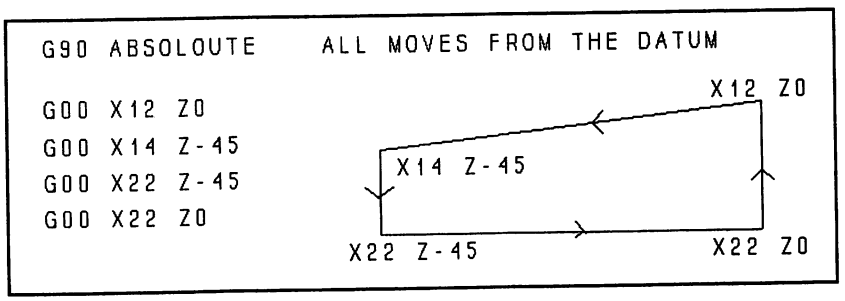
G81 Drilling Cycle Ex G81 Z-25 F122

Moves the Tool to the specified 'Z' depth and then feeds back to the start position.



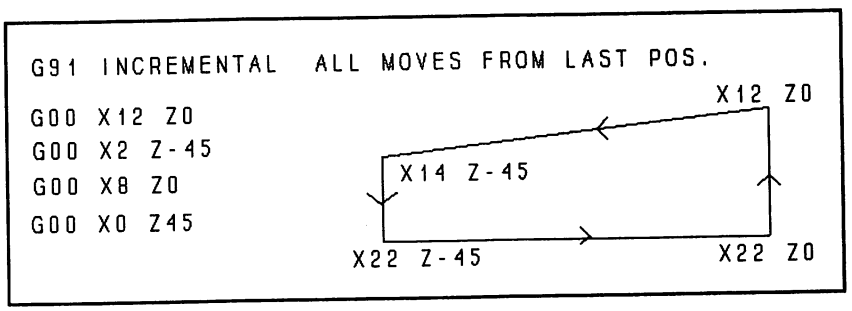
G90 Absolute Programing Ex G90

All moves relate to the datum position.



G91 Incremental Programing Ex G90

All moves are taken from the last programed position.



G70 Imperial Programming

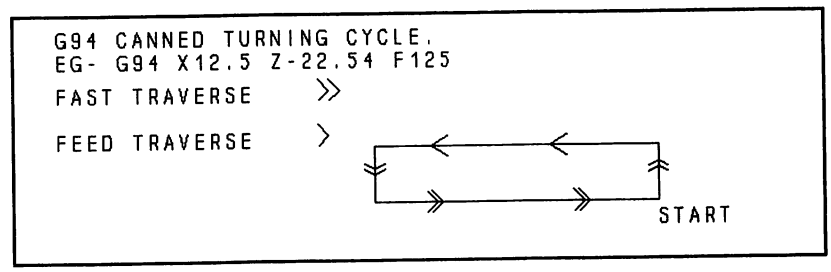
If you want to program in imperial insert the 'G70' code at the beginning of the program. This code is not always required as the units of measure are set in the settings.

G71 Metric Programming

If you want to program in metric insert the 'G71' code at the beginning of the program. This code is not always required as the units of measure are set in the settings.

G94 Canned Turning Cycle (parallel) Ex G94 X12.5 Z-22.54 F125

Use this cycle to remove rectangular blocks of material.



The 'G94' cycle has 4 moves:-

a) Fast Traverse to the cutting diameter.

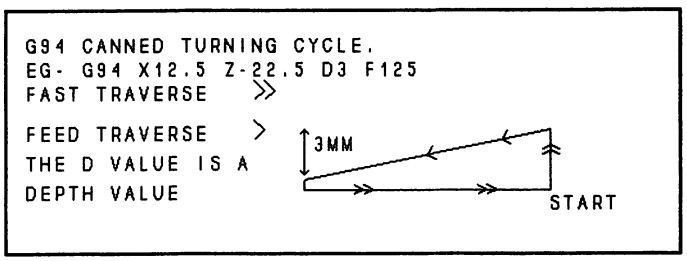
b) Feeds at the programmed rate to the final 'Z' position.

c) Feeds back to the start diameter.

d) Fast traverse back to the 'Z' start position.

G94 Canned Turning Cycle (Positive angle)
Ex G94 X12.5 Z-22.5 D3 F125

Specifying a positive 'D' value will produce a positive angle. The radial difference between the start diameter and the end diameter will be the 'D' value.



The 'G94' cycle has 4 moves:-

a) Fast Traverse to the cutting diameter.

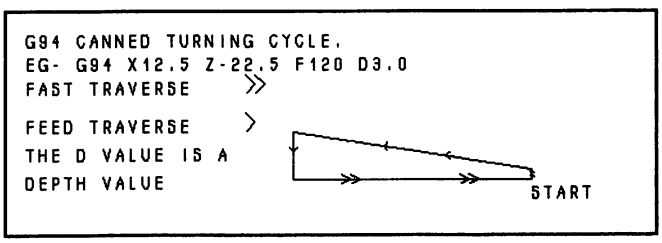
b) Feeds at the programmed rate to the final 'Z' position and to the start diameter + the 'D' value.

c) Feeds back to the start diameter.

d) Fast traverse back to the 'Z' start position.

G94 Canned Turning Cycle (Negative angle)
Ex G94 X12.5 Z-22.5 D-3 F125

Specifying a negative 'D' value will produce a negative angle. The radial difference between the start diameter and the end diameter will be the 'D' value.



The 'G94' cycle has 4 moves:-

a) Fast Traverse to the cutting diameter.

b) Feeds at the programmed rate to the final 'Z' position and to the start diameter - the 'D' value.

c) Feeds back to the start diameter.

d) Fast traverse back to the 'Z' start position.

G96 Constant Surface Speed On

Turns Constant surface speed on. The software will calculate the optimum cutting speed for the diameter of bar.

G97 Constant Surface Speed Off

Turns Constant surface speed off.

M CODES

M codes are miscellaneous codes that don't directly relate to moving the axes.

M03 Spindle On Ex M03 S1500

Switches the spindle on at the programmed speed specified by the 'S' value. All spindle speeds are shown as Revs Per Minute.

The spindle must be switched on after a new Tool has been selected.

```
Ex:- M06 T01
     M03 S1500
```

M05 Spindle Off Ex M05

Switches the spindle on.
The spindle must be switched off before a Tool Change.

```
Ex:- M05
     M06 T02
```

M30 End Of Program Ex M30

Used at the end of a part program. Should be preceded by a move to tool change and spindle off.

```
Ex:- G92
     M05
     M30
```

M06 Tool Change Ex M06 T01

Standard Tooling

Tells the system to wait for a Tool Change and prompts for the Tool Number. If the programmed tool number is the tool number you want then simply press return, otherwise type in the tool number followed by the return key.

Rotary Tooling

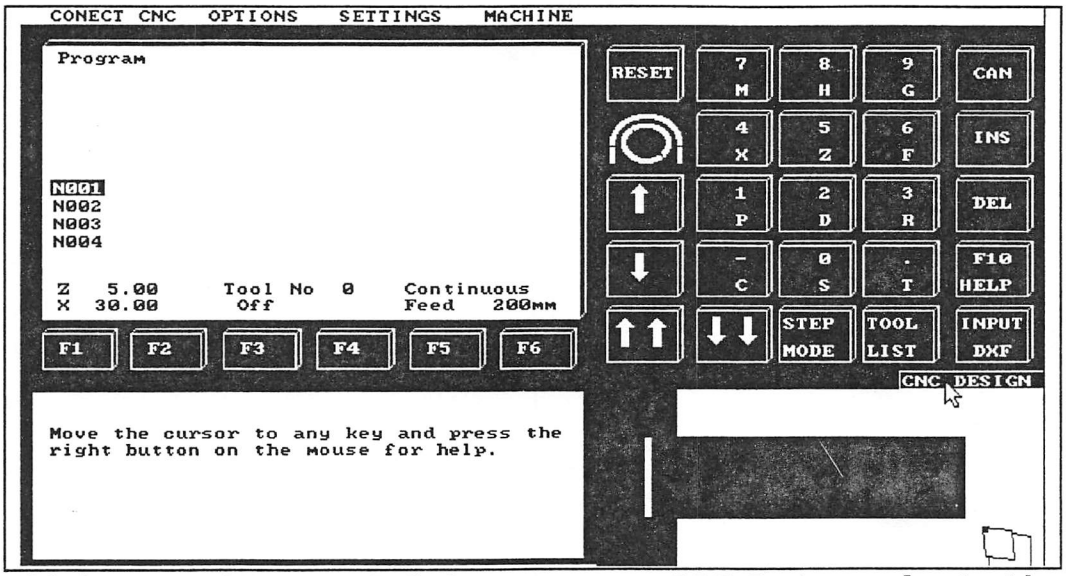
Selects the programmed tool number by indexing the rotary toolpost to the programmed tool position.

Use the 'G92 Move to Toolchange' code before issuing a tool change.

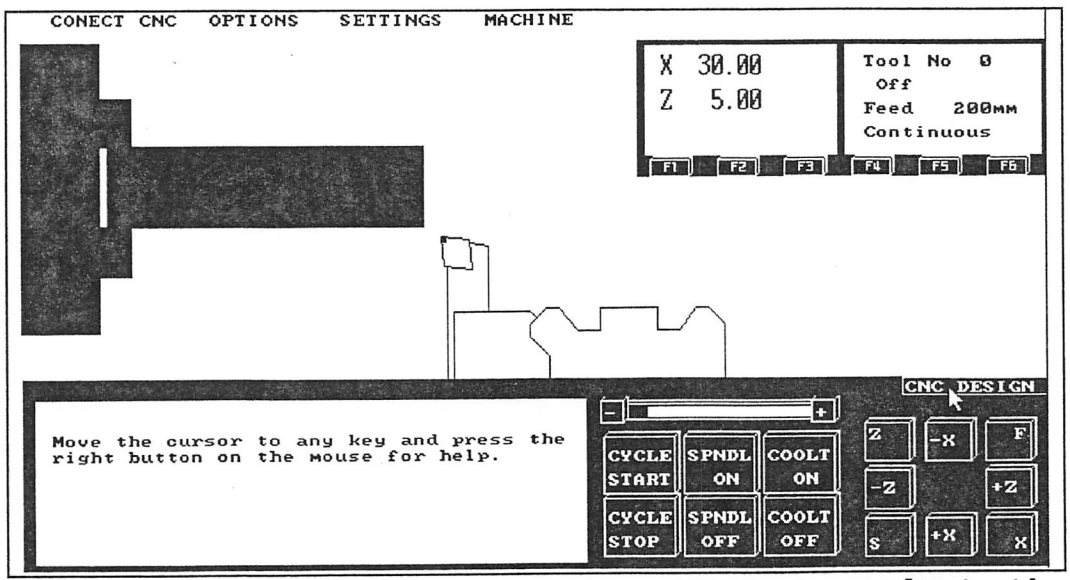
```
Ex:- G92
     M05
     M06 T02
```

PROFILE DESIGNER

CONECT LATHE PROFILE DESIGNER



Click on 'CNC DESIGN' or press ALT-D to select the Profile Designer.

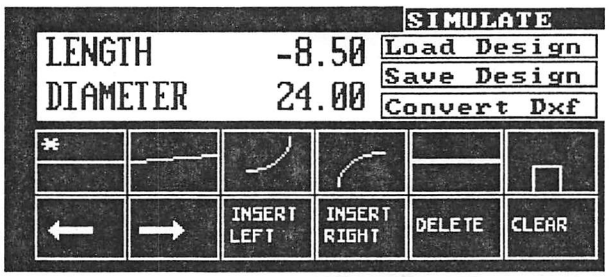


Click on 'CNC DESIGN' or press Alt-D to select the Profile Designer.

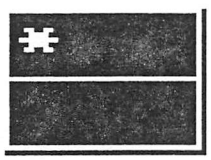
CONECT LATHE PROFILE DESIGNER

Selecting a Profile Shape.

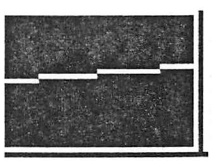
You can select from six different profile shapes from the menu by moving the cursor to the 'menu icon' and clicking with the mouse. Alternatively you can press key F1-F6 on the keyboard. After selecting a different shape an asterisk will highlight the active shape.



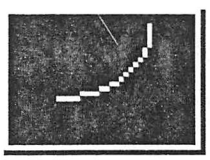
Cylinder (F1)



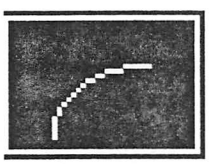
Taper (F2)



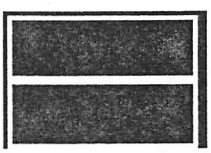
Clockwise Arc (F3)



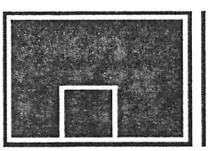
Anti Clockwise Arc (F4)



Thread (F5)



Groove (F6)



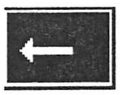
Important:

Before you insert a new shape make sure the correct shape has been selected.
The first time you run the software the default shape is always a cylinder.

CONECT LATHE PROFILE DESIGNER

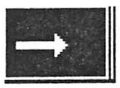
Profile Edit Options

Move to the Left (Ctrl Left Arrow)



Moves the current highlighted shape to the left. If there are no more profile shapes to the left, the first shape on the profile will be highlighted.

Move to the Right (Ctrl Right Arrow)



Moves the current highlighted shape to the right. If there are no more profile shapes to the right, the last shape on the profile will be selected.

Insert Left (Alt-L)

Inserts a new shape to the left of the current highlighted profile shape. If the profile is too long or too big on diameter the new shape will not be inserted.

Insert Right (Alt- R)

Inserts a new shape to the right of the current highlighted profile shape. If the profile is too long or too big on diameter the new shape will not be inserted.

Delete Shape

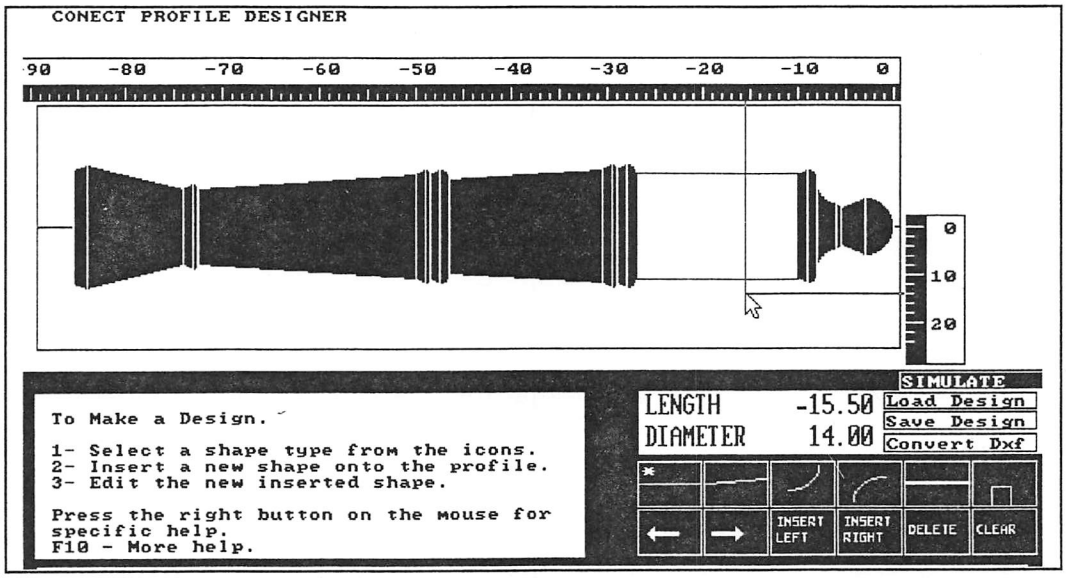
Deletes the current highlighted profile shape. The first profile shape to the left becomes the new highlighted profile shape.

Clear Profile

Clears all the profiles shapes except the first one. The first profile shape becomes the new active profile shape.

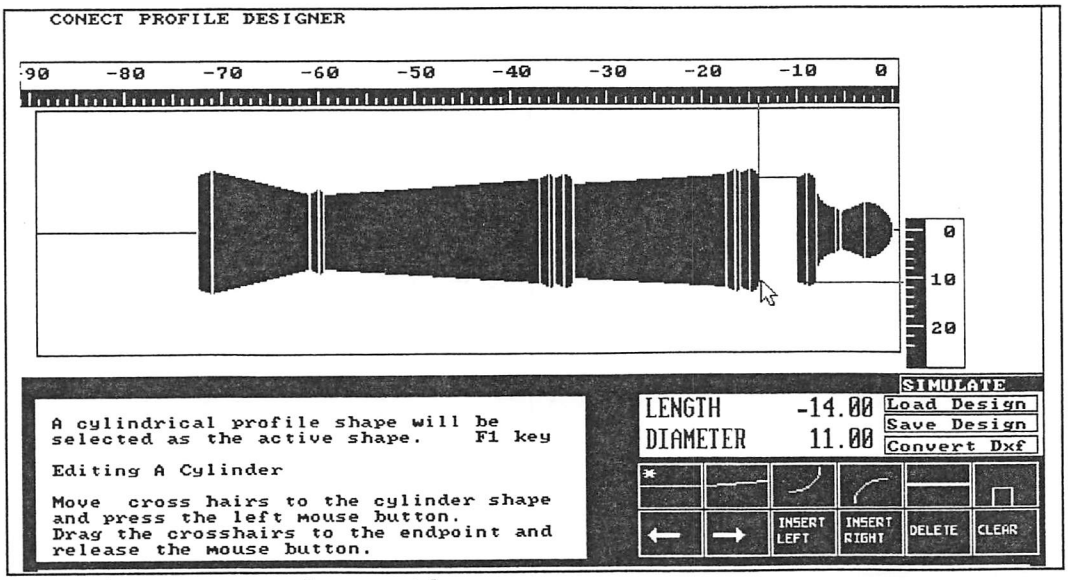
CONECT LATHE PROFILE DESIGNER

Editing Cylinders



Select start position

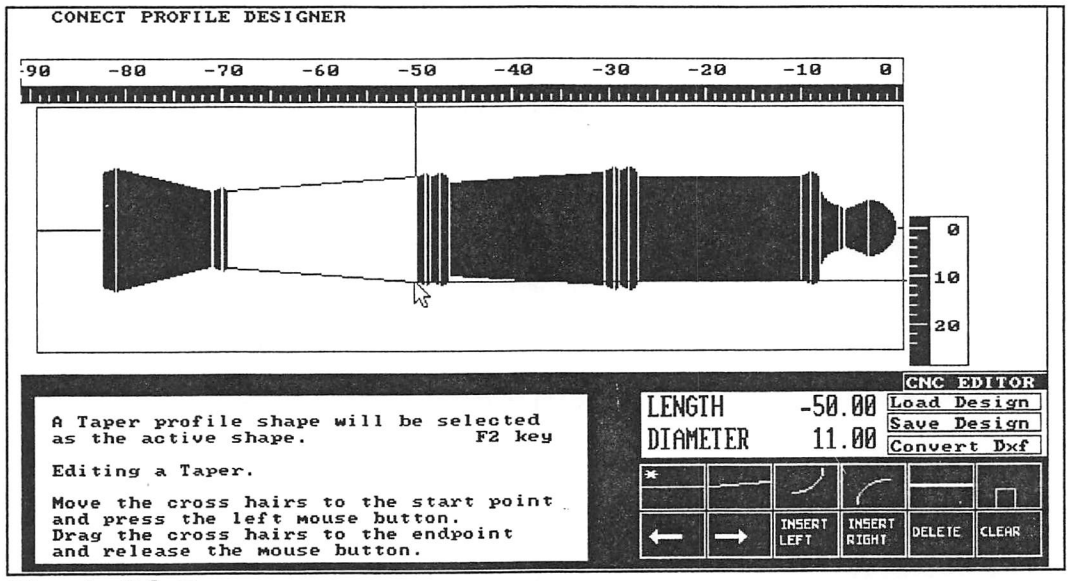
1- Move the cursor to the highlighted shape.
2- Press the left mouse button.
3- Keep the button pressed and drag the cross hairs.
4- Set Diameter and Length and release the mouse button.



Set Diameter and Length

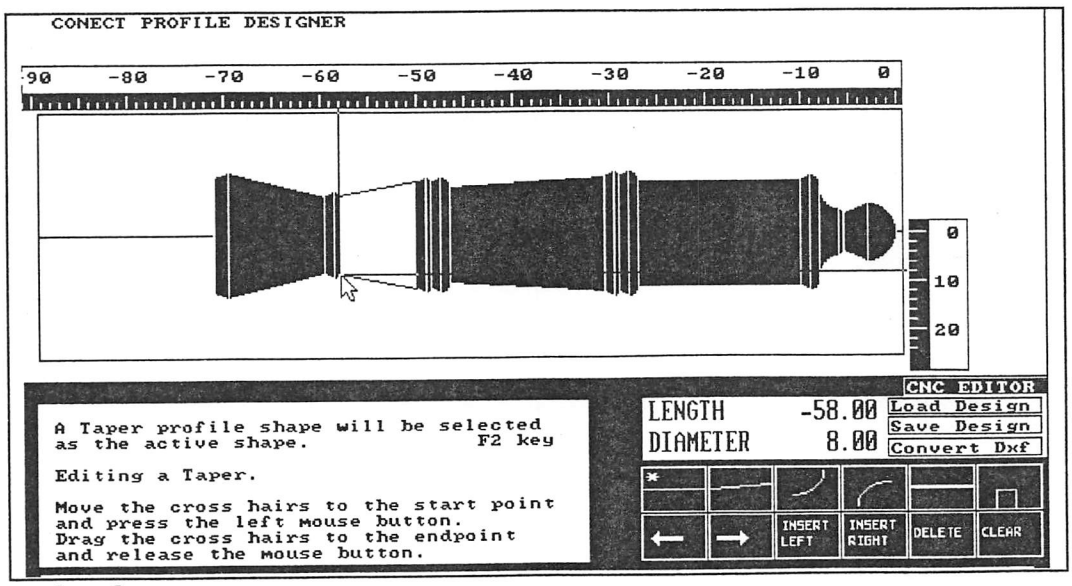
Editing Back Tapers

The angle of a back taper cannot be more than 30 degrees.



Start Of Taper

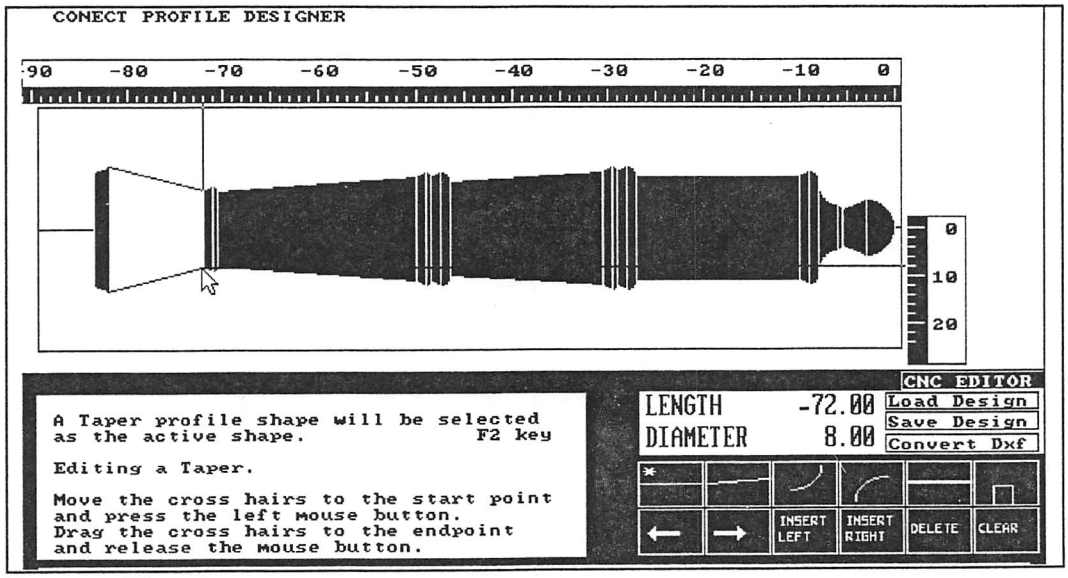
1- Position the crosshairs at Taper start Diameter.
2- Press the left mouse button.
3- Keep the button down and drag the cross hairs.
4- Set Diameter and Length and release the mouse button.



End Of Taper

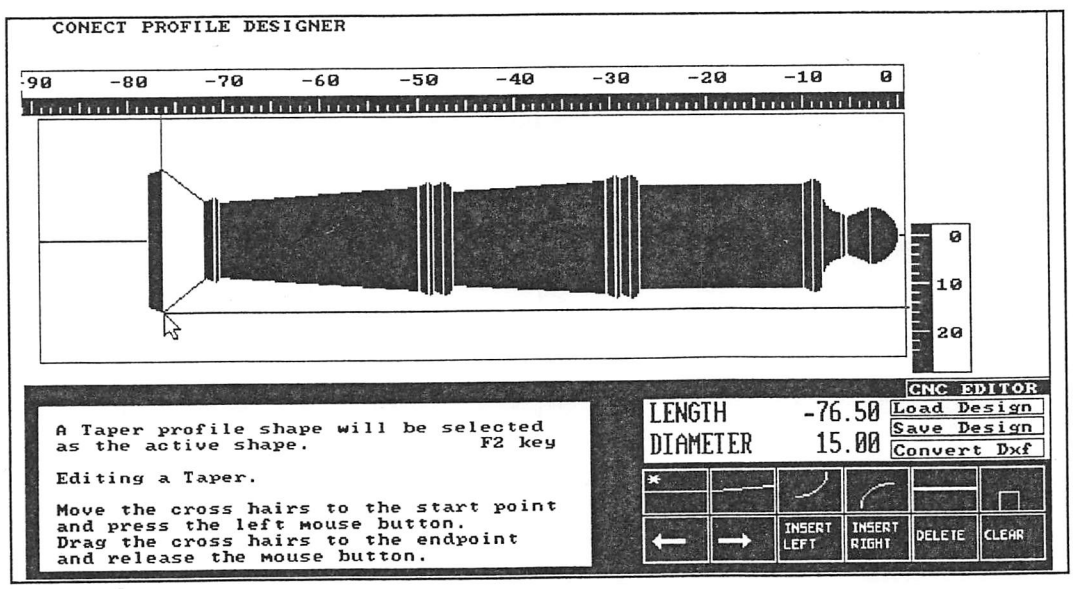
CONECT LATHE PROFILE DESIGNER

Editing Standard Tapers



Taper Start

1- Position the crosshairs at Taper start Diameter.
2- Press the left mouse button.
3- Keep the button down and drag the cross hairs.
4- Set Diameter and Length and release the mouse button.

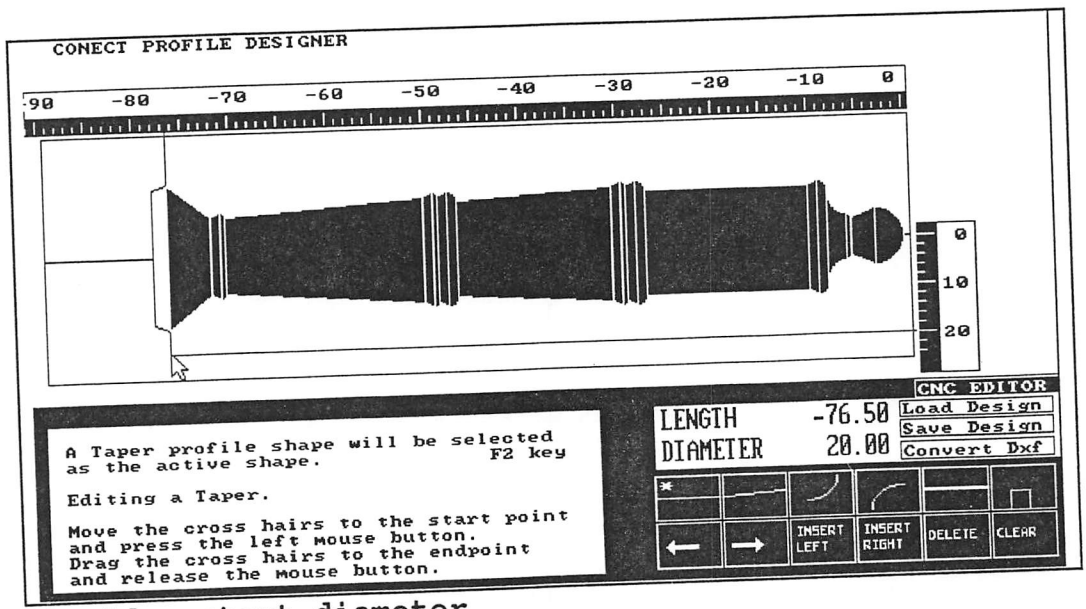


End Of Taper

CONECT LATHE PROFILE DESIGNER

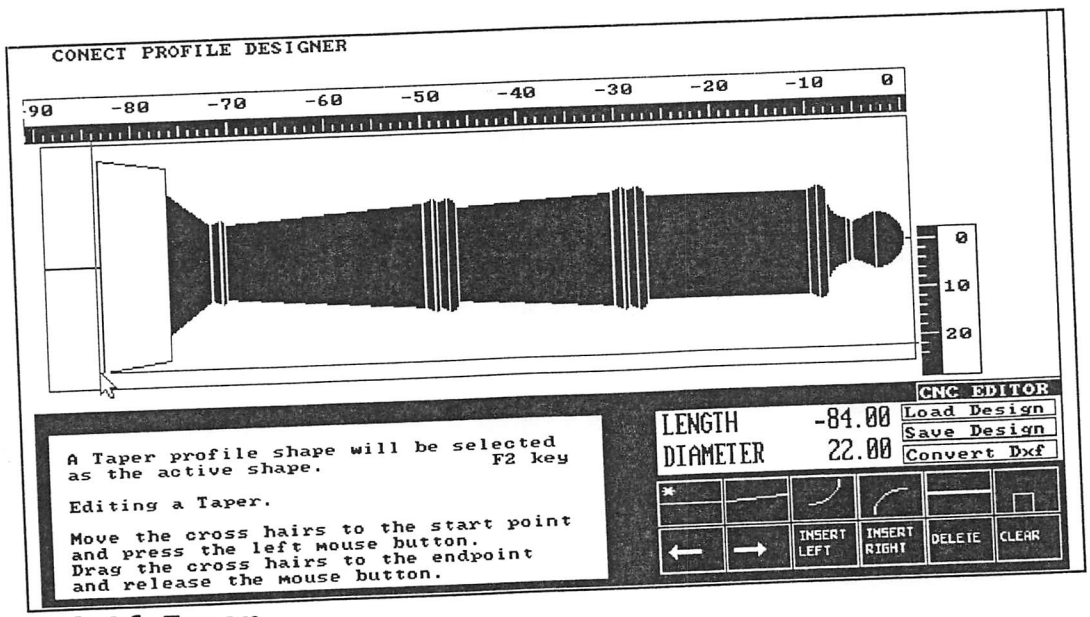
Editing Standard Tapers

This example shows how to add a shoulder diameter.



Shoulder start diameter

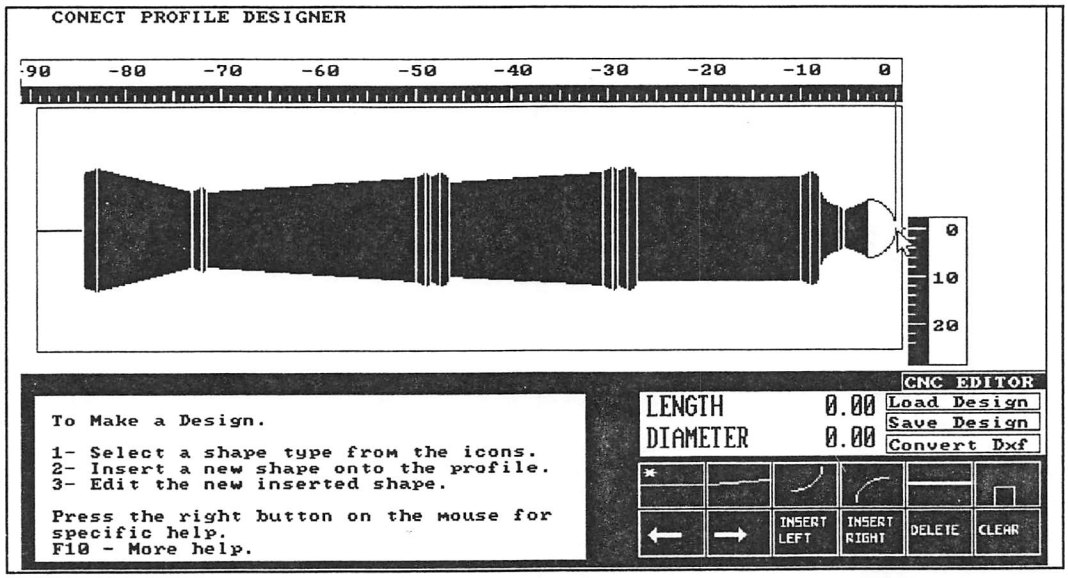
1- Position the crosshairs at Shoulder start Diameter.
2- Press the left mouse button.
3- Keep the button down and drag the cross hairs.
4- Set Diameter and Length and release the mouse button.



End Of Taper

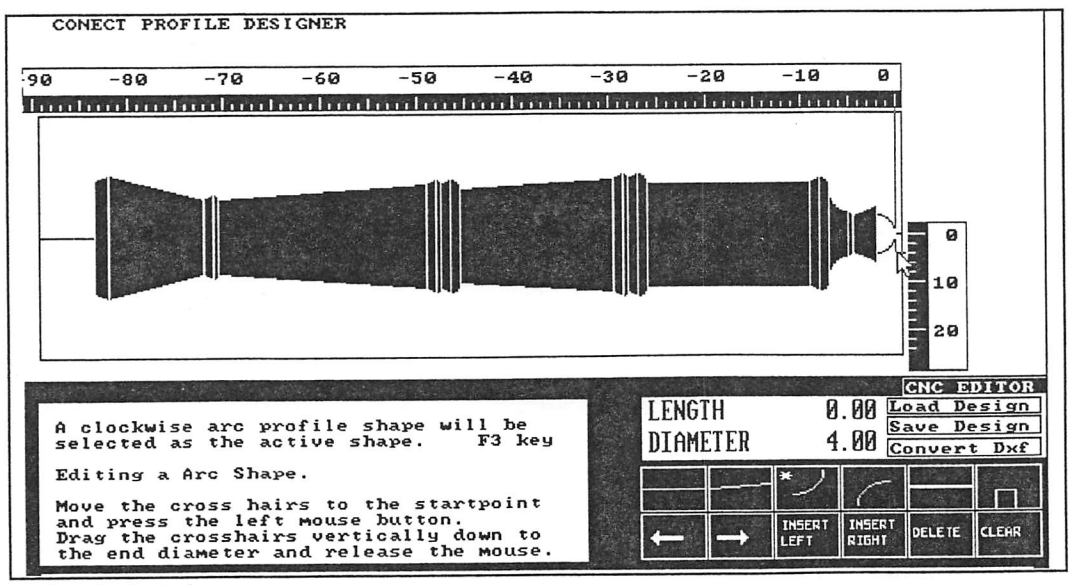
CONECT LATHE PROFILE DESIGNER

Editing Clockwise Arcs



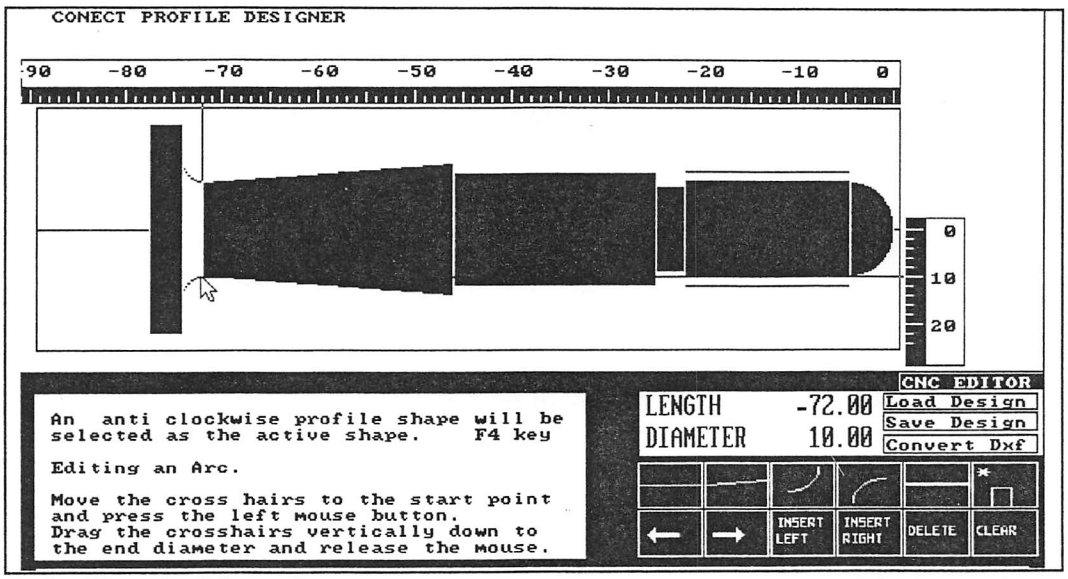
Set Start Diameter

1- Position the crosshairs at the start Diameter.
2- Press the left mouse button.
3- Keep the button down and drag the cross hairs vertically.
4- Set Diameter and release the mouse button.



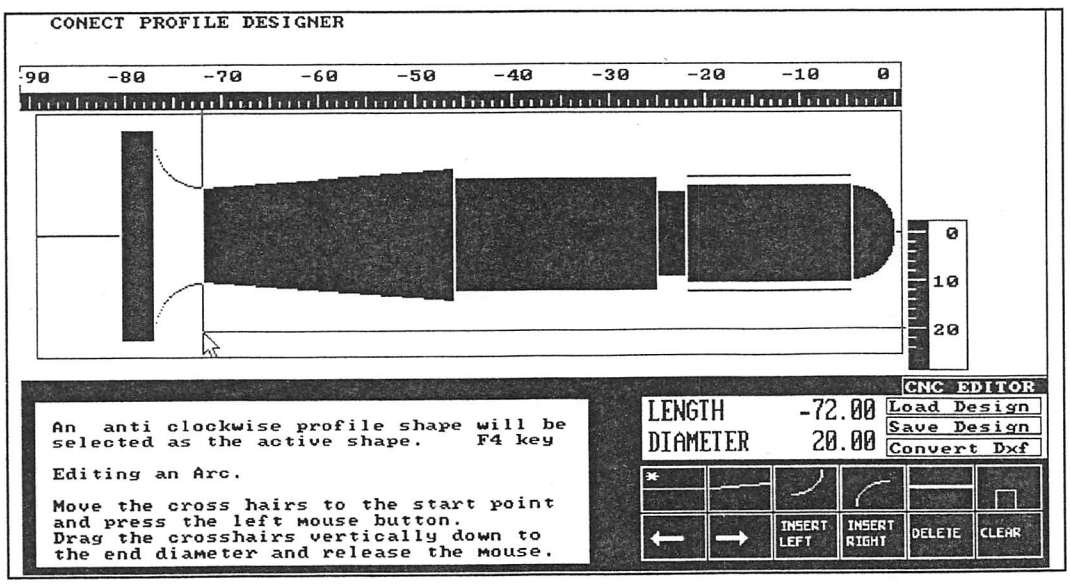
Set End Diameter

Editing Anti-Clockwise Arcs



Set Start Diameter

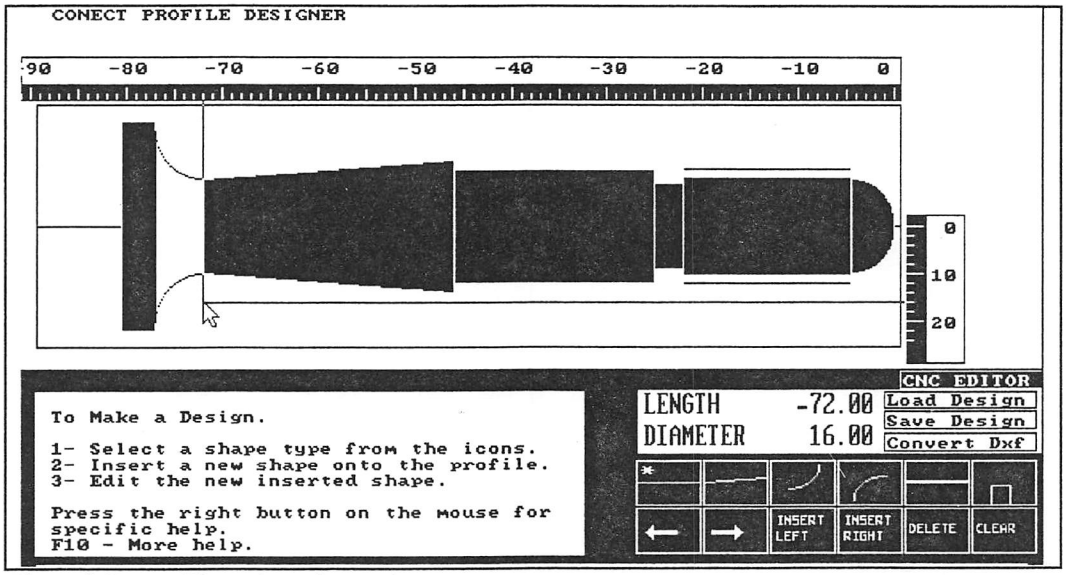
1- Position the crosshairs at the start Diameter.
2- Press the left mouse button.
3- Keep the button down and drag the cross hairs vertically.
4- Set Diameter and release the mouse button.



Set End Diameter

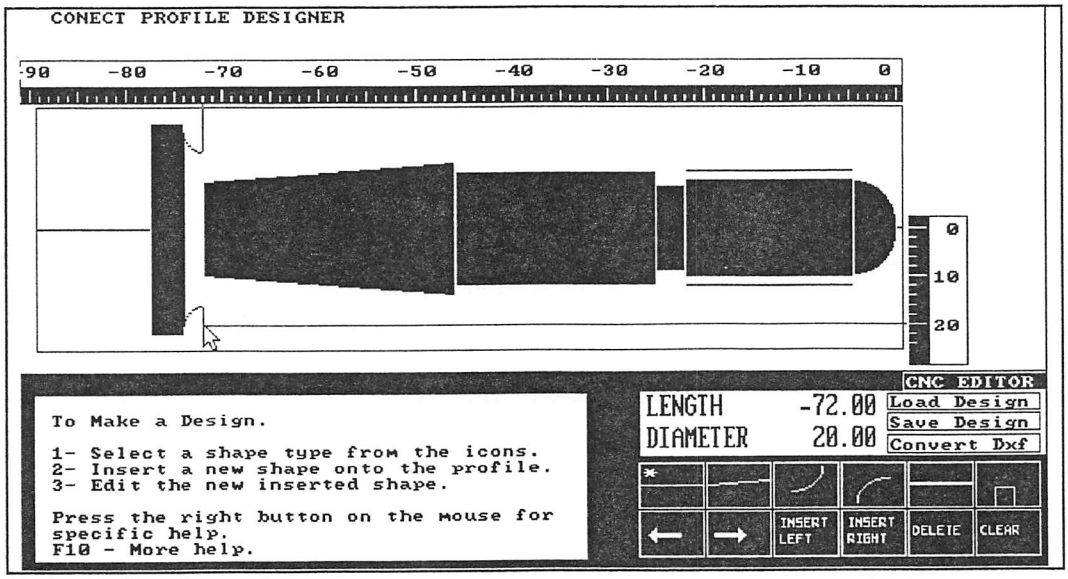
CONECT LATHE PROFILE DESIGNER

Editing Arcs With A Shoulder



Shoulder Start Diameter

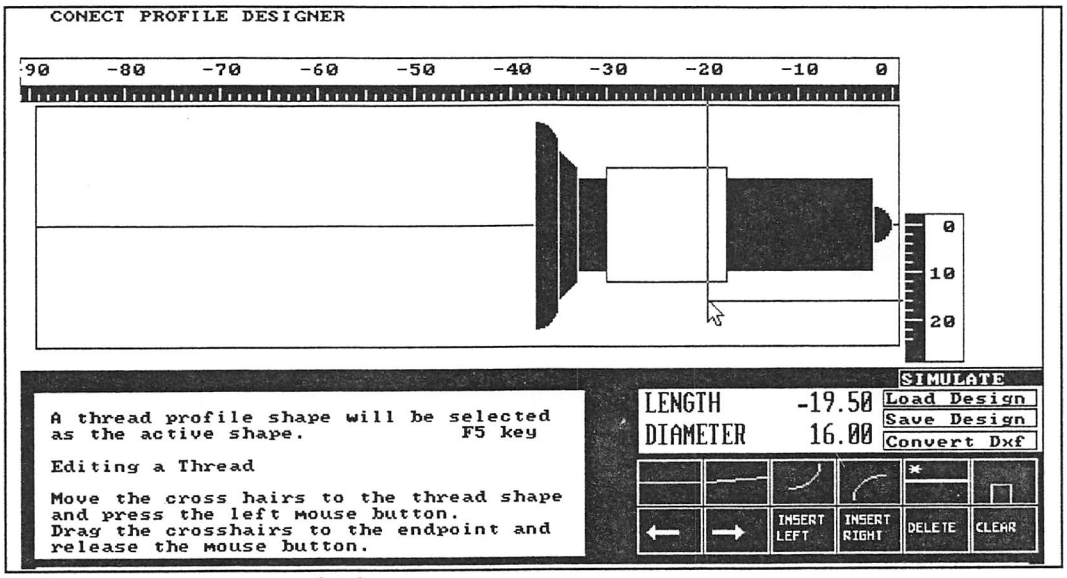
1- Position the crosshairs at the shoulder start Diameter.
2- Press the left mouse button.
3- Keep the button down and drag the cross hairs vertically.
4- Set Diameter and release the mouse button.



Set End Diameter

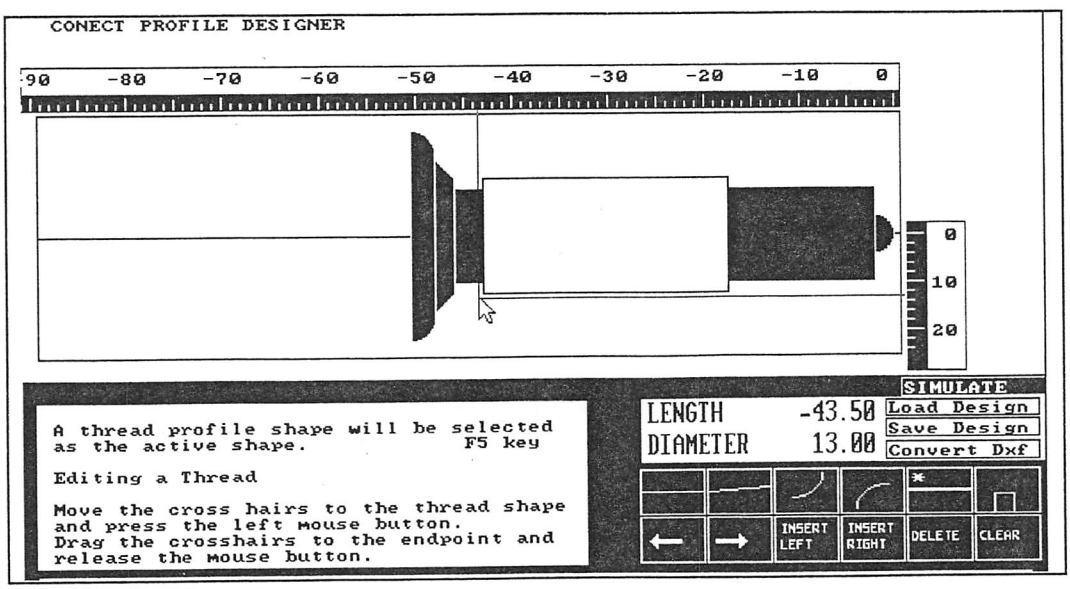
CONECT LATHE PROFILE DESIGNER

Editing Threads



Select Start Position

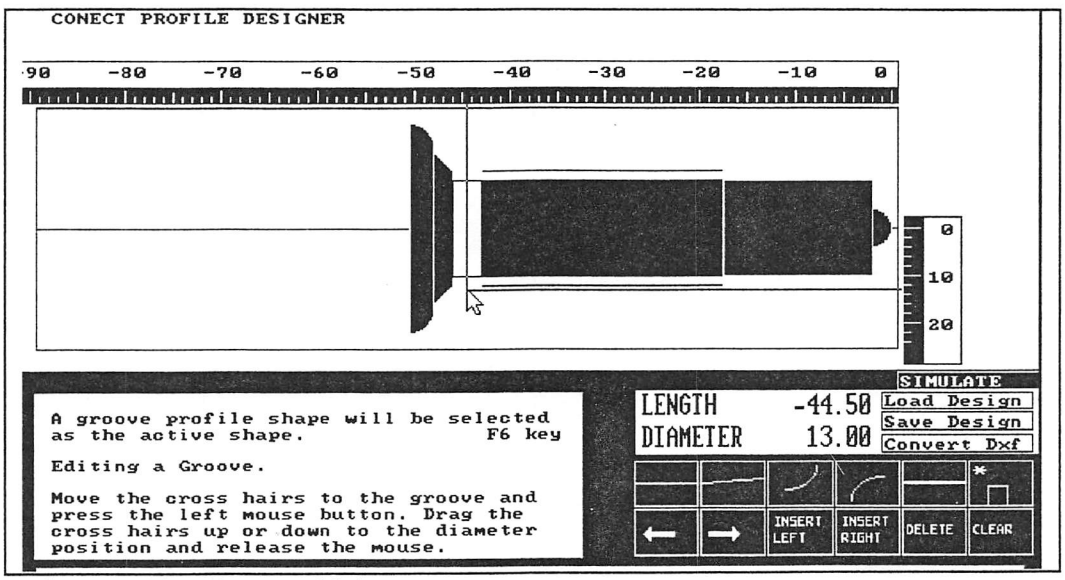
1- Move the cursor to the highlighted shape.
2- Press the left mouse button.
3- Keep the button down and drag the cross hairs.
4- Set Diameter and Length and release the mouse button.



Set End Diameter

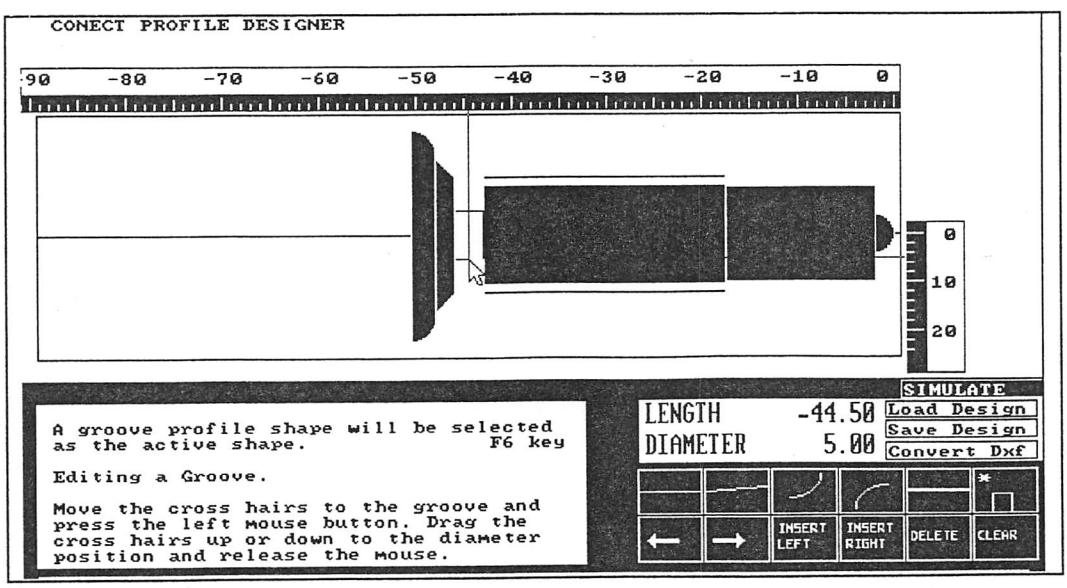
CONECT LATHE PROFILE DESIGNER

Editing Grooves



Select Start Position

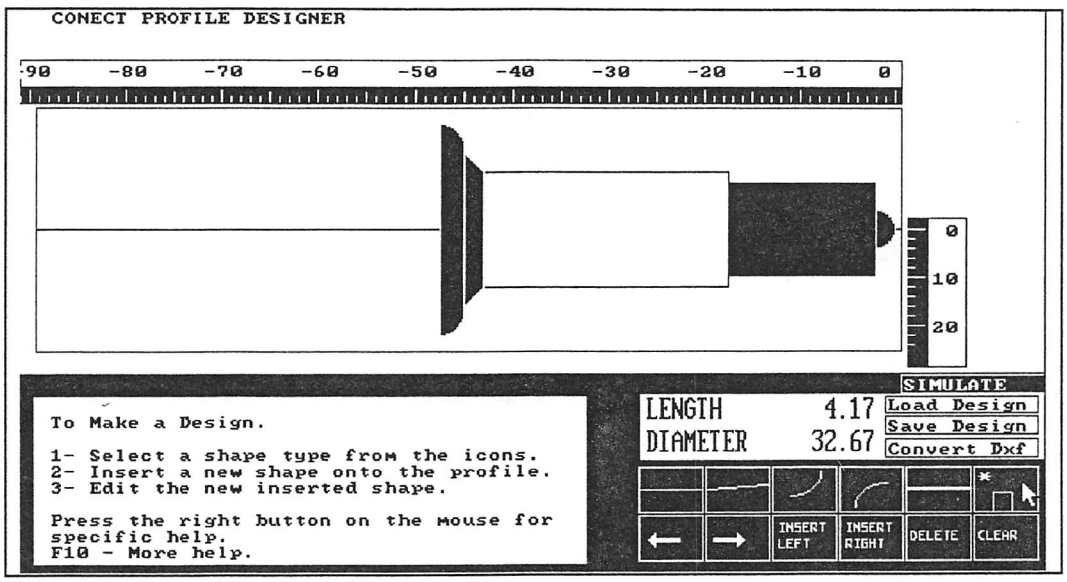
1- Move the cursor to the highlighted shape.
2- Press the left mouse button.
3- Keep the button down and drag the cross hairs vertically.
4- Set Diameter and release the mouse button.



Set The Diameter

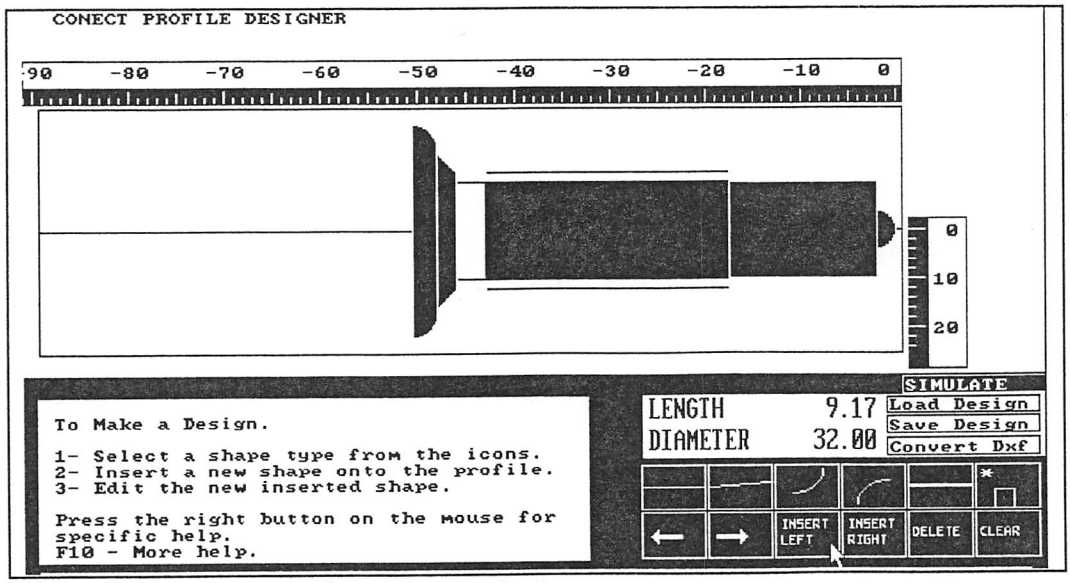
CONECT LATHE PROFILE DESIGNER

Insert A Threading Groove



Select The Groove Icon

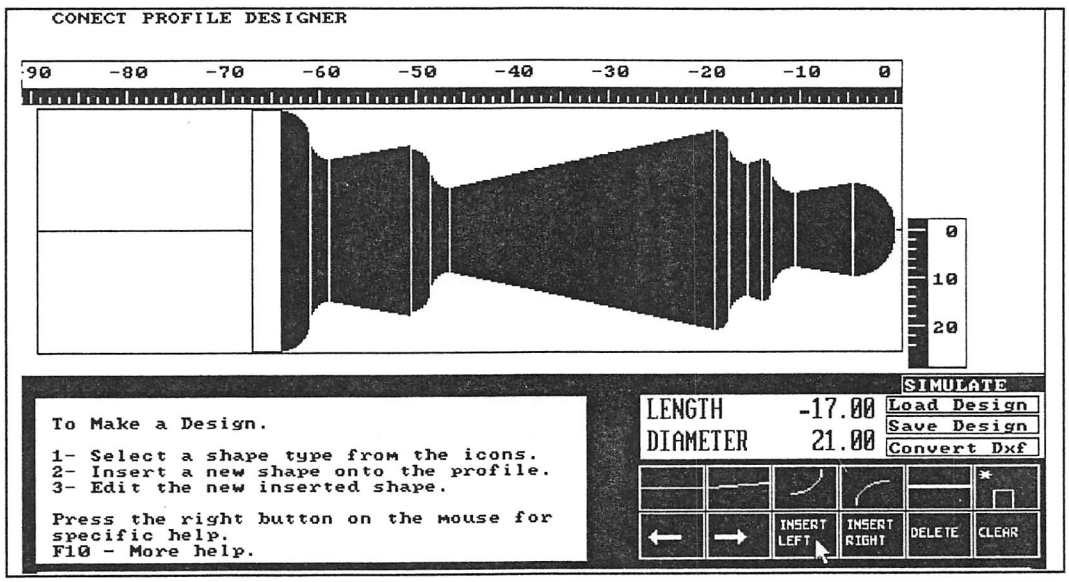
1- Highlight the groove icon.
2- Set the thread as the current profile shape.
3- Press the 'Insert Left' icon.
4- The groove Diameter will be set from Threading Tables.



Click On The 'Left Insert' Icon.

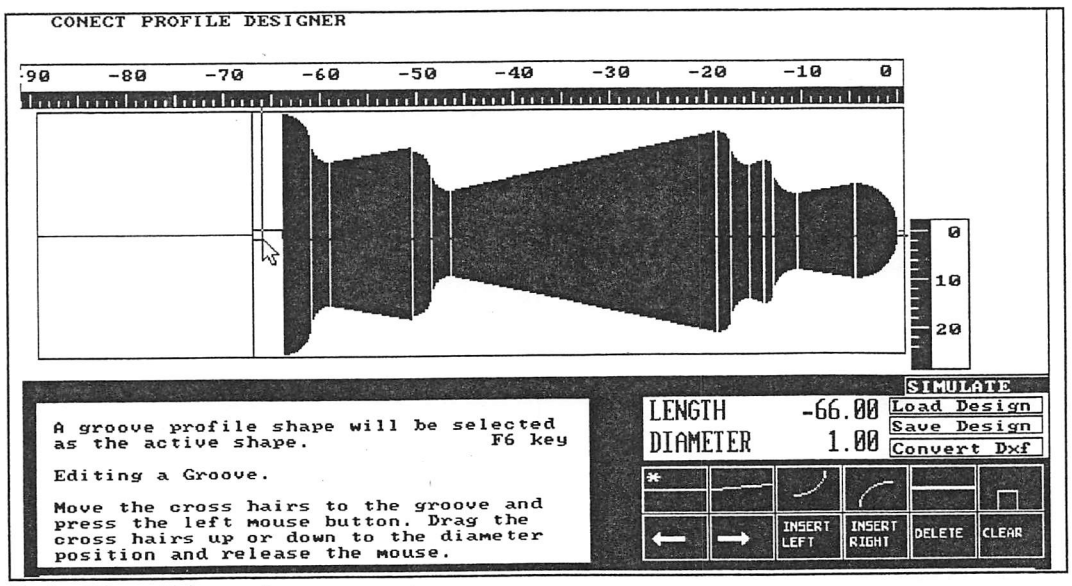
CONECT LATHE PROFILE DESIGNER

Editing A Parting Off Groove



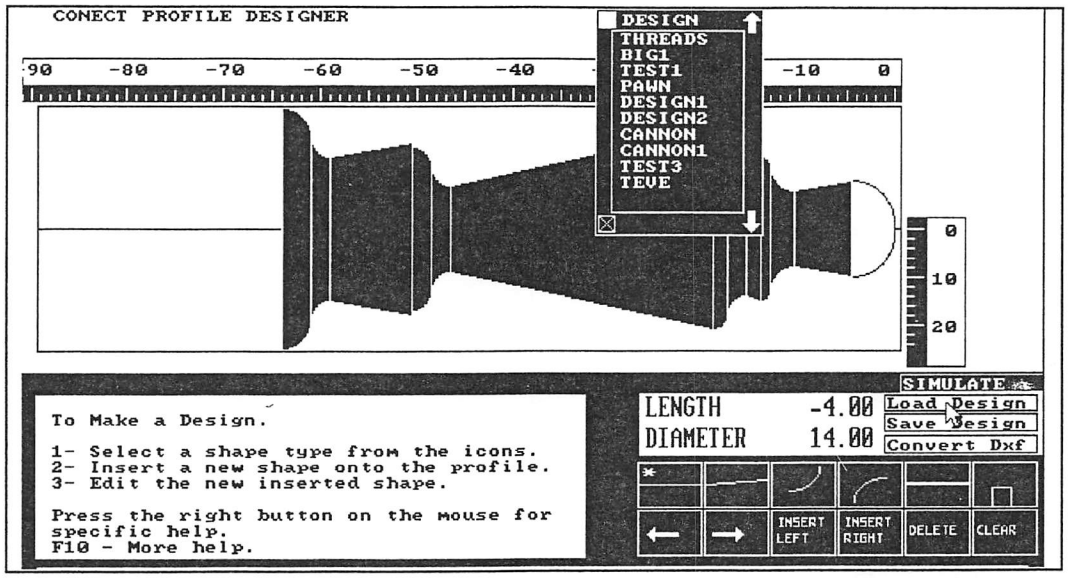
Insert A Groove

1- Highlight the groove icon.
2- Set the last profile shape as the current profile shape.
3- Press the 'Insert Left' icon.
4- Set the groove Diameter.



Set The Diameter

Loading and Saving Designs



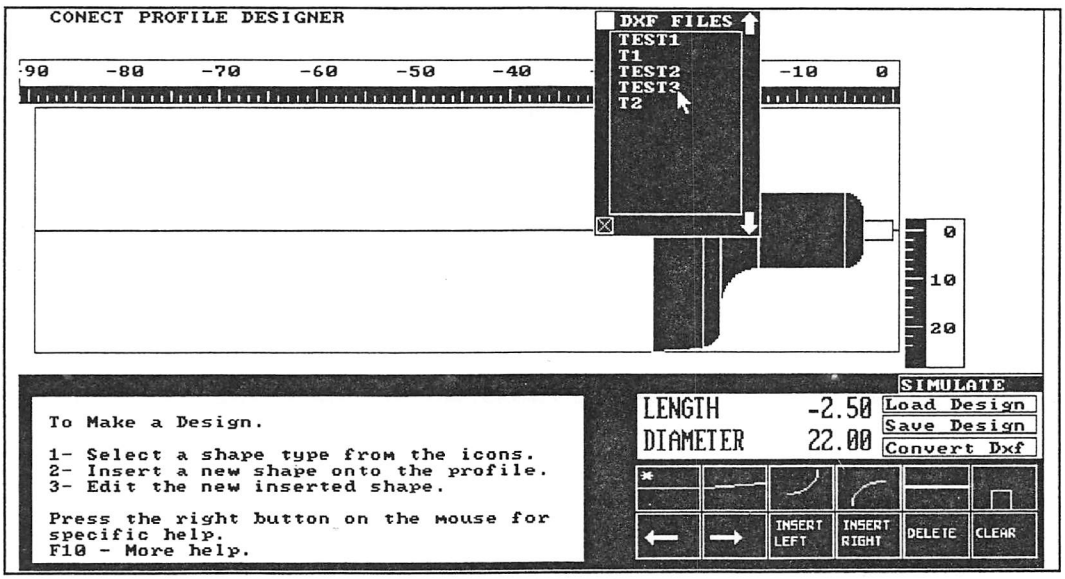
Loading a Design

Click on the 'Load Design' menu or press the 'L' key on the qwerty keyboard.
Select a file using the 'Up and Down' arrow keys or the mouse.
Press the Return key or click on the left mouse button to load.
Press the Esc key or click on the bottom left hand side of the pull down menu to clear.
The design will be displayed on the screen.

Saving a Design

Click on the 'Save Design' menu or press the 'S' key on the qwerty keyboard.
Type in the filename and press the return key. If the file already exists you will prompted to 'Overwrite y/n?'. Press the 'Y' key to overwrite or the 'N' key to leave.

Converting Dxf Files



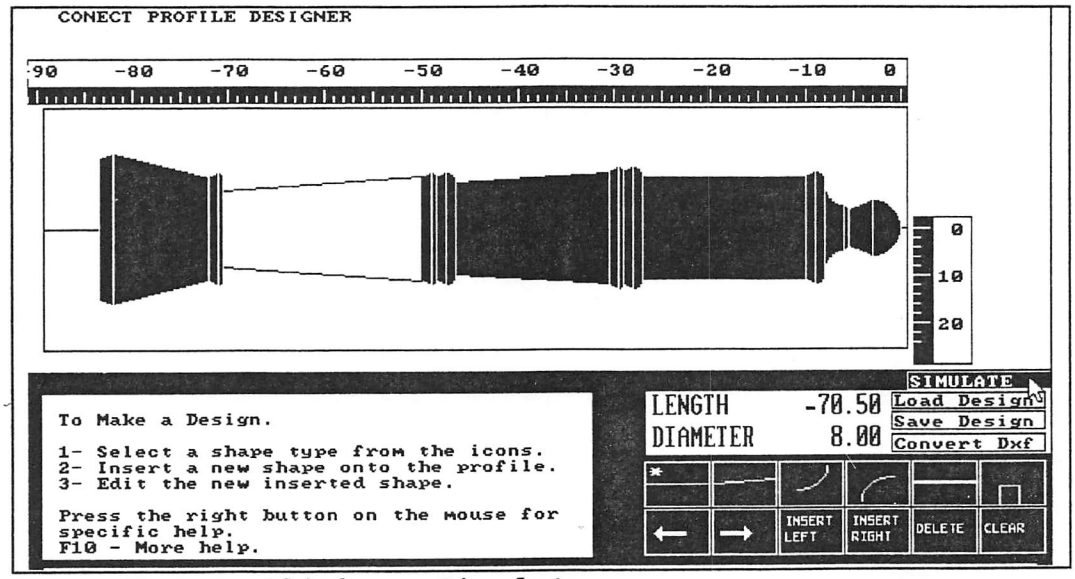
Click on the 'Convert Dxf' menu or press the 'D' key on the qwerty keyboard.
Select the filename with the 'Up and Down' arrow keys or by pointing with the mouse.
Press return or click on the left mouse button to confirm.
Press the Esc key or click on the bottom left hand side of the pull down menu to clear.

Refer to the 'DXF PROFILES' section on how to define dxf profiles and link them to the Conect Lathe software.

If you read in Dxf profiles from the Main Screen they will also be converted into design shapes.

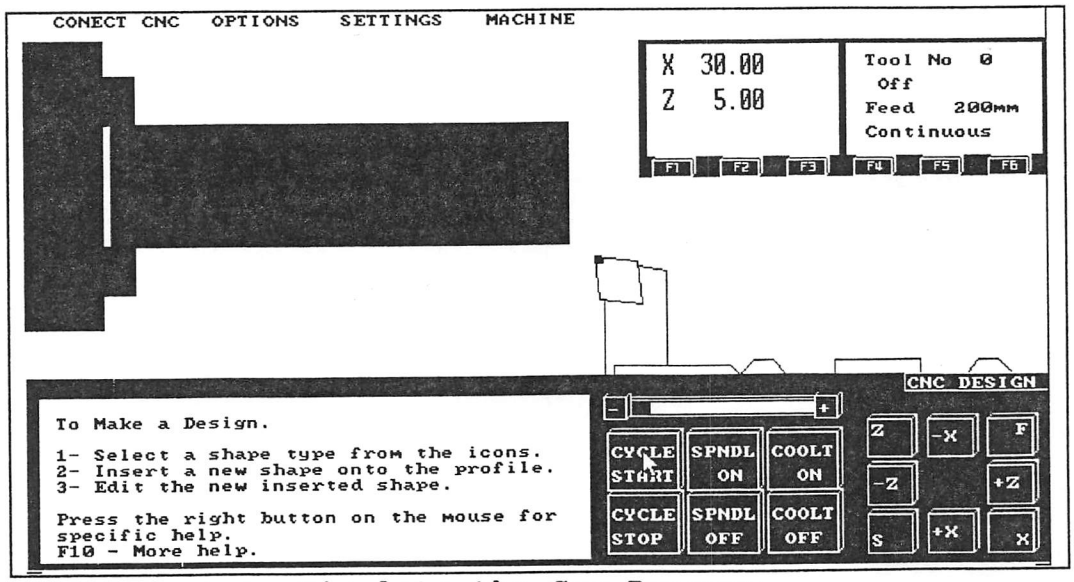
CONECT LATHE PROFILE DESIGNER

Creating Cnc Programs



Press Esc or Click on Simulate

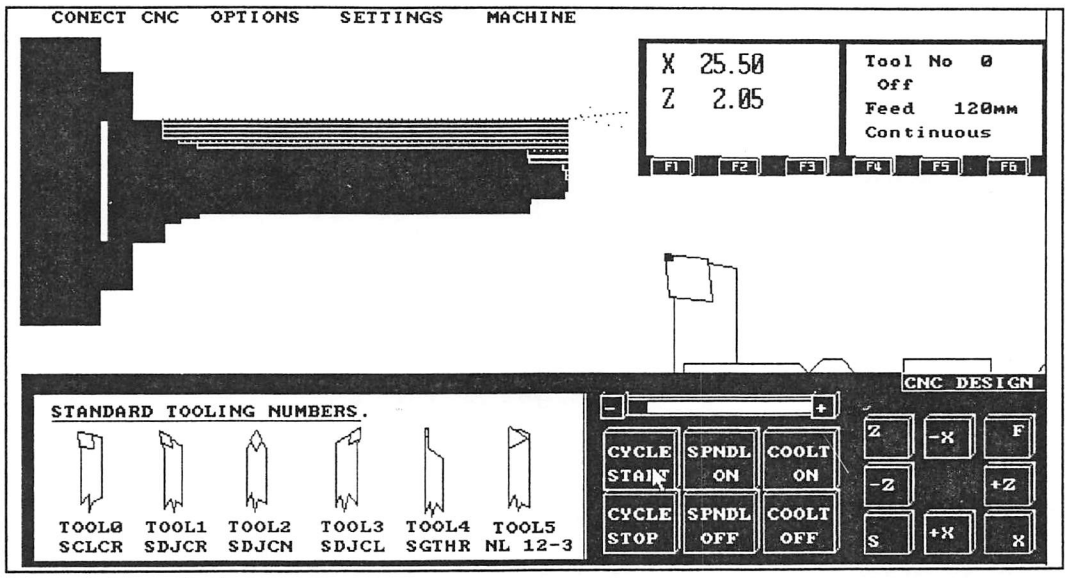
To generate a CNC program form your Design you can either press the Esc key or click on the 'Simulate Menu'. The program will be written in a few seconds. All machining and tooling information will be automatically inserted into the program.



Cycle Start to Simulate the Cnc Program

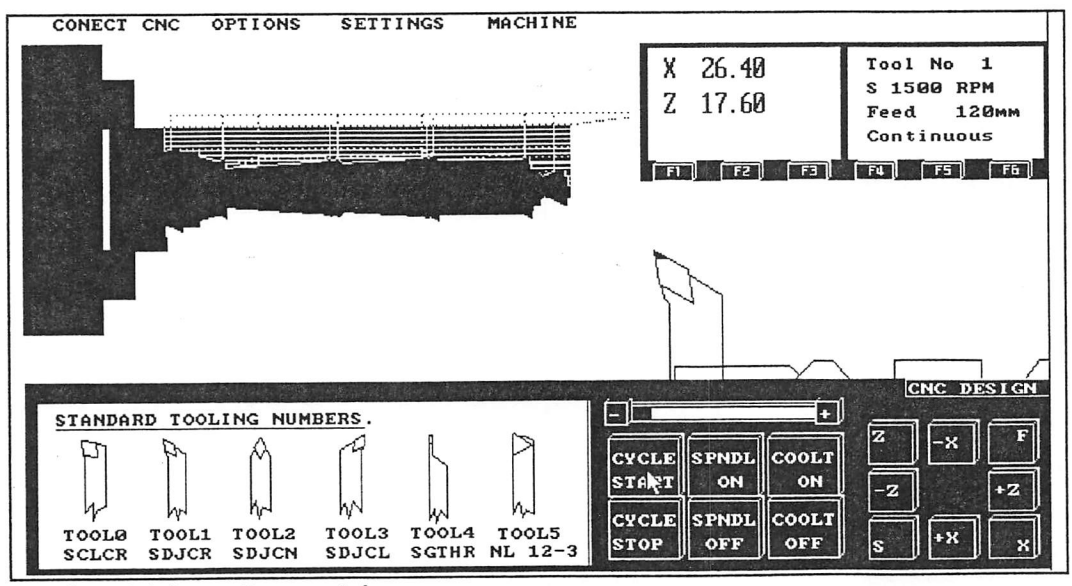
CONECT LATHE PROFILE DESIGNER

Creating Cnc Programs (Cannon)



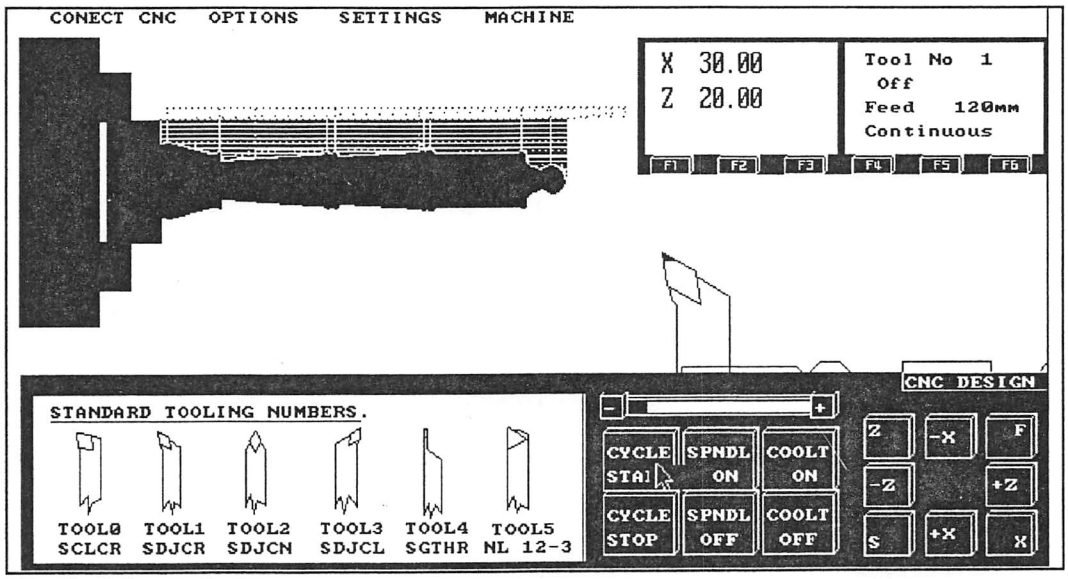
Roughing Cuts

1- All roughing cuts apart from backcutting will take place.
2- The roughing tool will move to its Toolchange position.
3- The finishing tool will be selected.
4- Any backcutting will take place.



Backcutting Operation

Creating Cnc Programs (Cannon)

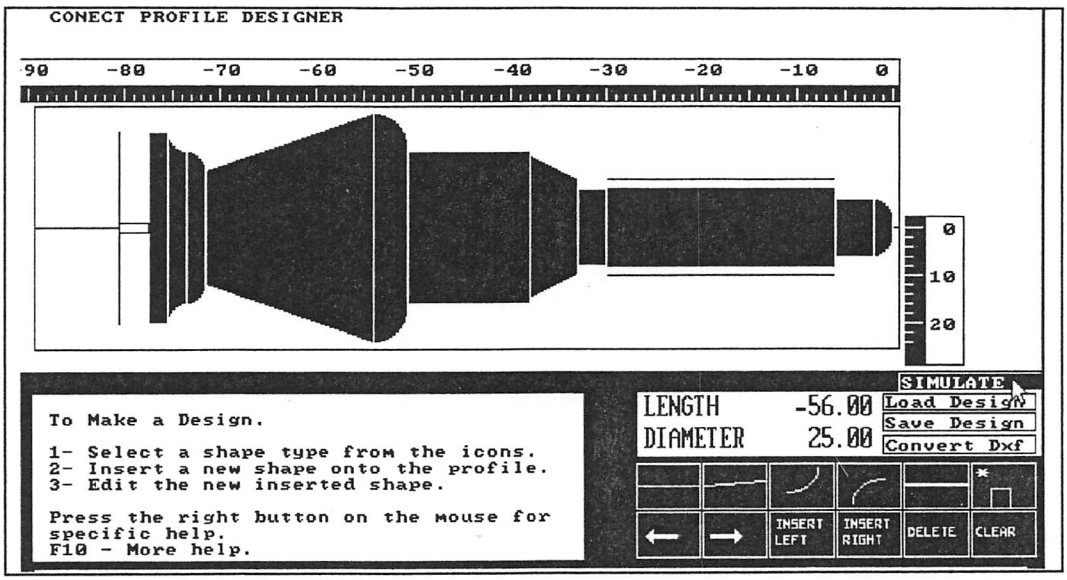


Finishing Cuts

After all the roughing cuts the finishing tool will be used to take a finishing cut along the profile.

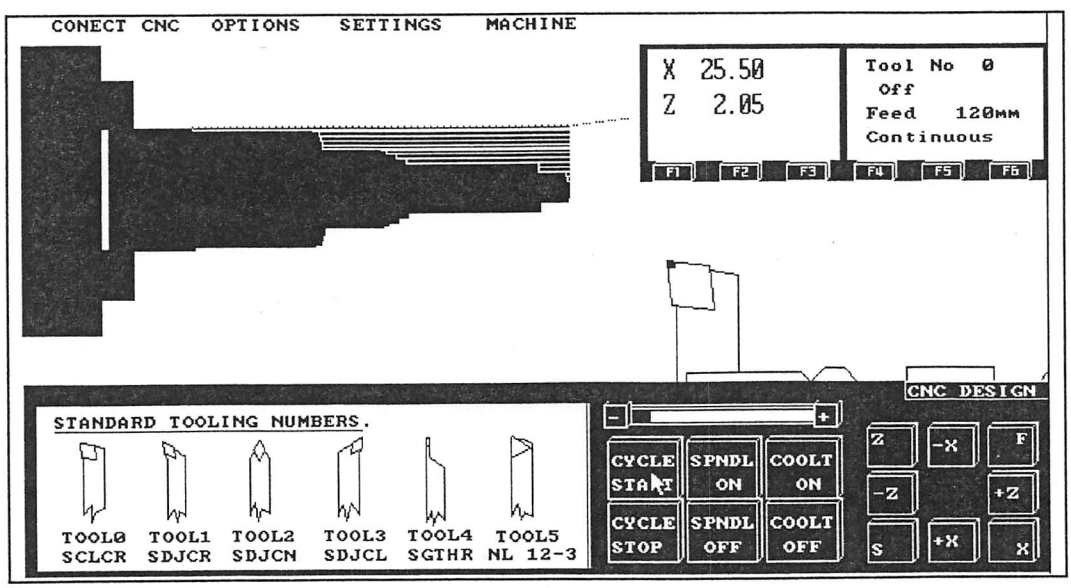
CONECT LATHE PROFILE DESIGNER

Creating Cnc Programs (Turn3)



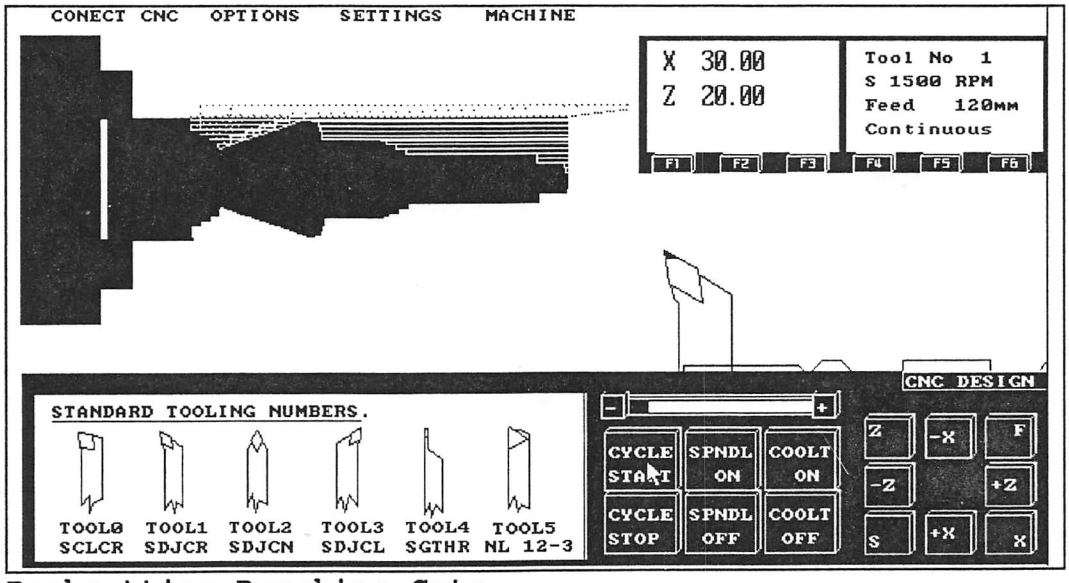
Turn3 Design Profile

This example includes all the machining operations that are available in the Profile Designer.



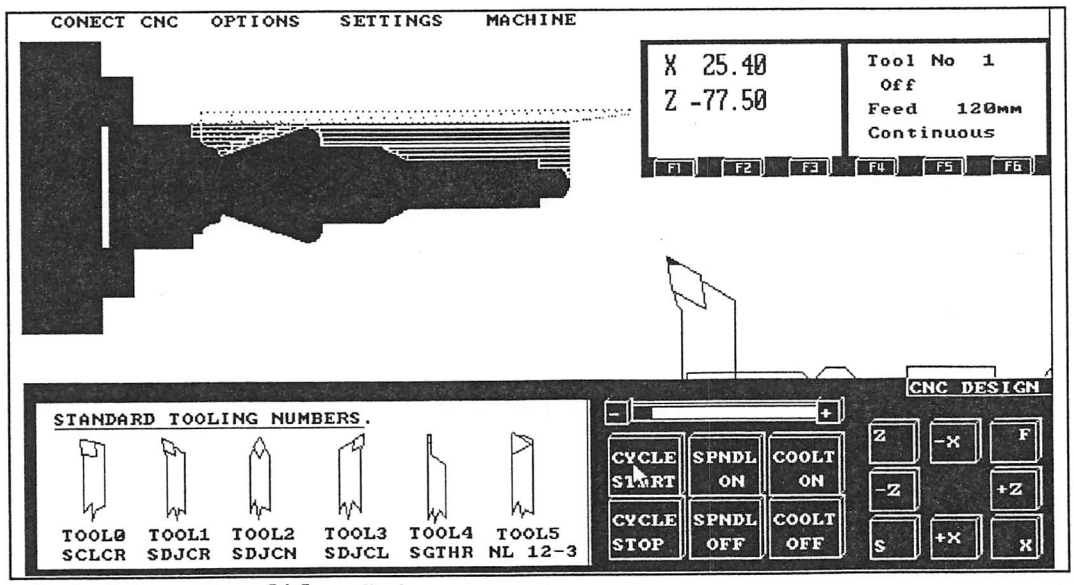
Roughing Cuts

Creating Cnc Programs (Turn3)



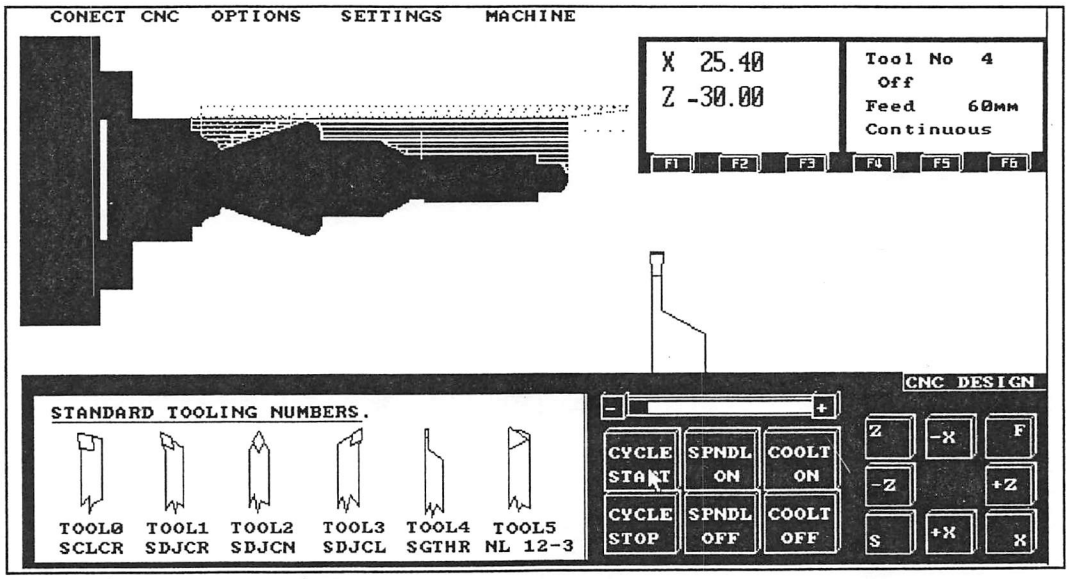
Backcutting Roughing Cuts

This example includes all the machining operations that are available in the Profile Designer.



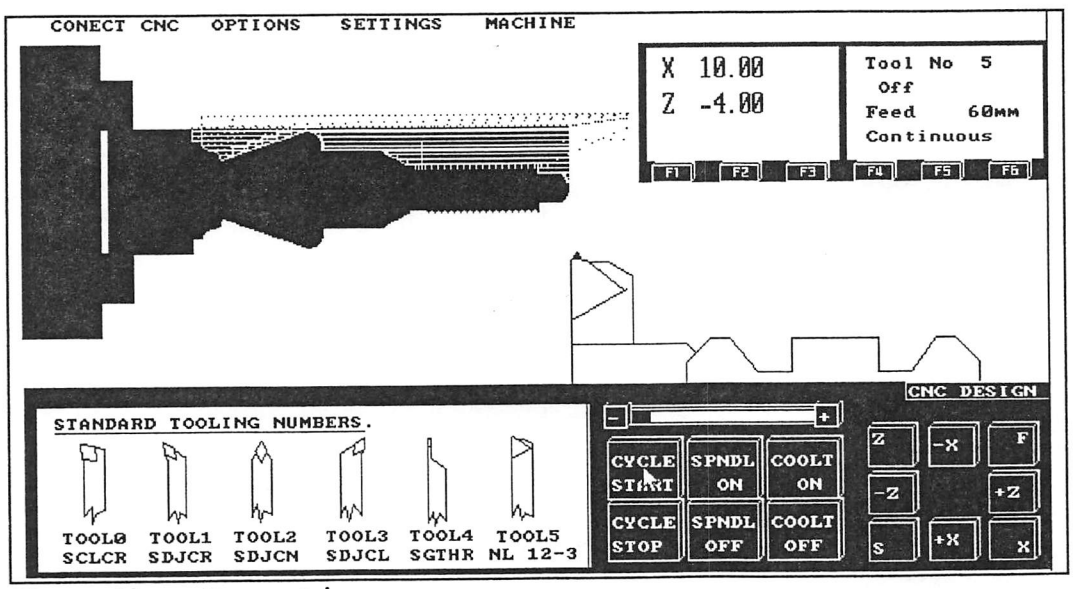
Finishing Profile Cuts

Creating Cnc Programs (Turn3)



Grooving Operations

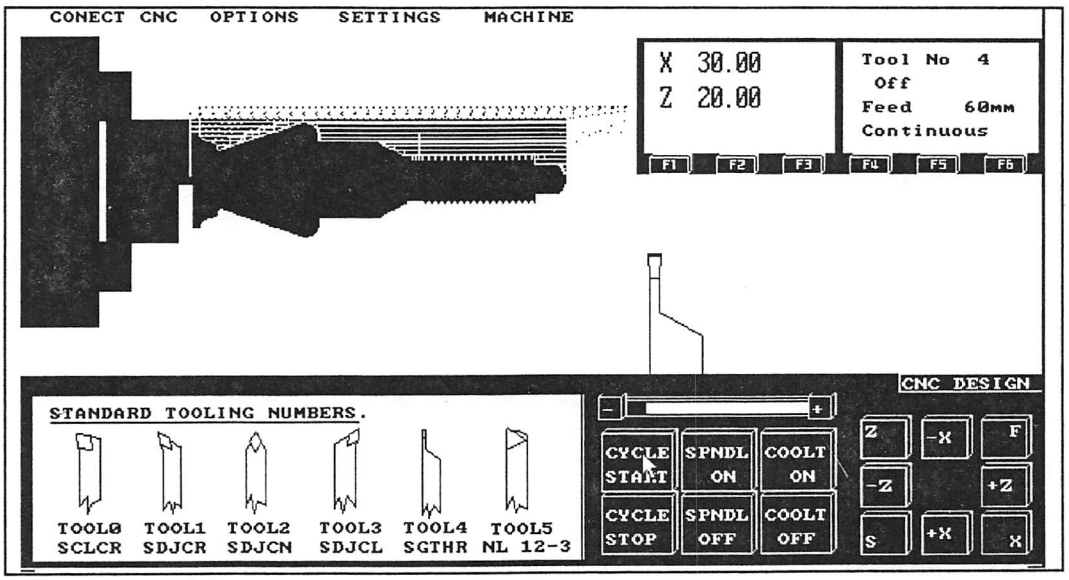
This example includes all the machining operations that are available in the Profile Designer.



Threading Operations

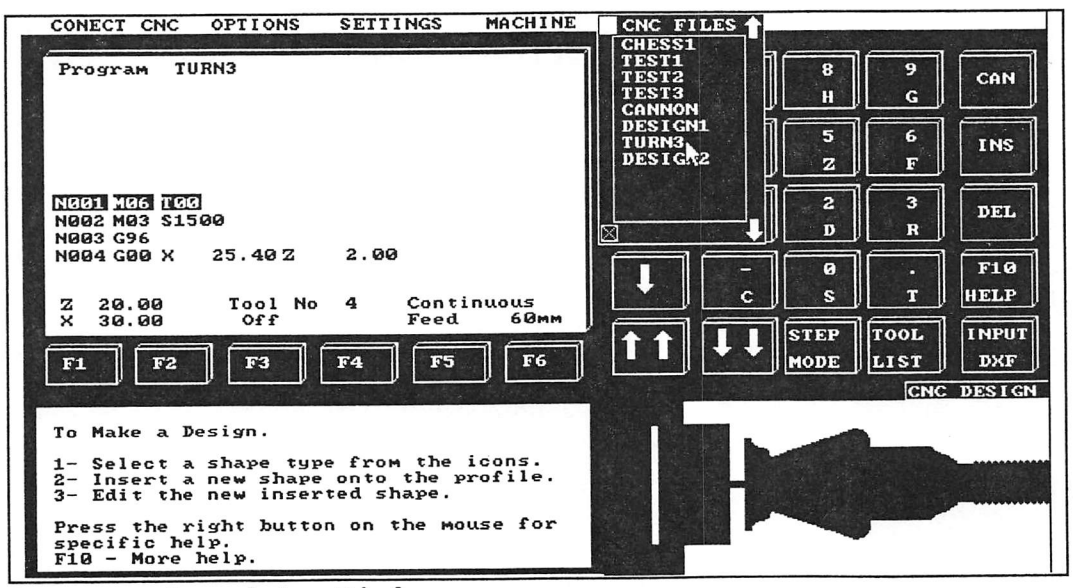
CONECT LATHE PROFILE DESIGNER

Creating Cnc Programs (Turn3)



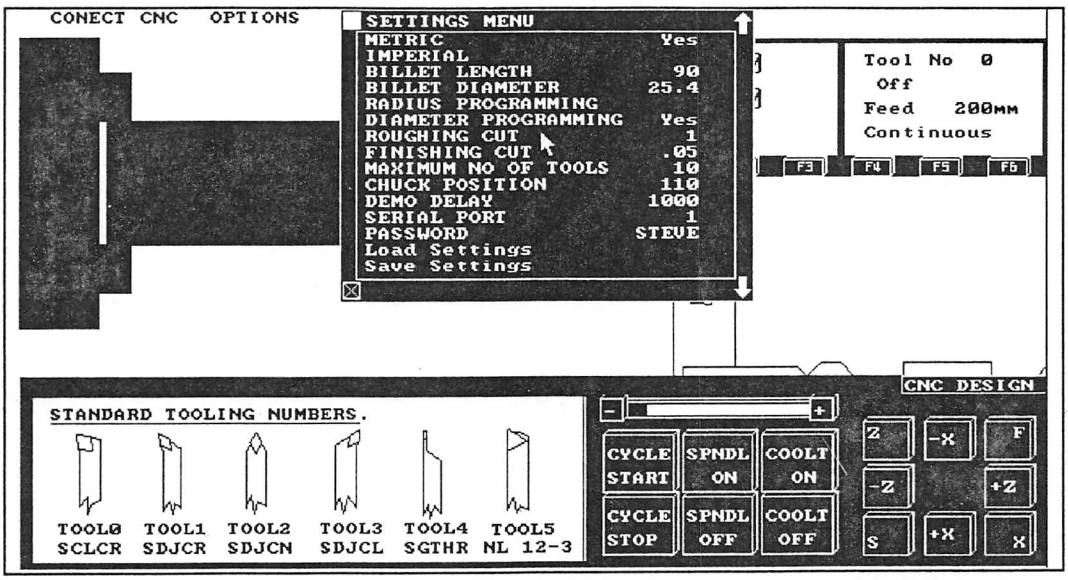
Parting Off

All Cnc programs that are generated from the Profile Designer can be saved to Disk as a Cnc program. For machining purposes simply load the Cnc program disk if you want to produce the part on the machine or simply to simulate on the screen.



Program Saved to Disk

Settings Menu



Before you use the Lathe Profile Designer it is important that the Settings are correct.

Units

Set the units of measure to Imperial or Metric.

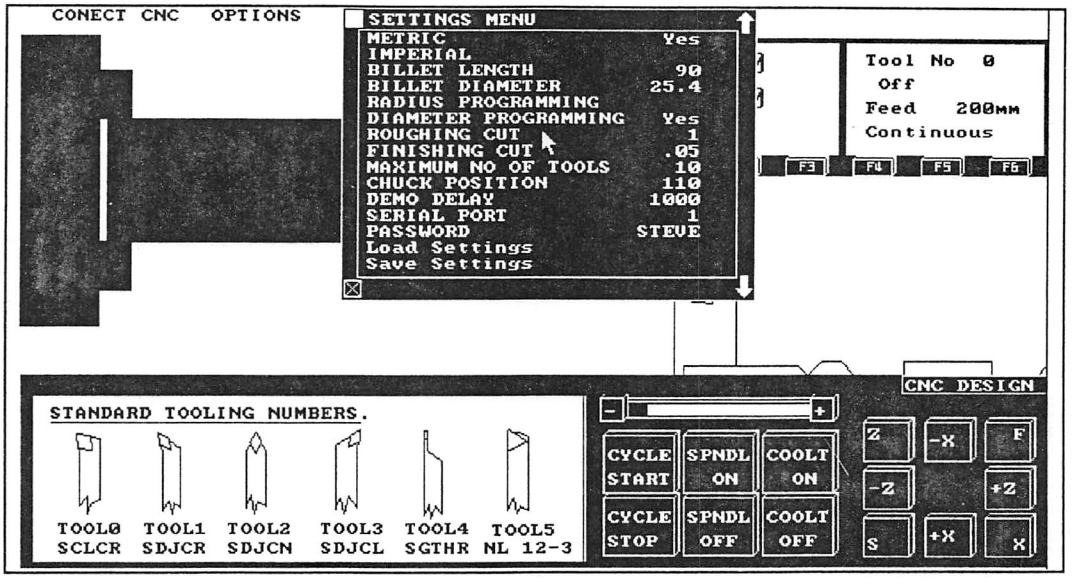
Billet Length and Billet Diameter

This should be set to the distance from the chuck jaw to the end of the bar. The profile designer will take the length and diameter setting and scale the rulers for the largest possible screen representation.

Loading Designs from Disk that are outside the billet dimensions will be accepted but the generation of the Cnc program will automatically calculate what is possible to machine without going into the chuck.

If you try to insert new profile shapes or edit existing profile shapes that go beyond the billet dimensions, they will not be accepted.

Settings Menu



Before you use the Lathe Profile Designer it is important that the Settings are correct.

Roughing Cut

A depth for the roughing cuts.

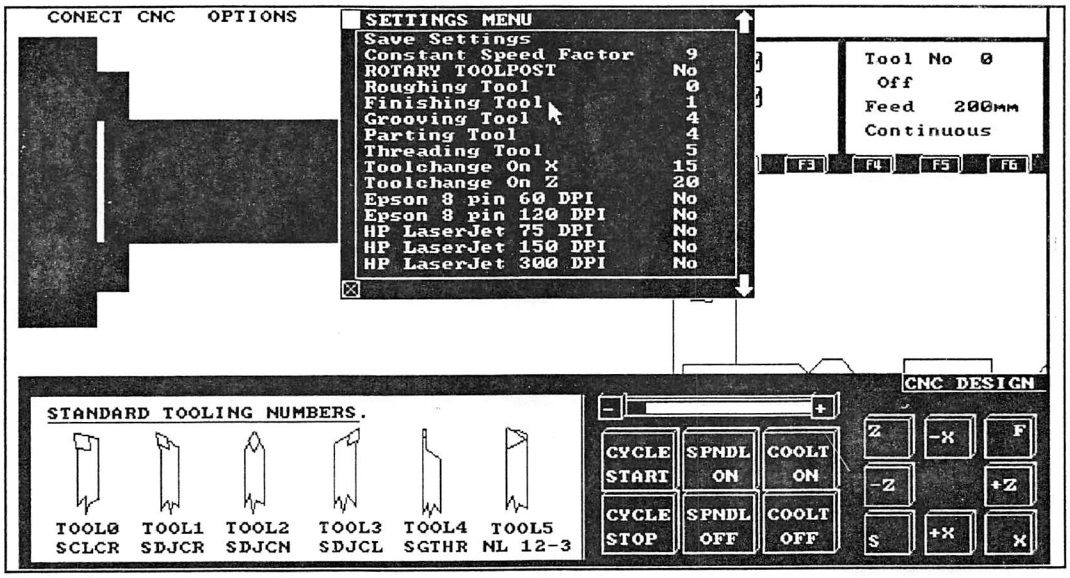
Finishing Cut

Amount of material to leave for the finishing profile cut.

Load/Save Settings

You can have any number of different settings files. For example for different material types.

Settings Menu



Settings Page 2

Before you use the Lathe Profile Designer it is important that the Settings are correct.

Constant Speed Factor (Recommend 9)

Select a number from 1 to 9 depending on the material.

Roughing Tool

This tool is used for the roughing cuts. The default Tool Number is 0.

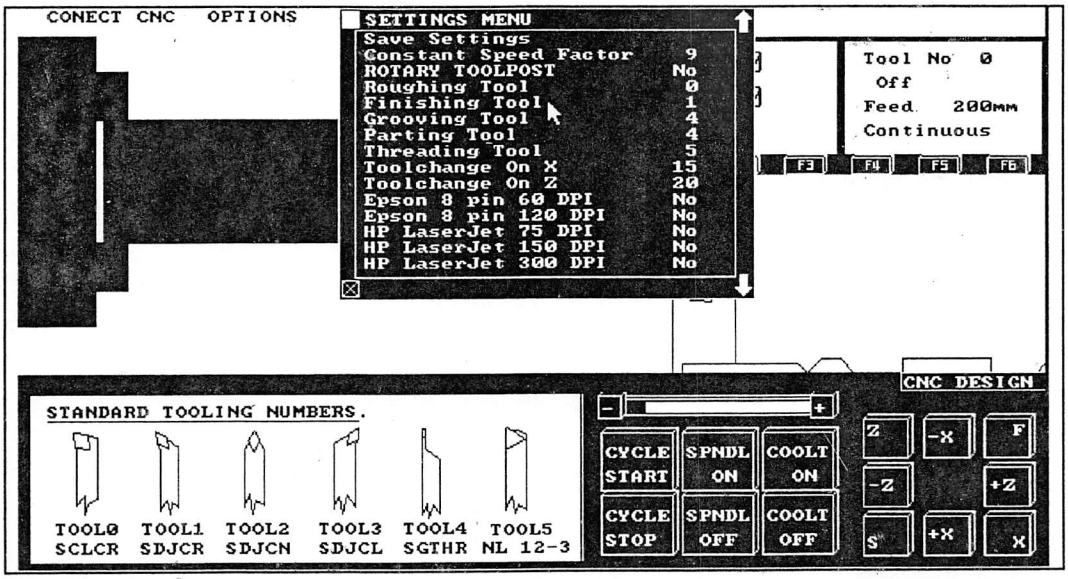
Finishing Tool

Used for the backcutting roughing cuts and the finishing profile cuts. The Default Tool Number is 1.

Grooving/Parting Tool

Used for all ther grooves and parting off. The default Tool Number is 4.

Settings Menu



Settings Page 2

Threading Tool

Used for all threading operations. The default Tool Number is no 5.

ToolChange On Z/ToolChange On X

These values are used for every toolchange.
When the backcutting roughing cuts are being machined the ToolChange On X value is used for the Tool to moveto before it goes to the backcutting position.
Set the ToolChange On X value to just above the radius of the bar.